15th Australasian Fluid Mechanics Conference
The University of Sydney, Sydney, Australia
13-17 December 2004

A Numerical Study Of A Circular Cylinder In The Wake Of An Airfoil

C. Y. Zhou[1], C. W. Sun[1], Y. Zhou[2], L. Huang[2]
[1]Shenzhen Graduate School, Harbin Institute of Technology
Xili Shenzhen University Town, Shenzhen 518055, China
[2]Department of Mechanical Engineering, The Hong Kong Polytechnic University
Hung Hom, Kowloon, Hong Kong, China

Abstract
A circular cylinder in the near wake of NACA 4412 airfoil in a cross flow is numerically studied using finite volume method for Reynolds Number Re=200 based on the cylinder diameter. The effects of the attack angle of the airfoil, the longitudinal and lateral spacing between the airfoil and the cylinder on the unsteady loading, vortex shedding frequency and vortex patterns of the cylinder are examined.

Introduction
A circular cylinder in an airfoil wake in a cross flow arises in many engineering applications, for example, in cooling fan applications where motor is supported by simple struts made of circular cylinders. The rotor wake impinges on the struts and the struts themselves experience vortex shedding. The interaction of rotor with a downstream cylindrical strut has been found to be the main noise source for cooling fans [2]. With the aim to investigate the rotor-strut interaction in order to understand the mechanism of noise generation in a cooling fan, Zhang et al. [6] carried out an experimental work to study the interaction of a circular cylinder with an airfoil near-wake by considering a stationary airfoil with a downstream cylinder where the effects of the lateral distance between the airfoil and the cylinder, the Reynolds number and the incidence angle on the aerodynamic loading on the cylinder were investigated. With the application of cooling fans in mind, their attention was focused on configurations of a cylinder being in close proximity of the blade trailing edge. The effect of the in-line distance between airfoil and cylinder was not investigated.

The flow field for a configuration of a circular cylinder in the wake of an airfoil has not been widely studied. The most relevant study is the research on the flow field with two cylinders in tandem arrangement or staggered in a cross flow, such as the work of Mochizuki et al [3] where the aerodynamic noise generation by the interaction of two tandem cylinders with different diameters was experimentally studied. A good review on the effects of interaction between circular cylinders in cross flow was given by Zdravkovich [5]. Recently Akosile & Summer [1] carried out an experimental work where the aerodynamic forces and the vortex shedding frequencies were measured for staggered two circular cylinders immerged in a uniform shear flow. The wake-body interaction and several critical incident angles were studied in detail. The mutual interference effect of two cylinders in tandem arrangement is very strong and this effect leads to a significant change in fluid forces acting on the cylinders depending on the spacing between them. There is a critical spacing at which fluid forces jump from a low value to a high value and then intermittently switch between those values. This is known as the jump phenomenon. The interactions between two or more cylinders have been extensively investigated. However the wake of bluff body behaves rather differently from that of a streamline body and the results of the investigation cannot be simply extrapolated to the situation to be studied in the present work.

In this paper a numerical study has been carried out for the configuration of a circular cylinder in an airfoil wake using the finite volume method. It is known that the longitudinal, lateral spacing and the attack angle of the airfoil are the important parameters in this configuration. The three groups of calculations are carried out to investigate the effects of the attack angle, the longitudinal and the lateral spacing on flow vortex patterns and the unsteady loading on the circular cylinder. The present work is of both practical and fundamental significance since the configuration is related to a number of engineering applications and also itself represents the generic fluid-structure interaction having important implication for fluid induced vibration and noise generation. The Reynolds number Re based on the diameter of the cylinder d is kept at 200 for all calculations.

Computational Modelling
An airfoil NACA4412 and a circular cylinder are arranged in a staggered configuration in a cross flow shown in figure 1, where d is the diameter of the cylinder, c the chord length of the airfoil, U_∞ the velocity of the uniform flow, α the attack angle of the airfoil, l and T are the longitudinal and lateral spacing between the airfoil and the cylinder respectively, and c/d=7.

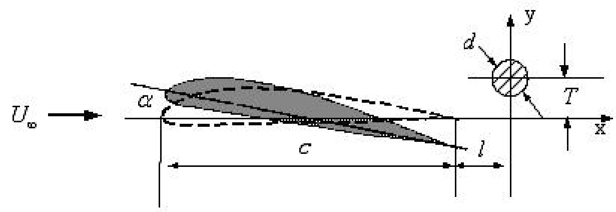


Figure 1: The configuration of a circular cylinder and an airfoil.

The flow is assumed to be unsteady, two-dimensional and laminar, and the incompressible fluid flow can be described by the following continuity and Navier-Stokes equations (non-dimensionalized):

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{\partial p}{\partial x}+\frac{1}{\mathrm{Re}}\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{\partial p}{\partial y}+\frac{1}{\mathrm{Re}}\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \tag{3}$$

where u and v are the velocities in x- and y- directions, Reynolds number Re is defined as Re=$U_\infty d/\nu$, ν is the kinematic viscosity.

The equations are numerically solved using finite volume method where the pressure-velocity coupling is achieved using the SIMPLEC method [4]. These procedures are implemented within a CFD code Fluent 6.0 used in the present study. Unstructured mesh is employed and the computational domain is a rectangle with 60d length and 24d width. The velocity at upstream boundary of the computational domain is set as the uniform velocity U_∞ and the outflow boundary condition is used at the downstream boundary. Time dependent simulation is conducted with the initial conditions of u=1 and v =0 in the entire computing domain. Computations are carried out in PENTIUN IV PCs with run times on the order of 40-60 hrs for grids with typical 10^5 computational cells and run of 50000 time steps.

The drag and lift coefficients are defined as

$$Cd = \frac{2F_x}{\rho U_\infty^2 d} \quad \text{and} \quad Cl = \frac{2F_y}{\rho U_\infty^2 d}$$

where F_x and F_y are the force components in x- and y- direction and ρ is the density of the fluid.

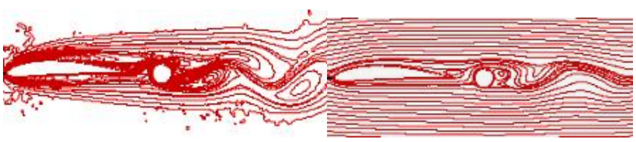
To validate the code and also obtain a baseline for comparison, a single circular cylinder in a cross flow and an isolated airfoil in a cross flow are first calculated respectively. The results of force coefficients, Strouhal number St and the vortex pattern for the single cylinder are compared with previous experimental and computational results reported [7] and good agreement is obtained. Calculations are then carried out in three groups:

- to investigate the effect of the attack angle α, where α varies from -10°, 0°, 7°, 10°, 15° and 20° with l/d=2.5 and T/d=0 fixed;
- to investigate the effect of the in-line distance l/d, where l/d varies from 1, 1.5, 2, 2.5, 3, 4, 5, 6 and 9 for α=5°&15° with T/d=0 fixed;
- to investigate the effect of lateral spacing T/d, where T/d varies from -2, -1, 0, 1 and 2 with l/d =1.5 and α=0° fixed.

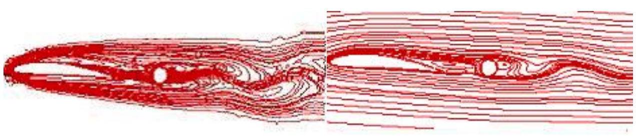
The effects of these parameters on unsteady forces, vortex shedding frequency and vortex patterns of the downstream circular cylinder are examined.

Effects of Attack Angle α

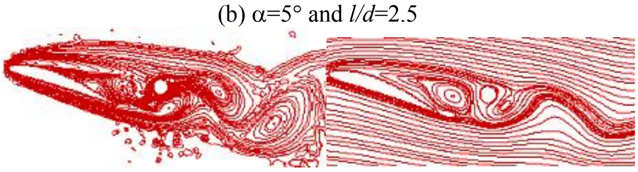
The results for the isolated airfoil show that the flow is attached when α=0° and flow separates when α=5° however no big vortex shed from the airfoil. As α increases the separation point moves upwards from the trailing edge and separation region becomes larger. As α increases further vortices shed forming a vortex street behind the airfoil. When α=15° the shedding frequency of this vortex street is 0.1102.



(a) α=0° and l/d=1.5



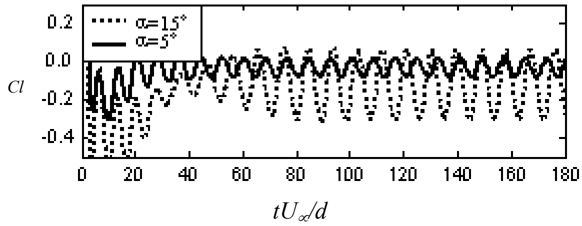
(b) α=5° and l/d=2.5



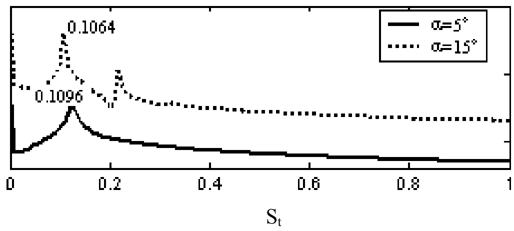
(c) α=15° and l/d=2.5

Figure 2: Vorticity contours and streamlines for α=0°, 5° and 15°.

When a circular cylinder is placed in the near wake of the airfoil, the flow separates from the airfoil earlier and separation point occurs more upwards. The flow separates from the surface of the airfoil and reattaches on the front surface of the cylinder when the attack angle is very small, and forms a separation region between the airfoil and the cylinder (see figure 2 for α=0° and 5°). The flow separates again from the cylinder and shed vortices generating a vortex street after the cylinder. As the cylinder experiences a much slower incident velocity the shedding frequency appears much lower than the value of 0.1885 for an isolated cylinder. This frequency is 0.1448 for the case of α=0° and l/d=1.5, and 0.1096 for α=5° and l/d=2.5. Obviously for the latter case the separation region is relatively larger and the influence of the separation region is therefore stronger. This influence can be seen from the power spectrum of the lift force on the cylinder where the peak of the shedding frequency is slightly broad-banded (see figure 3).

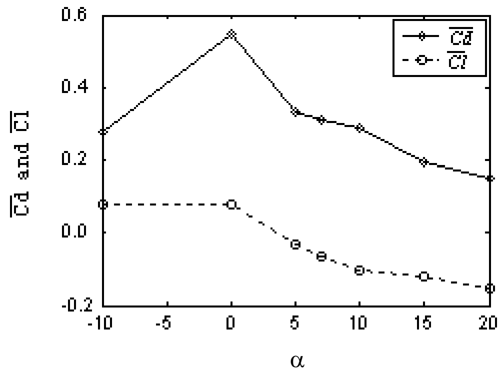


(a) Time histories of the lift force coefficient.



(b) Power spectrum from the lift time histories.

Figure 3: Time history of lift force coefficient and power spectrum from the histories for α=5° and α=15°.



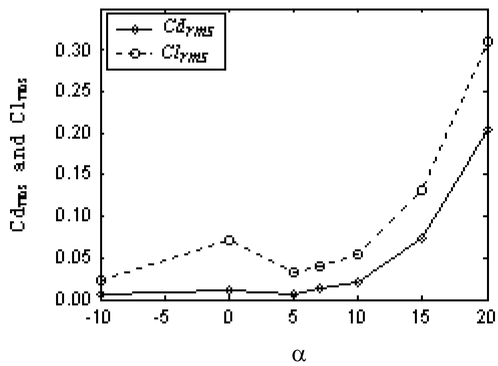


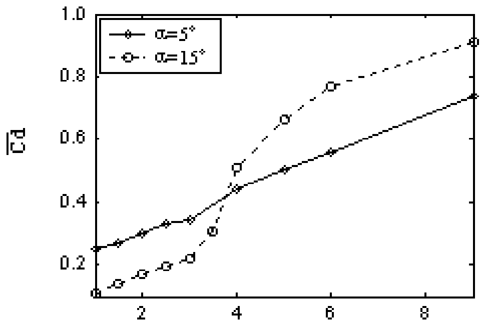
Figure 4: Variation of the force coefficients of the cylinder with α.

When the attack angle increases, the separation region of the airfoil becomes larger. When the l/d is small enough, the cylinder can be wrapped in the separation region. The vortex shedding from the cylinder is then suppressed. The airfoil with the separation region and the cylinder becomes a whole. Big vortices shed and form a Karmen vortex street behind the separation region with a shedding frequency very close to the one for an isolated airfoil. The case with α=15° and *l/d*=2.5 shown in the figures is in this situation where the shedding frequency is 0.1064 which is very close to the value 0.1102 for an isolated airfoil. The feature of the flow field is dominated by the characteristics of the airfoil in this situation.

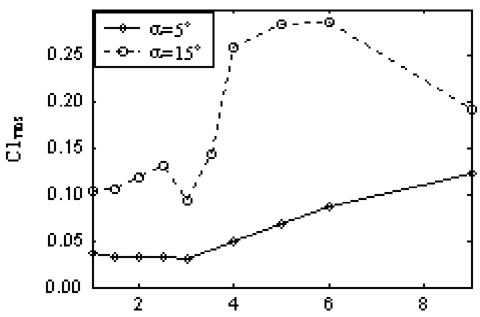
The drag coefficient appears to be much smaller than the value 1.3375 for the isolated cylinder for all cases studied in this section (see figure 4) due to the shadow effect of the upstream airfoil. Not surprisingly the maximum value occurs at α=0° since the cylinder has minimum influence from the airfoil. The drag coefficient reduces quickly as the attack angle increases as expected. It is seen in figure 4 that *Clrms* is very small and does not change much when α varies from 0° to 10°. However *Clrms* increases rapidly as α increases from 10° to 20°. This might be because the unsteady force is mainly due to the cylinder generated vortices when α varies from 0° to 10°, while for α>10°, the unsteady force exerted on the cylinder is due to the more violated shedding of vortices from the separation region with high vorticity density and the impingement of large-scale vortices generated from the separated shear flow from the leading edge (see figure 2 for α=15° and *l/d*=2.5).

Effects of In-line Distance *l/d*

Figure 5 show the variation of *Cd* and *Clrms* on the cylinder with *l/d* for α=5° & α=15°. The drag coefficient for α=15° increases as *l/d* and reaches a value of 0.9059 at *l/d*=9 which is still much smaller than the value 1.3375 for an isolated cylinder. The rms value of *Cl* behaviours differently for the two attack angles with



(a) Drag force



(b) Lift force

Figure 5: The variation of force coefficients on cylinder with *l/d*.

l/d. For α=5° *Clrms* remains unchanged until *l/d*=3 and than starts to increases with *l/d*. However for α=15° where the flow has a large separation region, the value of *Clrms* reaches a minimum value at *l/d*=3 and then increases rapidly to a high value at *l/d*=4 (see figure 5). This, to some extent, shows a similarity to the case of two tandem cylinders. At an in-line spacing of *l/d* > 3.5, the upstream cylinder sheds vortices and the force fluctuation on the downstream cylinder is larger than that at a smaller *l/d* when upstream cylinder fails to shed vortices.

The effect of *l/d* on the vortex shedding frequency is investigated by examining the power spectra of the lift time history and drag time history on the cylinder. The vortex shedding frequency varies with *l/d* and α, and very much depends on the value of α. The frequency value decreases and the separation region of the airfoil becomes larger and wider as α increases (see figure 6 also figure 3). The influence of airfoil wake lasts for a long distance. When α=0°, the vortex shedding frequency of the cylinder is 0.1597 at *l/d*=9 which is still much lower than the value 0.1885 for the isolated cylinder.

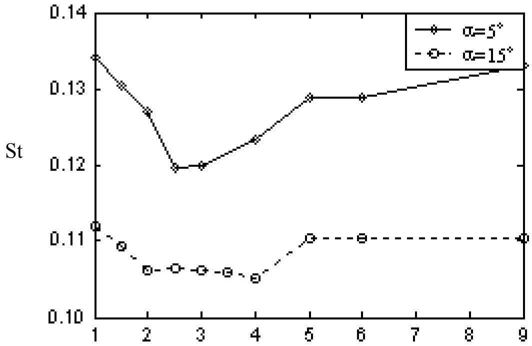
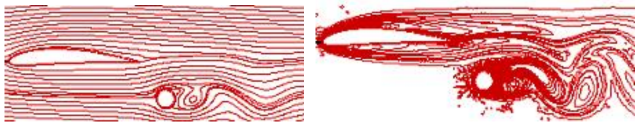


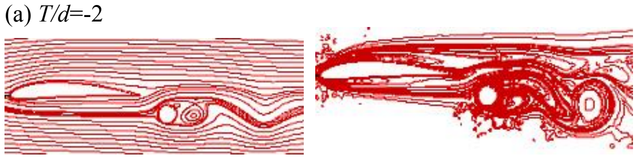
Figure 6: The variation of vortex shedding frequency with *l/d*.

Effects of Lateral Distance *T/d*

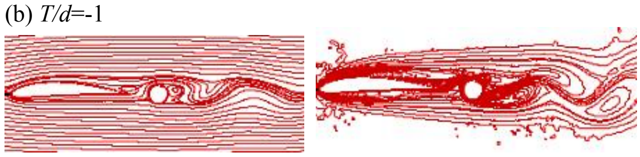
Figure 7 shows the streamlines and the vortex contours for different value of *T/d*. At *T/d*=0 shear layers formed from the upper and lower sides of the airfoil reattach on the front surface



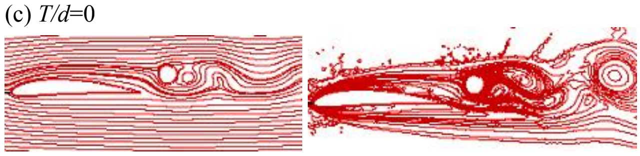
(a) *T/d*=-2



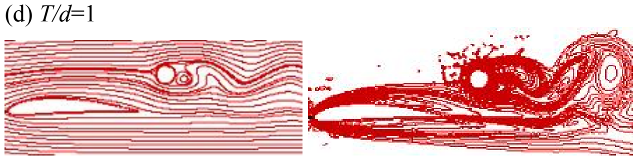
(b) *T/d*=-1



(c) *T/d*=0



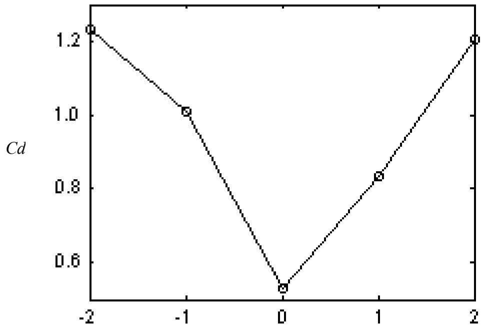
(d) *T/d*=1



(e) *T/d*= 2

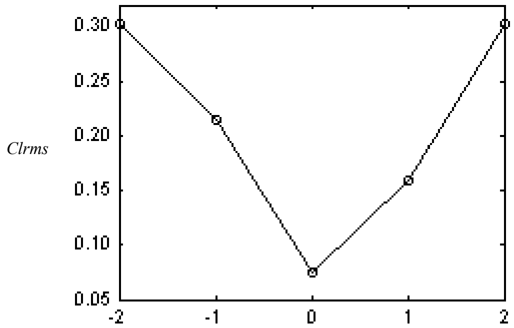
Figure 7: Streamlines and vortex patterns with different *T/d*.

of the cylinder and vortices shed from the cylinder and form a Karmen vortex street (see figure 7(c)). The drag force reaches a low value of 0.5329 (see figure 8) and the shedding frequency also reaches a low value 0.1448 (see figure 9).



T/d

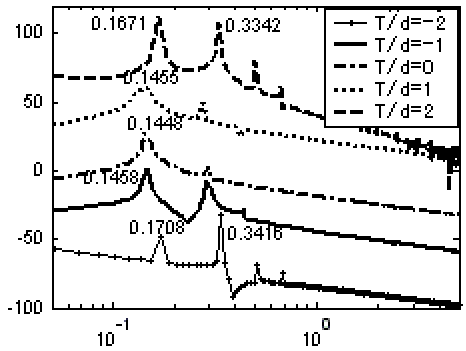
(a) Drag force



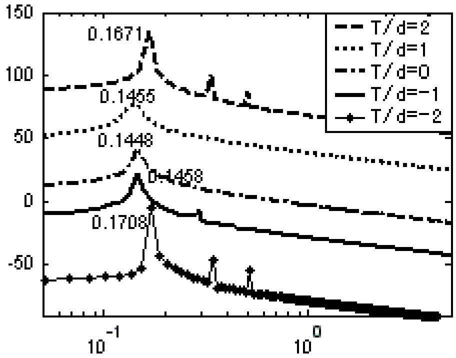
T/d

(b) Lift force

Figure 8: The variation of drag and lift coefficients with *T/d*.



St



St

Figure 9: The power spectrums of lift and drag coefficients with different *T/d*

It is obvious that the influence of the airfoil wake on the cylinder becomes weaker as the distance increases. The drag coefficient and the rms value of lift coefficient increase rapidly with *T/d*. When *T/d*=±2, the value of drag force coefficient and the shedding frequency are very close to the values of 1.3375 and 0.1885 respectively for an isolated cylinder (see figure 8 and 9). It is found that the vortex shedding frequency peak is small and noticeably broad-banded when *T/d*=1, and the value of the frequency is low and almost the same value for *T/d*=0. This lowering of St is attributed to the enlargement of the separation region in near wake of the airfoil due to the presence of the cylinder. This does not happen when *T/d*=-1 as the airfoil is not symmetrical.

Conclusions

The interaction between an airfoil wake and a downstream circular cylinder has been numerically examined using finite volume method for Re=200. The results indicate that the existence of an upstream airfoil has significant impacts on the drag and lift on the cylinder, the vortex pattern and the vortex shedding frequency, and also the presence of a circular cylinder in the near wake of the airfoil can cause flow to separate earlier and the separation point to move upwards. Many parameters play important roles in the interaction. The effects of the attack angle of the airfoil, the in-line and the lateral distances between the cylinder and the airfoil are investigated in this paper. The following conclusions are drawn.

For the cylinder in the near wake of the airfoil (e.g. *l/d*=2.5 studied in the present paper) when the attack angle is small, cylinder generated vortex street persists but with a much lower frequency than the value for an isolated cylinder; when the attack angle is larger the cylinder is wrapped by the separation bubble of the airfoil and the cylinder vortex shedding can be suppressed; increasing the attack angle will cause the drag force on the cylinder reduced and the lift enlarged.

There is a rapid rise in drag coefficient on the cylinder at *l/d*≈3.5 for the case of α=15°, which shows, to some extent, a similarity to the case of two tandem cylinders; force coefficients on the cylinder and shedding frequency increases with *l/d*.

As the lateral distance increases the influence of the airfoil wake on the cylinder declines quickly. When *T/d*=±2, the value of drag force coefficient and the shedding frequency are very close to the values of 1.3375 and 0.1885 respectively for an isolated cylinder.

References

[1] Akosile, O.O. and Summer, D., Staggered Circular Cylinders Immersed in a Uniform Planar Shear Flow, *Journal of Fluids and Structures,* **18**, 2003, 613-633.

[2] Huang, L., Characterizing Computer Cooling Fan Noise, submitted to *J. Acoust. Soc. Am.* (2003).

[3] Mochizuki, O., Kiya, M., Suzuki, S. and Komiyama, K., Aerodynamic Noise Generated by Two Circular Cylinders Arranged in Tandem, *J. of Wind Eng. Industrial Aerodynamics*, **50**, 1993, 107-115.

[4] Patankar, S.V., *Numerical Heat Transfer and Fluid Flow*, Hemisphere, Washington D. C., 1980.

[5] Zdravkovich, M.M., The Effects of Interaction Between Circular Cylinders in Cross Flow, *ASME Journal of Fluids Engineering*, **1**, 1987, 239-263.

[6] Zhang, H.J., Huang, L. and Zhou, Y., Aerodynamic Loading on a Cylinder Behind an Airfoil, submitted to *Physics of Fluids* (2003).

[7] Zhou, C.Y., So, R.M.C. and Lam, K., Vortex-induced Vibrations on an Elastic Circular Cylinder, *Journal of Fluids and Structures*, **13**, 1999, *165-189.*