EXPERIMENTAL INVESTIGATION OF A 2-D MODEL OF FLUID FORCES UPON A CYLINDER ARRAY IN AXIAL FLOW

Aurélien Joly
Alexandre Martin
Pierre Moussou
Luc Pastur
IMSIA UMR 9219
Palaiseau, France

Pierre Badel
Nicolas de Buretel de Chassey
EDF Lab Paris-Saclay
Palaiseau, France

Olivier Cadot
University of Liverpool
United Kingdom

ABSTRACT

The goal of the paper is to investigate experimentally and to a lesser extent numerically the Taylor-Lighthill-Païdoussis (TLP) model of fluid forces in axial flow, in the case of a cylinder array. This model consists of a 2-D representation of the fluid forces, assuming that the local fluid forces depend only on the local angle, curvature, velocity and acceleration of the structure. The experimental setup consists of a wind tunnel with a test section of length 2m and of width 24cm, surrounding a square array of 3x3 cylinders with pitch-to-diameter ratio 1.33. The Reynolds number is about 100 000. The vertical position of the central cylinder is tunable. Velocity profiles are measured by means of Pitot tubes. The steady fluid forces exerted on the central cylinder are recorded using a 6-axis load cell. The boundary layers develop with an increase of the velocity maximum, recalling the concept of entrance length in a pipe. Translating the central cylinder reduces the channel velocity in the narrowed regions. The fluid force acting on the translated cylinder tends to bring it back to its initial position, thus producing a positive fluid stiffness effect. Both the inhomogeneity along the flow direction and the added stiffness suggest significant discrepancies with the assumptions of the TLP model.

NOMENCLATURE

\( C_k \) Non-dimensional fluid stiffness
\( D \) Cylinder diameter
\( D_h \) Hydraulic diameter
\( L \) Cylinder length
\( L_e \) Entrance length
\( P \) Array pitch
\( \rho \) Fluid density
\( S \) Cylinder cross-section
\( U_l \) Measured upstream velocity
\( U \) Calculated mean flow velocity between cylinders
\( u \) Measured local axial velocity
\( x \) Coordinate in main flow direction
\( y, z \) Coordinates in transverse direction
\( w \) Cylinder deflection
\( \dot{w} \) Time derivative of the deflection
\( w' \) Space derivative of the deflection
\( Z \) Cylinder rigid-body displacement

INTRODUCTION

Fluid-structure interaction phenomena play a major role in the seismic design of fuel assemblies in pressurized water reactors. A fuel assembly consists of a 4 m
high and 20 cm wide bundle of 17x17 fuel rods tied together by means of spacer grids. In order to extract the heat produced by the nuclear fuel, the assemblies are submitted to an axial flow of water, from bottom to top. Thus, the basic fluid-structure interaction case at stake here boils down to the transverse motion of a square array of cylinders submitted to an axial flow.

This case fits in the tradition of the study of slender, flexible structures in axial flow. This tradition is based on the early works by Taylor [1] and Lighthill [2] on slender fish locomotion. Later studies, most of which were reviewed in his recent book [6], were mainly dedicated on the instabilities arising in axial flow. However, experimental evidence on full-scale or downsized models of fuel assemblies have shown that, for the case of seismic loading in otherwise normal operating conditions (e.g. fluid velocity of about 5 m/s), the dominant fluid forces are of stabilizing nature. The references report above all a high fluid damping, which is proportional to the flow velocity (see e.g. [7]), but also a stiffening effect of the flow on the fuel assembly, i.e. an added stiffness [8].

Fluid damping has been the focus of some recent studies, for the case of a single cylinder oscillating in axial flow [9] as well as for a full-scale fuel assembly [10]. These studies, while making use of the traditional Taylor-Lighthill-Païdoussis model (TLP), also tackled its limits and suggested updates for the values of some coefficients used in the TLP equations.

The goal of this paper is to investigate experimentally, and to a lesser extent numerically, the TLP model of fluid forces in axial flow, in the case of a cylinder array. We focus on the translational fluid stiffness of the central cylinder of the array. After giving more details on the different terms of the TLP model, we introduce the experimental setup as well as the numerical methods of a preliminary CFD model (computational fluid dynamics). The results are then given in terms of fluid forces and velocity profiles. They are eventually discussed with respect to the TLP model.

THEORETICAL BACKGROUND

In this section, we describe the classical TLP model in the case of a single flexible cylinder in axial flow. The cylinder is assumed to undergo Euler-like bending deformations. The longitudinal deformations are neglected. The transverse deflection (in y direction) is denoted by \(w(x,t)\), \(x\) being in the main flow direction. In the following, \(\dot{w} = \partial w/\partial t\) and \(w' = \partial w/\partial x\).

The representation of the fluid forces introduced by Païdoussis [3] distinguishes between two main contributions: (i) the inviscid force, as introduced by Lighthill [2] and derived via a potential flow calculation in the frame of the slender-body approximation [11]; (ii) the viscous force, using a model intuited by Taylor [1] and recently updated by Divaret et al. [9]. The inviscid force on a cylinder section is perpendicular to the deformed cylinder axis. It can in turn be decomposed into three terms (see e.g. [12]): the added mass term, proportional to the acceleration of the section \(\ddot{w}\) and independent of the flow velocity; the Coriolis term, proportional to the rate of change of angle of incidence of the section \(\dot{w}'\) and to the main flow velocity \(U\); the centrifugal term, proportional to the cylinder curvature \(w''\) and to the square of the main flow velocity. A major contribution of Lighthill consists in finding all these terms proportional to the inviscid added mass, in the assumption of potential flow:

\[
F_{\text{Lighthill}} = -\chi \rho S \left( \ddot{w} + 2U \dot{w}' + U^2 w'' \right), \tag{1}
\]

where \(\rho\) is the fluid density and \(S\) the cylinder cross-section. \(\chi\) is equal to 1 for a single unconfined circular cylinder.

As for the Taylor-Divaret force at low angles of incidence (< 5°), its tangential component remains constant, with friction coefficient \(C_D = 0.012\) (experimental value from [9]), and its normal component is proportional to the apparent angle of incidence \((w' + \dot{w}/U)\), with the slope \(C = 0.11\) [9]:

\[
F_{\text{Taylor-Divaret}} = \frac{1}{2} \rho U^2 DC_D t - \frac{1}{2} \rho U^2 DC \left( w' + \frac{\dot{w}}{U} \right) n, \tag{2}
\]

where \(n\) and \(t\) are the normal and tangential unit vectors of the deformed cylinder axis. Two main assumptions underlie the expression given above:

(i) independence of the global deformed shape (2-D fluid force assumption): the fluid force on a section of the deformed cylinder is the same as if that section was part of a long rigid cylinder with same angle
of incidence and transverse velocity as the considered section;
(ii) independence of the motion time history (quasi-steady assumption): the instantaneous velocity of the section is simply accounted for by introducing the apparent angle of incidence \((w' + wi/U)\), as already mentioned.

The TLP-model described above, sometimes augmented by additional terms to account for specific phenomena, has been extensively used in order to predict instability thresholds in terms of flow velocity, be it buckling or flutter, and to estimate their characteristics (magnitude, frequency) [6]. This model has also allowed for a satisfying explanation of fluid damping on fuel assemblies [10]. In this paper, we shall assess its performance for the case of steady displacements in a cylinder array.

**EXPERIMENTAL APPARATUS AND NUMERICAL METHODS**

As stated in the introduction, the goal of the experimental and numerical studies presented here are to gain knowledge on the fluid forces arising from the displacement of the central cylinder of the array. The experimental setup consists of a wind tunnel with test section of length \(L = 2\) m and width 24 cm surrounding a square array of 3x3 rigid cylinders with pitch-to-diameter ratio \(P/D = 1.33\) (Fig. 1). The vertical position of the central cylinder is tunable. Thin-walled aluminium cylinders of diameter 4.5 cm have been chosen, so as to avoid static deflection under their own weight. The walls of the test section are equipped with half cylinders, such that the developing boundary layers at the wall mimic the ones of a larger number of surrounding cylinders. The central cylinder is supported at its centre only, so as to enable force measurements (see below). The support has a diameter of 2.2 cm. The eight neighbour cylinders are supported at both ends via a grid of cylindrical elements of diameter 0.8 cm.

The mean velocity in the test section \(U\) is calculated by multiplying the measured upstream velocity \(U_i\) by the area ratio \(4(P/D)^2/(4(P/D)^2 - \pi) \simeq 1.79\). \(U\) can reach up to 30 m/s, which gives a maximum Reynolds number of \(UD_h/v_{air} \simeq 100000\), where \(D_h = D(4(P/D)^2 - \pi)/(P/D + \pi - 1) \simeq 5.14\) cm is the hydraulic diameter. This value is not far from the real Reynolds number in a reactor core (about 400000). The turbulent intensity at the entrance of the test section upstream of the noses is about 0.5%. The incoming flow is not perfectly symmetric \((\pm 3\%\), the flow being slightly faster on the \(y < 0\) side).

Velocity profiles along the vertical axis on both sides of the central cylinder and in different sections along the model (x-axis) are measured with an accuracy better than 2% by means of thin Pitot-like tubes, using a Scanivalve DSA 3217/16Px pressure scanner at a sample rate of 100 Hz and averaging over 10 s for each measurement point. The steady fluid forces exerted on the central cylinder are measured with an accuracy of \(\pm 0.02\) N using an AMTI MC3A-100 6-axis load cell, by recording at a sample rate of 1 kHz and averaging over 30 s.

The preliminary numerical results presented here were obtained via simulations performed with Code_Saturne, an EDF in-house open CFD tool based on a collocated finite volume approach [13]. The fluid volume is discretized with a conform hexahedral mesh generated using the software SALOME 7.8.0. The mesh is made up of 5.7 million cells, see Fig. 2. The time step was chosen such that the Courant–Friedrichs–Lewy condition \(CFL < 1\) is met everywhere but at the nose tips. The code uses a centred scheme for the velocity and a simple algorithm for the velocity-pressure coupling. The simulations performed are Reynolds-averaged Navier–Stokes (RANS) simulations with a \(k - \omega\) shear stress transport (SST) turbulence model, which was previously found to be well-suited to the prediction of the fluid force exerted upon a single cylinder in near-axial flow [9]. The mesh is refined close to the walls and a two scale wall law is chosen. The geometry of the numerical model is identical to the one of the experiment, except for the supports of the cylinders (Fig. 1) and for some specificities of the test rig downstream of the cylinder array, which were in both cases not included in the numerical model. We impose a constant and uniform velocity of 10 m/s at the inlet (Re = 60000) and a constant static pressure at the outlet. The sides of the domain and the cylinder walls are modelled as smooth walls. The preliminary simulations have been carried out in the neutral configuration, where the central cylinder is at its original position.
FIGURE 1: EXPERIMENTAL SETUP.

FIGURE 2: CROSS SECTION OF THE UPSTREAM SIDE OF THE MESH.

ENTRANCE LENGTH

Figure 3 shows the results of the velocity measurements. $U$ is the mean velocity between the cylinders, $u$ the local velocity in $x$-direction and $Z$ the vertical position of the central cylinder. The downward peaks at $x/D_h = 3$ are effects of the wake of the supporting elements of the cylinders. They soften and eventually disappear as $x$ increases.

First focusing on the grey curve, for which the central cylinder is at its original position, we observe that the boundary layers at the cylinder walls develop and stabilise after some 20 hydraulic diameters. This recalls the concept of entrance length in a pipe. Using the expression $L_e/d = 1.6Re^{1/4}$ from [14] and replacing $d$ by $D_h$ yields $L_e = 24D_h$ for the conditions of the experiment. The developed boundary layers lead to lower velocities in narrow regions of the cross section and higher velocities in wide areas.

The entrance length is also found in the numerical simulations. Figure 4 shows the axial velocity along a line parallel to the main flow direction and at the centre of the region between four cylinders. To allow for comparison between numerical and experimental results, we added the experimental points corresponding to the six velocity measurement profiles and translated both curves so that their values fit at $x/D_h = 3$. The numerical results exhibit the same trend as the experimental ones, but the plateau reached after $x/D_h = 25$ is much higher. This might be related to the absence in the numerical model of the central supporting element, which in the experiment induces perturbations in the flow and leakage at the top and bottom walls of the wind tunnel.

Focusing now on the experimental velocity profiles in the cases where the central cylinder is translated by $\pm 0.7(P - D)$ (orange and blue curves of Fig. 3), we observe that the channel velocity is reduced by about 5% in the narrowed region, while the velocity maximum remains nearly unchanged. This confirms the first observation that the established flow field is non-uniform, the local velocity being related to some local equivalent channel width. The reorganization of the flow field, from its uniform distribution at the entrance section to its established non-uniform distribution after the entrance length, may question the 2-D representation of the fluid forces previously introduced.

FLUID STIFFNESS

Figure 5 shows the steady transverse fluid force $F_z$ on the central cylinder as a function of its vertical position $Z$. The curve exhibits a quasi-linear behaviour. Considering that the non-zero value at $Z = 0$ is only related to small asymmetries of the experimental setup and uniformity imperfections of the incoming flow, the force can be expressed as follows:

$$F_z = -\frac{1}{2} \rho U^2 DLc_h \frac{Z}{P - D},$$ (3)
FIGURE 3: NON-DIMENSIONAL VELOCITY PROFILES ALONG VERTICAL AXIS AT $y = 0.5P$ AND $U = 16\text{ m/s}$ ($Re = 54000$) FOR THREE DIFFERENT POSITIONS OF THE CENTRAL CYLINDER.

FIGURE 4: EVOLUTION OF THE AXIAL VELOCITY ALONG THE MODEL. EXPERIMENTAL DATA ($Re = 54000$) AND NUMERICAL RESULTS ($Re = 61000$) FITTED AT $x/D_h = 3$.

where experimental values for $C_k$ range from $4.9 \times 10^{-3}$ to $5.9 \times 10^{-3}$. The non-dimensional displacement $Z/(P - D)$ has been chosen such that its value is $\pm 1$ when the cylinders are in contact. Note that the limited accuracy of the load cell, which leads to an estimated uncertainty of $\pm 4 \times 10^{-4}$ on the fluid force coefficient, does not allow to conclude on the asymmetric aspect of the curve.

The sign of the slope reveals a positive fluid stiffness effect: the fluid force acting on a translated cylinder tends to bring it back to its initial position. As mentioned previously, such a trend has already been observed on a statically bended fuel assembly model [8]. However, it was not identified in recent numerical simulations on a bended cylinder inside an array in axial flow [15]. We have not found in the literature any reference to the added stiffness phenomenon for the case of a translated cylinder.

DISCUSSION AND CONCLUSION

The positive fluid stiffness introduced in the previous section cannot be predicted by the simple TLP model presented at the beginning of the paper: zero curvature ($\psi'' = 0$), zero angle of incidence ($\psi' = 0$) and static displacement ($\psi = 0$) lead to zero Lighthill force and Taylor-Divaret force in the normal direction (which is here the same as the direction transverse to the main flow). The velocity measurements that we carried out suggest that the development of the boundary layers and thus the
three-dimensional distribution of the flow field in a cylinder array might be related to the measured fluid stiffness, although the relationship between those two phenomena is yet to be determined.

Hence, the TLP model should be considered as approximate with respect to the development of the boundary layers along the cylinders, which gradually modify the local reference velocity, and also with respect to lateral redistributions of the flow from one channel to the next, which are required to explain the presence of a lateral fluid force. These two observations stem from basic fluid dynamics, and they constitute, at least in the case of a confined cylinder array, a meaningful limitation of the 2-D approach. Thus, any value of the transverse fluid force coefficient of a confined cylinder should be related to its length.

Attempts were made in the literature in some specific cases to account for the non-uniformity of the flow field, e.g. by adding a pressure loss gradient [5] or by modelling the effect of a thick boundary layer [16]. However, in those two examples, the zero space- and time-derivatives of the cylinder deflection \( w \) would also lead to zero fluid forces in the normal direction.

Therefore, further investigations will be carried out in order to augment the TLP model with a term compatible with the results presented here, after having studied the influence of the central vertical support on those results. These investigations imply gaining knowledge on the space distribution of the fluid stiffness. This will be the scope of upcoming numerical simulations. This task will also be attempted experimentally by means of pressure measurements, with the challenge of having to measure tiny pressure differences.

REFERENCES