February 2011

MASTER THESIS

U-RANS Simulation of fluid forces exerted upon an oscillating tube array

Author:
Lise Divaret

Examiner:
Dr. Gunilla Efraimsson
KTH, Stockholm, Sweden

Supervisor:
Dr. Pierre Moussou
EDF R&D, Clamart, France
The aim of this master thesis is to characterize the fluid forces applied to a fuel assembly in the core of a nuclear power plant in case of seism. The forces are studied with a simplified two-dimensional model constituted of an array of 3 by 3 infinite cylinders oscillating in a closed box. The axial flow of water, which convects the heat in the core of a nuclear power plant, is also taken into account. The velocity of the axial flow reaches 4m/s in the middle of the assembly and modifies the forces features when the cylinders move laterally.

The seism is modeled as a lateral displacement with high amplitude (several cylinder diameters) and low frequencies (below 20 Hz). In order to study the effects of the amplitude and of the frequency of the displacement, the displacement taken is a sine function with both controlled amplitude and frequency. Four degrees of freedom of the system will be studied: the amplitude of the displacement, its frequency, the axial velocity amplitude and the confinement (due to the closed box).

The fluid forces exerted on the cylinders can be seen as a combination of three terms: an added mass, related to the acceleration of cylinders, a drift force, related to the damping of the fluid and a force due to the interaction of the cylinder with residual vortices. The first two components will be characterized through the Morison expansion, and their evolution with the variation of the degree of freedom of the system will be quantified. The effect of the interaction with the residual vortices will be observed in the plots of the forces vs. time but also in the velocity and vorticity map of the fluid.

The fluid forces are calculated with the CFD code Code_Saturne, which uses a second order accurate finite volume method. Unsteady Reynolds Averaged Navier Stokes simulations are realized with a k-epsilon turbulence model. The Arbitrary Lagrange Euler model is used to describe the structure displacement. The domain is meshed with hexahedra with the software gmsh [1] and the flow is visualized with Paraview [2]. The modeling techniques used for the simulations are described in the first part of this master thesis.

**Keywords:** Fluid-structure interaction, CFD, Forces, added mass, drag coefficient, Morison expansion, Unsteady Reynolds Averaged Navier Stokes simulations, Arbitrary Lagrange Euler model.
ACKNOWLEDGEMENTS

This master thesis work has been carried out at the AMA (Acoustic and Mechanical analyses) group at EDF R&D under the supervision of Dr. Pierre Moussou. I would like to thank the AMA group for its welcome and its working atmosphere.

I thank Dr. Pierre Moussou for his supervision, support and continual encouragements.

I give my sincere gratitude to Dr. Gunilla Efraimsson for her good advices and comments during my master thesis.
# Table of contents

1. Introduction .......................................................................................................................... 10  
   1.1. Fuel assembly seism design .......................................................................................... 10  
   1.2. Model and assumptions .............................................................................................. 11  
   1.3. Outline of the calculations ......................................................................................... 16  
2. Models and hypothesis used in the simulations ................................................................. 17  
   2.1. URANS model ............................................................................................................ 17  
   2.2. k-ε and k-ω models ................................................................................................... 18  
   2.3. The two-dimensional model ....................................................................................... 18  
   2.4. Boundary conditions ................................................................................................. 19  
   2.5. Numerical software and module ............................................................................... 20  
      2.5.1. Code_Saturne .................................................................................................... 20  
      2.5.2. ALE module ..................................................................................................... 20  
      2.5.3. Steps of the simulations ................................................................................... 21  
   2.6. Axial velocity in the assembly ..................................................................................... 21  
      2.6.1 Implementation in Code_Saturne ....................................................................... 21  
      2.6.2. Source term in the Reynolds equations ............................................................... 22  
      2.6.3. Estimation of the source term ............................................................................ 22  
   2.7. The Morison expansion ............................................................................................... 23  
3. Mesh quality and time step ................................................................................................. 25  
4. Elaboration of inertial and drag force coefficients ............................................................. 30  
   4.1. Variation of the forces in the x-direction: difference between the outer and the inner cylinders ...................................................................................................................... 30  
   4.2. Calculations of Fx in different configurations .............................................................. 30  
   4.3. Relation between residual vortices and force shape ................................................... 33  
5. Determination of the mass and drag coefficients with Morison expansion ....................... 38  
   5.1. Relevance of the Morison expansion to describe the forces in the x-direction .......... 38  
   5.2. Mass and drag coefficients .......................................................................................... 39  
6. Prospects .............................................................................................................................. 42
6.1. Experimental validation ................................................................................................. 42
6.2. Modelling robustness ..................................................................................................... 42
6.3. A coherent temporal model for the forces in the x-direction ....................................... 43
6.4. Characteristic time of the axial velocity ........................................................................ 44
Conclusion ................................................................................................................................ 48
Bibliography .............................................................................................................................. 49
APPENDIX ................................................................................................................................. 50
  Appendix 1: ustsns.F ............................................................................................................. 50
  Appendix 2: article related to the master thesis ................................................................. 52
List of figures

Figure 1: Sketch of a fuel assembly (www.world-nuclear.org) ................................................ 10
Figure 2: Model sketch ............................................................................................................. 12
Figure 3: Partial view of the cylinder array mesh, the lateral parts are not shown ................ 25
Figure 4: Variation the maximum edge ratio vs. displacement .............................................. 27
Figure 5: Variation the skew vs. displacement ......................................................................... 28
Figure 6: Variation the Courant number vs. displacement ...................................................... 29
Figure 7: Variation of the forces in the x-direction with time for the 4 cylinders (base case) 30
Figure 8: Fx vs. time for 3 different amplitude of displacement, cylinder 3 ......................... 31
Figure 9: Fx vs. time for 3 different amplitude of displacement, cylinder 4 ......................... 32
Figure 10: Fx vs. time for 3 different frequencies of displacement, cylinder 3 ..................... 32
Figure 11: Fx vs. time for 3 different frequencies of displacement, cylinder 3 ...................... 33
Figure 12: Variation of forces and velocity in the x-direction with time ................................. 35
Figure 13: Variation of the forces in the x-direction and the vorticity in the z-direction with time .......................................................................................................................................... 36
Figure 14: Variation of forces in the x-direction and pressure with time ................................. 37
Figure 15: Force in the x-direction for the cylinder 3 and Morison expansion associated ...... 38
Figure 16: Force in the x-direction for the cylinder 4 and Morison expansion associated ...... 38
Figure 17: Mass and drag coefficients vs. Keulegan-Carpenter number, Stokes number, Confinement and Axial Reynolds number ........................................................................................................................................ 41
Figure 18: Comparison between the k-ε and the k-ω models for Fx on the cylinder 3 ............ 43
Figure 19: Comparison between the k-ε and the k-ω models for Fx on the cylinder 4 ............ 43
Figure 20: Velocity in the z-direction before and when the cylinder array is moving .......... 44
Figure 21: Variation of the axial velocity vs. displacement ...................................................... 46
Figure 22: Variation of forces in the x-direction and turbulent energy with time ................. 47
List of tables

Table 1: Parameters of the reference case and dimensionless number associated ............... 14
Table 2: Reference case ............................................................................................................ 14
Table 3: Simulation parameters ............................................................................................... 15
Table 4: Parameters studied and dimensionless numbers associated .............................. 31
Table 5: List of the simulations with their parameters ............................................................ 39
List of equations

Equation 1: Averaged Reynolds equations ................................................................. 17
Equation 2: Unsteady Reynolds Averaged Navier-Stokes equations in two dimensions .......... 18
Equation 3: 2-dimensional Reynolds equations in the z-direction ........................................ 22
Equation 4: source term .......................................................................................... 22
Equation 5: Morison formula .................................................................................... 24
1. Introduction

1.1. Fuel assembly seism design

In the nuclear industry, the seismic design of reactor cores requires the structural analysis of fuel rod assemblies undergoing transient excitations. The fuel assemblies are classically described by vibrating beams, and the fluid is usually taken into account with the help of an added mass and an added damping coefficient incorporated into the structure model. Such a representation is of current usage, but the progress of CFD makes a more thorough evaluation of the forces more feasible today.

A seism can be seen as a transient acceleration of the base of the core of the nuclear power plant. This acceleration sets the fuel assemblies in motion. It creates a risk of buckling for the grid assemblies which hampers the insertion of the control rods required to stop the nuclear reaction. Safety rules are constantly evolving; hence it is useful to have a precise model of the forces applied to the assembly in case of seism.

The core of a nuclear power plant comprises several fuel assemblies. Each assembly is around 3m high and is constituted by an array of 17 by 17 fuel rods which are immersed in water. The rods exhibit a diameter equal to 9mm and are equally spaced by a 3mm gap: the pitch to diameter ratio is about 1.33. The heat of the nuclear reaction is transferred by an axial flow along the assemblies. During a seism, the assemblies are assumed to move at low frequencies (below 20 Hz) and the displacements can reach a couple of rod diameters.
The case of an array of cylinders in an axial or a lateral flow has been widely studied [3,4,5] but in this situation, the flow created by the lateral displacement and the axial flow have approximately the same Reynolds number (around $10^4$). As the axial flow reaches 5m/s in the assembly, the flow is fully turbulent.

The objective of the master thesis is to study the forces exerted on a reduced fuel assembly moving laterally in an axial flow. The structures are supposed infinitely rigid and the fluid forces exerted on the rods are calculated with the help of CFD.

### 1.2. Model and assumptions

The case studied in the master thesis is a simplified assembly constituted of 3 by 3 infinite rods in a closed box. This simplified assembly is moving laterally (in the x-direction) as a rigid body. As the aim of this master thesis is to study the effects of the amplitude and the frequency of the cylinder vibrations, the motion chosen is a sine function with controlled amplitude and frequency.

The reason for choosing a controlled sine function displacement is to facilitate the analysis of the fluid dynamics. Such a representation cannot be used as it stands for seismic analysis, since non-linear terms in the fluid forces forbid to expand a transient displacement in Fourier terms. However, this approach constitutes a first step in the understanding of fluid dynamics.

During a simulation, the cylinder array will move from 2 to 30 periods (it depends on the frequency studied). For low frequencies, the time step is driven by the axial flow which restricts the number of periods simulated (10 periods). To avoid peaks in the force curves, the velocity of the cylinder array is also taken equal to 0 when the array begins to oscillate (see part 2.5.2).
An axial flow is set in the z-direction: its velocity reach 4m/s in the cylinder array and the Reynolds number of the axial flow is around $10^5$. The turbulence of the axial flow is supposed to be fully developed (with no large turbulent structures in the z-direction). In this case, a two dimension model is assumed to be relevant and the mesh used will be one cell high.

A sketch of the model is presented in the Figure 2. The forces applied by the fluid on the nine cylinders will be studied over time. Nevertheless the model exhibits two symmetries which simplify the number of cylinders to consider:

- The arrangement of the assembly and its displacement (a sine motion) are symmetrical with respect to the x-axis. Hence only the two first lines of the array have to be considered.
- The arrangement of the assembly is symmetrical with respect to the y-axis and the cylinders displacement is a periodic odd function. This leads that the assembly can be studied during only a half period or the two first column of the assembly can be studied during one period.

As a consequence only four cylinders need to be considered in the analysis of the forces indicated in the Figure 2. The cylinders 3 and 4 are more representative of a larger assembly as they are not directly close to the walls. The cylinders 1 and 2 will illustrate some effects of the confinement.

One objective of the master thesis is to determine parameters of which the forces on the cylinders has a large dependance. Four parameters will be tested:

- the frequency of the cylinder displacement (F)
- the amplitude of the cylinder displacement (X)
- the axial pressure gradient ($\Delta P/\Delta z$), which drives the axial velocity $V_z$
- the cylinder confinement in the x-direction ($\Delta x$)
Four dimensionless numbers will be used based on these four parameters:

- The Stokes number related to the frequency of the cylinder displacement
- The Keulegan-Carpenter number related to the amplitude of the cylinder displacement
- An axial Reynolds number related to the axial velocity
- A dimensionless confinement related to the cylinder confinement in the x-direction

The Stokes number gives a dimensionless frequency:

\[ \beta = \frac{D^2 f}{v} = Re \frac{D f}{U} \]

It is the characteristic response time of the fluid divided by the characteristic time of the motion of the cylinder array. It is here used with an analogy with the fluid-particle two phase flows where the Stokes number is the characteristic response time of the fluid divided by the characteristic response time of the particle.

If the Stokes number is small, that is much less than 1, it means that the cylinder array motion is slightly coupled to the fluid motion. This suggests that a quasi static approach can be taken.

If the Stokes number is large, the cylinder array motion and the fluid motion are coupled. This suggests that a URANS approach has to be taken (if the characteristic time of the rod displacement is much higher than the characteristic time of the turbulence). In the simulations performed during this master thesis work, \( \beta = [162 - 1215] \), such that a quasi-static approach cannot be taken.

The dimensionless oscillation amplitude can be described by the Keulegan-Carpenter number [6]:

\[ KC = 2\pi \frac{X}{D} \]

This number is typically used to describe the relative importance of the drag forces over inertia forces for an oscillatory fluid flow. As in our case the fluid oscillates at the same frequency as the cylinder array, this description makes still sense.

The Reynolds number is the ratio between the viscous and the convective time scales:

An axial Reynolds number Re can be used to describe the velocity in the z-direction: the maximum velocity in the array (at the point P1) is taken as the reference velocity.
The Table 1 sums up the parameters studied with their range of investigation and the dimensionless number associated.

\[
Re = \frac{\tau_v}{\tau_c} = \frac{D^2/v}{D/U} = \frac{DU}{v}
\]

A reference case is also chosen and will be used as a reference for the comparisons between the simulations (see Table 1).

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency of displacement</td>
<td>5 Hz</td>
</tr>
<tr>
<td>Amplitude of displacement</td>
<td>2 D</td>
</tr>
<tr>
<td>Axial velocity</td>
<td>4m/s</td>
</tr>
<tr>
<td>Confinement</td>
<td>11.5 D</td>
</tr>
</tbody>
</table>

Table 2: Reference case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency of displacement</td>
<td>5 Hz</td>
</tr>
<tr>
<td>Amplitude of displacement</td>
<td>2 D</td>
</tr>
<tr>
<td>Axial velocity</td>
<td>4m/s</td>
</tr>
<tr>
<td>Confinement</td>
<td>11.5 D</td>
</tr>
</tbody>
</table>

Table 1: Parameters of the reference case and dimensionless number associated

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Dimensionless number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency of displacement</td>
<td>( \beta = D^2F/v )</td>
</tr>
<tr>
<td>Amplitude of displacement</td>
<td>( KC = 2\pi X/D )</td>
</tr>
<tr>
<td>Axial velocity</td>
<td>( Re = DU/v )</td>
</tr>
<tr>
<td>Confinement</td>
<td>( C = \Delta x/D )</td>
</tr>
<tr>
<td>Name of the parameter</td>
<td>Function</td>
</tr>
<tr>
<td>-----------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Turbulence model</td>
<td></td>
</tr>
<tr>
<td>Wall law</td>
<td></td>
</tr>
<tr>
<td>I</td>
<td>Length of the computational domain (in the y-direction)</td>
</tr>
<tr>
<td>v</td>
<td>Viscosity</td>
</tr>
<tr>
<td>ρ</td>
<td>fluid density</td>
</tr>
<tr>
<td>P</td>
<td>Pressure reference</td>
</tr>
<tr>
<td>dt</td>
<td>Time step for calculation</td>
</tr>
<tr>
<td>NT</td>
<td>Number of time step</td>
</tr>
<tr>
<td>Np</td>
<td>Number of periods studied</td>
</tr>
<tr>
<td>Nc</td>
<td>Number of cells</td>
</tr>
</tbody>
</table>

Table 3: Simulation parameters
1.3. Outline of the calculations

The chapter 2 includes a presentation of the models and hypothesis used for the simulations.

In the chapter 3, a mesh study is done in order to have reliable simulations.

In the chapter 4, a study of the velocities, the pressure and the vorticity is realized in a first simulation done on the base case (Table 2). The forces in the x-direction show a difference of amplitude between the inner and the outer cylinders and an asymmetry for the outer cylinders.

In the chapter 5, a parameter study is realized in order to determine the influence of four parameters: the frequency and the amplitude of the oscillations of the cylinder array, the axial velocity of the fluid and the confinement in the x-direction. The curves of the forces in the x-direction are fitted with a model based on the generalization of the Morison equation [4]. The variation of the coefficients of the Morison expansion with the four parameters is thereafter studied.

The chapter 6 gives the limits of the calculations done and proposes further studies.
2. Models and hypothesis used in the simulations

2.1. URANS model

\[
\frac{\partial U_x}{\partial t} + \frac{\partial}{\partial x} \left[ U_x U_x + (\bar{u}_x \bar{u}_x) \right] - \nu \left[ \frac{\partial U_x}{\partial x} + \frac{\partial U_x}{\partial y} + \frac{\partial U_x}{\partial z} \right] + \frac{\partial}{\partial y} \left[ U_x U_y + (\bar{u}_x \bar{u}_y) \right] - \nu \left[ \frac{\partial U_x}{\partial y} + \frac{\partial U_y}{\partial y} + \frac{\partial U_z}{\partial z} \right] + \frac{\partial}{\partial z} \left[ U_x U_z + (\bar{u}_x \bar{u}_z) \right] - \nu \left[ \frac{\partial U_x}{\partial z} + \frac{\partial U_z}{\partial z} + \frac{\partial U_z}{\partial z} \right] = 0
\]

Equation 1: Averaged Reynolds equations

An earthquake corresponds to a displacement of the fuel rods in a larger domain. The borders of the domain can be considered as fixed and the rods are moving with a controlled displacement. In this case, LES simulations would take too much time but a steady approach cannot describe the problem. An unsteady RANS (Reynolds-Averaged Navier-Stokes) appears here as a good compromise.

The URANS model is similar to the RANS model but the transient term is taken into account. As in the RANS model, the pressure and the velocity are decomposed into a mean component and a fluctuating component.

This can be done if the characteristic time of the rod displacement (which will be the characteristic time of the averaged velocity) is much higher than the characteristic time of the turbulence. The characteristic time \( t_{\text{turb}} \) of the convective effect can be evaluated with the help of the characteristic length (taken as the distance \( d \) between the cylinders: \( d = 3 \text{e-3m} \)) and the axial velocity (which is about 3-5m/s). As the characteristic frequencies of an earthquake are below 30 Hz, and as the simulations presented in this thesis are done for frequencies below 15 Hz, the characteristic time of the rod displacement \( t_{\text{rod}} \) is greater than \( 7\text{e-2s} \). The ratio between \( t_{\text{rod}} \) and \( t_{\text{turb}} \) will be in any case greater than 70 and around 200 in the base case (where the frequency of the rod displacement is equal to 5 Hz). Hence it is possible to use a URANS model.

The Reynolds stress \( \tau_{ij} = \bar{u}_i \bar{u}_j \) is unknown in the Reynolds equations: with the RANS or URANS model, a turbulence model is also necessary to close the problem.
2.2. k-ε and k-ω models

The k-ε and k-ω models are high Reynolds models: they describe the flow outside the viscous sub-layers, where the viscous effects are negligible compared to the turbulent effects [7,8]. They are first order and two equation models. A standard k-ε model has been used for all simulations. A comparison is done with the k-ω model in the section 6.2.

2.3. The two-dimensional model

In a fuel assembly, the uranium rods are maintained with grid assemblies. To improve the fuel rod cooling efficiency, the grids have mixing blades, which create a rotating flow in the z-direction. Hence the flow between the rods is not a priori 3-dimensional. However 3-dimensional simulations would take too much time and would not correspond to the aim of this master thesis which is a feasibility study on simplified cases.

To simplify the calculations, it can be assumed that far from the grids, the flow is parallel and the turbulence fully developed. As a consequence, a 2-dimensional approach can be considered. This assumption on the topology of the flow in the z-direction engenders two hypotheses which simplify the Averaged Reynolds equations:

- the mean velocities are independent of z
- the Reynolds stress is independent of z

These assumptions simplify the Navier-Stokes equations and the pressure can be expressed in the form:

\[
\frac{\partial U_x}{\partial t} + \frac{\partial}{\partial x} \left[ (U_x U_x) + (\bar{u}_x \bar{u}_x) - 2\nu \left( \frac{\partial U_x}{\partial y} + \frac{\partial U_x}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left[ (U_x U_y) + (\bar{u}_x \bar{u}_y) - \nu \left( \frac{\partial U_x}{\partial y} + \frac{\partial U_x}{\partial x} \right) \right] = 0
\]

\[
\frac{\partial U_y}{\partial t} + \frac{\partial}{\partial x} \left[ (U_y U_x) + (\bar{u}_y \bar{u}_x) - \nu \left( \frac{\partial U_y}{\partial y} + \frac{\partial U_y}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left[ (U_y U_y) + (\bar{u}_y \bar{u}_y) - 2\nu \left( \frac{\partial U_y}{\partial y} + \frac{\partial U_y}{\partial x} \right) \right] = 0
\]

\[
\frac{\partial U_z}{\partial t} + \frac{\partial}{\partial x} \left[ (U_z U_x) + (\bar{u}_z \bar{u}_x) - \frac{\nu}{2} \left( \frac{\partial U_z}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left[ (U_z U_y) + (\bar{u}_z \bar{u}_y) - \frac{\nu}{2} \left( \frac{\partial U_z}{\partial y} \right) \right] + \frac{\partial}{\partial z} \left( \frac{\nu}{\rho} \right) = 0
\]

Equation 2: Unsteady Reynolds Averaged Navier-Stokes equations in two dimensions

In the 2 dimension Reynolds equations, in the equation in the z-direction, the mean velocities and the Reynolds stress are independent of z (by hypothesis). As all the variables of the equation (2) are independent of z, the z-component of the pressure gradient cannot depend of z, which means that the pressure linearly depends on z. Hence, the pressure can be written:

\[
\rho \frac{\partial U_z}{\partial t} + \frac{\partial}{\partial x} \left[ (U_z U_x) + (\bar{u}_z \bar{u}_x) - \frac{\nu}{2} \left( \frac{\partial U_z}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left[ (U_z U_y) + (\bar{u}_z \bar{u}_y) - \frac{\nu}{2} \left( \frac{\partial U_z}{\partial y} \right) \right] + \frac{\partial}{\partial z} \left( \frac{\nu}{\rho} \right) = 0
\]
\[ p = \overline{p}_{2D}(x, y, t) + K(x, y, t)z \]

In the Reynolds equation in the \( x \) and \( y \)-directions, the mean velocities and the Reynolds stress are independent of \( z \). With the same arguments as above, the pressure gradient in the \( x \) and \( y \)-directions is necessarily independent of \( z \):

\[
\begin{align*}
\frac{\partial p}{\partial x} &= \frac{\partial \overline{p}_{2D}(x, y, t)}{\partial x} + \frac{\partial K(x, y, t)}{\partial x} z = f(x, y, t) \\
\frac{\partial p}{\partial y} &= \frac{\partial \overline{p}_{2D}(x, y, t)}{\partial y} + \frac{\partial K(x, y, t)}{\partial y} z = g(x, y, t)
\end{align*}
\]

where \( f \) and \( g \) are functions of \( x, y \) and \( t \).

As a consequence,

\[
\frac{\partial K(x, y, t)}{\partial x} = \frac{\partial K(x, y, t)}{\partial y} = 0
\]

Hence \( K \) is uniform.

The pressure can be expressed in the form:

\[ p = \overline{p}_{2D}(x, y, t) + K(t)z \]

A 2-dimensional model has been created but it also includes parameters in the \( z \)-directions: the velocity in the \( z \)-direction (the axial velocity) is taken into account. Then this model seems appropriate to describe the issue.

### 2.4. Boundary conditions

It would be necessary to modify the standard \( k-\varepsilon \) model to compute the viscous near-wall region. This requires refined meshes close to the walls and generates higher computation time, which can be avoided by using wall functions.

Pope [9] lists some of the near-wall effects:

- a low Reynolds number which tend to zero at the wall
- an high shear rate
- a two-component turbulence: as the wall is approached the turbulence tends to the two-component limit
- an impermeability condition at the wall

As the standard \( k-\varepsilon \) model is not adapted to this region, a law of the wall is used to avoid solving the equations in the viscous sublayer [10]. This law of the wall is used in the first
mesh close to the wall: this is an analytical law which represents the behavior of the viscous sublayer.

The law of the wall used in all the master thesis simulations is a law of the wall with two velocity scales. It is the model used by default in Code_Saturne. It often reduces some default of the $k$--$\varepsilon$ model. The two velocity scales used are:

- $u_k$, the friction velocity obtained for the turbulent energy.
- $u'$, the wall friction velocity obtained from the Reynolds stress

The distance from the wall is measured in wall units:

$$y^+ = \frac{yu'}{v},$$

where $y$ is half of the wall cell height.

The law of the wall requires that $y^+>30$ for the first cell, which means greater than the viscous sublayer height. As a consequence, the law of the wall imposes restrictions on the size of the mesh, at least close to the wall. Hence, the results of the simulations are not independent of the mesh size.

### 2.5. Numerical software and module

#### 2.5.1. Code_Saturne

Code_Saturne [11] is EDF’s general purpose computational fluid dynamics software based on a co-located Finite Volume approach. Its basic capabilities enable the handling of either incompressible or expandable flows with or without heat transfer and turbulence (e.g. mixing length, 2-equation models, v2f, Reynolds stress models, Large Eddy Simulations).

#### 2.5.2. ALE module

The Arbitrary Lagrangian-Eulerian (ALE) Method ALE is the resolution of the Navier-Stokes equations in a moving coordinate system, where the additional grid velocities are added in the convective terms.

$$\frac{\partial u}{\partial t} + (u - u_0) \cdot \nabla u + \frac{1}{\rho} \nabla p - \nu \nabla^2 u = 0$$

$u$ is the fluid velocity

$u_0$ is the mesh velocity.

In the case studied, the cylinder array is moving in the $x$-direction and the nodes of the mesh can move in the $x$ and $y$-directions. The software Code_Saturne allows for controlling the
displacement of the nodes which are on a wall. This will be done to avoid cell negative volumes but the inner nodes will be moved automatically.

To avoid peaks of forces, the velocity of the array is equal to 0 when the array begins to move. This is done by multiplying the function of displacement (a sine) by a function exponentially converging to 1. The time constant of the exponential is chosen in order to have the possibility to study few periods for the forces in the x-direction.

2.5.3. Steps of the simulations

The simulations were done in two steps:
1) The axial flow was simulated until the mean flow becomes stationary
2) The ALE simulations were performed

During the simulations the forces in the x-direction for the cylinders are written in a file for each time step. They will be analyzed in the chapter 4.

2.6. Axial velocity in the assembly

2.6.1 Implementation in Code_Saturne

An axial flow in the z-direction has to be created with a velocity around 4 m/s in the array (at the point P1, see Figure 3). Some simulations has been done in a stationary case, without starting the ALE module (the cylinder is not moving) to determine the number of steps necessary to make the mean flow converge to its asymptotic value.

The usual way of creating an axial flow in the z-direction is to set a pressure gradient in the z-direction. The software Code_Saturne does not allow to define a pressure gradient, but two other ways of creating the flow are possible:

- in controlling the fluid flow through an input surface and defining the output surface
- with a condition of invariance by translation in the z-direction and a definition of the pressure gradient by adding a source term in the momentum equation.

The condition of invariance in the z-direction leads to the second way of creating the flow in the z-direction since it is the only way to express the symmetry condition. Therefore a source term as to be added in the z-component of the momentum equation.
2.6.2. Source term in the Reynolds equations

The 2-dimensional Reynolds equations in the z-direction can be written with a source term which corresponds to the pressure gradient in the z-direction.

\[
\frac{\partial U_z}{\partial t} + \frac{\partial}{\partial x} \left( U_z \overline{u_x} + (\overline{u_z} \overline{u_x}) - \frac{\nu}{2} \left( \frac{\partial U_z}{\partial x} \right) \right) + \frac{\partial}{\partial y} \left( U_z \overline{u_y} + (\overline{u_z} \overline{u_y}) - \frac{\nu}{2} \left( \frac{\partial U_z}{\partial y} \right) \right) - S_z = 0
\]

**Equation 3: 2-dimensional Reynolds equations in the z-direction**

\[ S_z(x, y, t) = -\frac{\partial}{\partial z} \left( \frac{p}{\rho} \right) \]

**Equation 4: source term**

The pressure model for the 2-dimensional model (see part 2.6.3) can be applied in the formula of the source term:

Therefore, the source term in the z-direction is constant, and its value will control the mean velocity in the z-direction.

2.6.3. Estimation of the source term

As seen before, the pressure gradient in the z-direction is directly coded in Code_Saturne as a constant source term in the momentum equation. The mean velocity in the z-direction will depend on the value of the source term. Hence it is necessary to find the value of the source term in order to generate a proper velocity.

The walls around the cylinders create a straight and uniform duct in which the flow is incompressible and the fully turbulent. The pressure gradient in the axial direction can be estimated from the Darcy-Weishnach formula which gives the fluid shear stress (imposed on the wall of the duct).

\[ \tau = \frac{\rho U^2 f}{2} \]

where

- \( f \) = dimensionless friction factor
- \( U \) = average flow velocity over the cross section
- \( \rho \) = fluid density
This formula is not directly related to the Reynolds equation but is used to give an estimation of the fluid shear stress.

The momentum equation can be written

\[
\frac{\partial U_x}{\partial t} + \frac{\partial}{\partial x_j} \left[ (U_x U_j) + \frac{p}{\rho} \delta_{xj} - \frac{\tau_{xj}}{\rho} \right] = 0
\]

The fluid is stationary and there is no velocity in the x and y directions and the velocity in invariant with z. Then the equation is integrated over the volume.

\[
\int_V \frac{\partial}{\partial x_j} p \delta_{xj} dV = \int_V \frac{\partial}{\partial x_j} \tau_{xj} dV
\]

\[
\int_S p \delta_{xj} dS = \int_S \tau dS,
\]

where S is the section of the duct, l, the length of the lateral surface, h is the height of the duct used for the integration. The pressure is taken between the two sections called « sup » and « inf ».

\[
(p^{sup} - p^{inf})S = \tau (2l_{wall1} + 2l_{wall2} + 9l_{cyl})h
\]

With taking the Darcy-Weishnach formula for \( \tau \), the pressure gradient can be estimated:

\[
S_z = \frac{\Delta P}{h} = \tau \frac{P}{S} = \frac{\rho U^2}{2} \frac{f P}{4 S}
\]

, where P is the wetted perimeter of the cross section

The friction coefficient \( f \) can be estimated from the curves of friction factor as a function of Reynolds number, which can be found in a handbook. A friction coefficient \( f= 2.0E-2 \) is taken and this value will be used in all the simulations.

This pressure gradient is implemented in a FORTRAN file which is compiled at each simulation (see Appendix 1).

2.7. The Morison expansion

The Morison expansion will be used to describe the forces in the x-direction for the cylinders of the assembly. It a simple model used for low Reynolds numbers (Re<1000). In this case, the model will be fitted to the force curve to see if it can give a simple description of the shape of the forces. A way to improve the model will be described in the chapter 6.3.
Morison (1950) [4] proposed that the force in the x-direction of one cylinder that oscillates in a fluid at rest is expressed with an inertial term (proportional to the acceleration of the cylinder) and by a damping term (no linear and depending on the velocity of the cylinder). The coefficients are the added mass coefficient and the drag coefficient [12] respectively.

The Morison formula is:

\[ F_{x}^{\text{Morison}} = -\alpha_{m} \frac{\rho \pi D^2}{4} \ddot{x} - \alpha_{d} \frac{1}{2} \rho D |\dot{X}| \dot{X}, \]

**Equation 5: Morison formula**

where \( \alpha_{m} \) is the dimensionless coefficient of the added mass term, assumed to be proportional to the acceleration of the cylinder. In the above expression, the drag term \( \alpha_{d} \) follows a quasi-steady law, and unsteady effects are overlooked. Nevertheless, such an expression has the merit of simplicity, and as the term \( \dot{x} |\dot{X}| \) is reasonably close to a sine function, the identification of a couple of dimensionless coefficients \( \alpha_{m} \) and \( \alpha_{d} \) is straightforward.

In the cases studied, the Morison expansion is generalized to the case of an array of cylinders oscillating in an axial flow. For each cylinder, a Morison expansion is fitted with the least square method to determine the added mass and the drag coefficient.
3. **Mesh quality and time step**

As shown in the Figure 3, the fluid mesh around the cylinders is constituted of hexahedrals only as required by the best practice in fluid dynamics, with a small width in the axial direction associated with one cell. The mesh generator *GMSH* [1] was used to generate the mesh which consists of 19560 cells.

![Figure 3: Partial view of the cylinder array mesh, the lateral parts are not shown](image)

Three indicators can be controlled to evaluate the quality of the mesh: the skew of the mesh, namely the maximum edge ratio and the Courant number.

**Maximum edge ratio**

The maximum edge ratio is the largest ratio of two edges of a cell. For good results, it should be lower than 1.3. In this case, the value is greater but still reasonable. In the Figure 4, the variation of the maximum edge ratio is due to the displacement of the array of cylinders: a compression reduces the maximum edge ratio and a dilatation tends to increase it.

**Skew**

The skew gives information about the angles deformations of a cell. It should be lower than 0.5.

The Figure 5 shows that the skew is controlled: the maximum skew values are in the region close to the cylinders which is not deformed when the cylinders are moving. The compressed and dilated cells are always rectangular so the skew does not change for these cells.

**Courant number**
The Courant number should be lower than 4-5. This is a condition necessary to ensure the convergence of the equations. The value of the Courant number can be decreased by decreasing the time step or the velocity of the flow, or by increasing the mesh size. The most sensitive meshes are those near the cylinder which have a high velocity in the x-direction when the cylinder is moving or the one far from the cylinder and the walls which have a high velocity in the z-direction. These meshes have the highest Courant number.

The observation of the variation of the Courant number in the Figure 6 compared to the evolution of the velocity in the z-direction (Figure 20) shows that the Courant number is driven by the flow in the z-direction. As a consequence, the time step depends on the velocity in the z-direction.
Figure 4: Variation the maximum edge ratio vs. displacement
Figure 5: Variation the skew vs. displacement
Figure 6: Variation the Courant number vs. displacement
4. Elaboration of inertial and drag force coefficients

4.1. Variation of the forces in the x-direction: difference between the outer and the inner cylinders

The displacement of the cylinder array in the x-direction is a sine function with a given amplitude and frequency. Figure 7 shows the forces in the x-direction of the four cylinders studied for the reference case.

![Figure 7: Variation of the forces in the x-direction with time for the 4 cylinders (base case)](image)

The force plot shows different behaviors between the outer and the inner cylinders. The force in the x-direction is symmetrical for the inner cylinders whereas the outer cylinders exhibit a very different force shape when they are in the head or in the tail of the row. The maximal forces are observed for the leading outer cylinders, which is what would be expected by analogy with a row of cylinders moving at constant speed. The force amplitudes for the central cylinders or for the cylinders at the back of the pack are smaller: these cylinders are hydrodynamically screened by the leading cylinders.

4.2. Calculations of Fx in different configurations

The parameters studied are described in the Table 4.
Table 4: Parameters studied and dimensionless numbers associated

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Formula</th>
<th>Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency of displacement</td>
<td>$\beta = \frac{D^2 F}{\nu}$</td>
<td>$[162 - 1215]$</td>
</tr>
<tr>
<td>Amplitude of displacement</td>
<td>KC = $2\pi X/D$</td>
<td>$[0.2 - 25.12]$</td>
</tr>
<tr>
<td>Axial velocity</td>
<td>Re = $DU/\nu$</td>
<td>$[17.10^3 - 55.10^3]$</td>
</tr>
<tr>
<td>Confinement</td>
<td>C = $\Delta x/D$</td>
<td>$[3 - 11.5]$</td>
</tr>
</tbody>
</table>

For each simulation, the forces on the cylinders 3 and 4 are plotted: these cylinders are representative of an inner and an outer cylinder. Figure 8-11 below show the forces for different amplitudes and frequencies. In the first two figures, the shape of the forces varies a lot with the amplitude of the displacement for both inner and outer cylinders. On the contrary, the shape of forces does not vary a lot with the frequencies especially for the inner cylinders. Moreover the dimensionless forces for the central cylinders (the force divided by the drag force related to the maximal velocity of the array) are similar which suggests that the force applied to the central cylinder is mainly a drag force in the range of frequency considered.

**Figure 8:** $F_x$ vs. time for 3 different amplitude of displacement, cylinder 3
Figure 9: Fx vs. time for 3 different amplitude of displacement, cylinder 4

Figure 10: Fx vs. time for 3 different frequencies of displacement, cylinder 3
4.3. Relation between residual vortices and force shape

Figure 12-14 show the velocity in the x-direction, the vorticity in the z-direction and the pressure and the forces in the x-direction for the cylinders 3 and 4 for six different times, respectively. It is important to notice that the force is never in phase with the displacement nor with the velocity. There is a delay between the maximal velocity and the maximal force for the inner and outer cylinders, which is a reason for the fluid damping effect.

The following section describes the evolution of the forces with the displacement for one period.

$t=0$

Initially, the velocity of the cylinder array is maximal, the cylinder 3 is in the row tail and its force is close to its minimum and slightly increasing as is the case for the central cylinder.

$t=\pi/2$

The cylinder array reaches its maximal position; the velocity is equal to 0. The forces continue to increase for both cylinders.

$t=\pi/2+\epsilon$

The array of cylinders is coming back, the cylinder 3, shifts from a trailing to a leading position. It interacts with its wake, and, as a consequence there is a plateau on the force curve. The force of the central cylinder approximately reaches its maximum.

$t=\pi$
The velocity is maximal as for $t=0$ but the cylinder 3 is leader. Its force is almost maximal but for a short delay between the maximum of velocity and the maximum of forces. For the central cylinder, the forces are also close to their maximum.

$t=3\pi/2$

The cylinder array reaches its maximal position; the velocity is equal to 0. The forces are decreasing for both cylinders.

$t=2\pi$

Same as for $t=0$.

The variation of the forces with the vorticity in the z-direction shows the same kind of evolution. At the time $t=\pi/2+\epsilon$, it is visible that the lead cylinders interact with residual vortices: this is a confirmation of the observations of the velocity in the x-direction. These vortices correspond to low pressure region of the pressure at the same time, see figure 14. The interaction with these vortices generates a local pressure increase, which increases the fluid force.

The pictures of the pressure over time show that the sign of the force can be easily deduced from the pressure difference between the left and the right side of the cylinders. The pressure difference between the right and the left side of the cylinder is higher for the lead cylinders than for the central or back cylinders. This explains why the maximal forces in the x-direction are observed for the lead cylinders.
Figure 12: Variation of forces and velocity in the x-direction with time
Figure 13: Variation of the forces in the x-direction and the vorticity in the z-direction with time
Figure 14: Variation of forces in the x-direction and pressure with time
5. Determination of the mass and drag coefficients with Morison expansion

5.1. Relevance of the Morison expansion to describe the forces in the x-direction

The Figure 15 and the Figure 16 show the forces on the x-direction and the Morison expansion associated for the cylinders 3 and 4. For the outer cylinders, the force is not symmetric and as a consequence the Morison expansion (which is a symmetric model) does not fit exactly to the force curve. However, the amplitude is reasonably well approximated. When it comes to the central cylinders, the Morison expansion gives a fair description of the forces in the x-direction, since these forces are symmetrical and almost sinusoidal.

Figure 15: Force in the x-direction for the cylinder 3 and Morison expansion associated

Figure 16: Force in the x-direction for the cylinder 4 and Morison expansion associated
5.2. Mass and drag coefficients

The Morison expansion provides a model of the fluid forces with an added mass and a damping term. The Morison coefficients are estimated for 18 simulations listed in the Table 5. Thereafter the influence of the four parameters studied on the mass and drag coefficients can be quantified.

<table>
<thead>
<tr>
<th># simulation</th>
<th>β</th>
<th>KC</th>
<th>Axial Re</th>
<th>Δx/D</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>162</td>
<td>12,56</td>
<td>37621</td>
<td>11,5</td>
</tr>
<tr>
<td>2</td>
<td>405</td>
<td>12,56</td>
<td>37621</td>
<td>11,5</td>
</tr>
<tr>
<td>3</td>
<td>405</td>
<td>6,28</td>
<td>37621</td>
<td>11,5</td>
</tr>
<tr>
<td>4</td>
<td>405</td>
<td>0,314</td>
<td>37621</td>
<td>11,5</td>
</tr>
<tr>
<td>5</td>
<td>405</td>
<td>25,12</td>
<td>37621</td>
<td>11,5</td>
</tr>
<tr>
<td>6</td>
<td>810</td>
<td>12,56</td>
<td>37621</td>
<td>11,5</td>
</tr>
<tr>
<td>7</td>
<td>405</td>
<td>12,56</td>
<td>17442</td>
<td>11,5</td>
</tr>
<tr>
<td>8</td>
<td>405</td>
<td>12,56</td>
<td>54566</td>
<td>11,5</td>
</tr>
<tr>
<td>9</td>
<td>1215</td>
<td>12,56</td>
<td>37621</td>
<td>11,5</td>
</tr>
<tr>
<td>10</td>
<td>405</td>
<td>12,56</td>
<td>37621</td>
<td>5,0</td>
</tr>
<tr>
<td>11</td>
<td>405</td>
<td>12,56</td>
<td>37621</td>
<td>3,0</td>
</tr>
<tr>
<td>12</td>
<td>243</td>
<td>12,56</td>
<td>37621</td>
<td>11,5</td>
</tr>
<tr>
<td>13</td>
<td>324</td>
<td>12,56</td>
<td>37621</td>
<td>11,5</td>
</tr>
<tr>
<td>14</td>
<td>648</td>
<td>12,56</td>
<td>37621</td>
<td>11,5</td>
</tr>
<tr>
<td>15</td>
<td>405</td>
<td>0,691</td>
<td>37621</td>
<td>11,5</td>
</tr>
<tr>
<td>16</td>
<td>405</td>
<td>0,195</td>
<td>37621</td>
<td>11,5</td>
</tr>
<tr>
<td>17</td>
<td>972</td>
<td>12,56</td>
<td>37621</td>
<td>11,5</td>
</tr>
<tr>
<td>18</td>
<td>405</td>
<td>2,512</td>
<td>37621</td>
<td>11,5</td>
</tr>
</tbody>
</table>

Table 5: List of the simulations with their parameters

The variation of the added mass and of the drag coefficients with the Keulegan-Carpenter number, the Stokes number, the axial Reynolds number and the confinement respectively, is given in the table of plots below. The objective is to determine the most influential parameters for the mass and for the damping term.
Comments on the variation of the mass and the drag coefficients (Figure 17):

The variation of the mass coefficient with the four parameters shows that the mass coefficient mainly depends on the Keulegan-Carpenter number and the Stokes number. There is only a small influence of the axial Reynolds number and the confinement. As the drag is expected to be similar for high amplitudes, the variation of the mass coefficient for high Keulegan-Carpenter numbers can only be explained by an influence of the axial velocity or a limit of the two-dimensional number.

The variation of the drag coefficient with the four parameters shows that there is a strong dependance on the Keulegan-Carpenter number. But for high Keulegan-Carpenter numbers, the Stokes number and the axial Reynolds number becomes significant parameters, whereas the confinement has only a small influence. As a consequence, the confinement in the x-direction has only a small influence for the range of value studied (relatively small effect of the confinement). The case of a higher confinement would surely show a higher influence of this parameter.

The high variations of the coefficient with the parameter studied, particularly the variation of the drag coefficient with the Keulegan-Carpenter number show that the Morison expansion is not adapted to describe the forces in the x-directions: that is to say that other terms have to be added to the expansion.
Figure 17: Mass and drag coefficients vs. Keulegan-Carpenter number, Stokes number, Confinement and Axial Reynolds number
6. Prospects

The simulations give converged solutions (spatially and for each time step) and now their validity and the model robustness ought to be verified.

6.1. Experimental validation

To validate the results of the simulations, experiments will be done in similar conditions. This will be done in the PhD study which will follow the master thesis. As a 2 dimension model is a rather harsh assumption, a first experiment will show if there are 3 dimensional effects are important or not.

6.2. Modelling robustness

The turbulence model used is a k-ε model: as a k-ω SST model is expected to give better results close to the wall, the test case has been run with both turbulence models. Figure 18 and Figure 19 show that results are similar except for the outer cylinders when they are in the lead position: in this case, the maximal amplitude 25% greater with the k-ε model than with the k-ω SST model since the k-ε model tends to overestimate the dissipation.
Moreover, the robustness of mesh has to be tested: as the size of the cells is here driven by the wall law, different size of cells should be tested with another wall law. Different cell sizes have to be tried in the wake.

6.3. A coherent temporal model for the forces in the x-direction
It would be useful to derive a time model of the force from the mass and the drag coefficients. To obtain a reliable model, the effect of the interaction of the lead cylinders with residual vortices has to be deeper studied as well as the effect of the confinement.

A crucial point in the future deals with the strong dependence of the drag force with the Keulegan-Carpenter number. Here difficulties to elaborate a time model of the force with such a strong dependency are expected, and maybe another type of expansion will be required.

The effects of the confinement in the x-direction have already been evaluated with some simulations but the effect of the confinement in the y-direction has not yet been studied. Moreover, as this confinement is created by the other assemblies in the core of the nuclear power plants, the confinement is changing during an earthquake. Hence, the case of a changing confinement during the cylinder motion should be studied.

6.4. Characteristic time of the axial velocity

When the array of cylinders is moving, the velocity in the z-direction does not follow the motion. The region where the cylinder was initially localized remains a low velocity region (Figure 20). Hence, it is possible to define a time constant for the velocity in the z-direction.
This can be done for instance by defining the characteristic time before the velocity in the z-direction reaches 95% of its final value when a pressure gradient is applied (with no motion of the array of cylinders). This characteristic time is about 1s but it depends on the confinement. To have more information about the characteristic time for the setting of the axial flow, further simulations will be necessary, especially with high confinement.

When the cylinder array is moving, the cylinders are going from low velocity regions to high velocity regions, especially for high displacement amplitudes. The cylinders are not surrounded by a constant axial velocity when they are moving, particularly for the outer cylinders. As the maximal velocity in the channel depends on the confinement, the case of high confinement also has to be studied. Moreover, as the outer cylinders are going through higher axial velocity regions than the inner cylinders, the effect of the axial velocity should be different for this type of cylinders. This may be the reason of the strong dependency of the drag coefficient with the Keulegan-Carpenter number. Further work is needed to assess this point.

The Figure 22 shows the turbulent energy for different instants. The repartition of the turbulent energy is mainly driven by the axial flow. The turbulent energy is especially higher for the regions with a high gradient of velocity in the z-direction. Further away from the array of cylinders, the turbulent energy is concentrated close to the walls, as usual in a channel flow.
Figure 21: Variation of the axial velocity vs. displacement
Figure 22: Variation of forces in the x-direction and turbulent energy with time
Conclusion

This master thesis consisted in a feasibility study of a fuel assembly seism design. The important aspect of this work was to quantify the influence of the degrees of freedom of the system and to define the different phenomena involved in the variation of the forces.

The added mass and the drag coefficients of the cylinders have been defined through the Morison expansion. They are depending on the seism features, that is the amplitude and the frequency but also on the parameters of the structure, the axial velocity and the confinement to a lesser extent. The added mass and the drag coefficients permit to build a force model which gives a rough approximation of the forces given by the simulations but the influence of the residual vortices with the lead cylinders has to be deeper studied.

These results require a validation both experimental and numerical (with a variation of the mesh size and the time step). The significant hypothesis of a two-dimension flow can be easily checked by experiments.
Bibliography


APPENDIX

Appendix 1: ustsns.F

This file is used to create an axial velocity. It is compiled at each new calculation with Code_Saturne.

C===================================================================
C 1. INITIALISATION
C===================================================================

IDEBIA = IDBIA0
IDEBRA = IDBRA0
IPP = IPPRTP(IVAR)
IF(IWARNI(IVAR).GE.1) THEN
  CHAINE = NOMVAR(IPP)
  WRITE(NFECRA,1000) CHAINE(1:8)
ENDIF
IPCROM = IPPROC(IROM (IPHAS))
C===================================================================

C===================================================================
C 2. EXAMPLE OF AN ARBITRARY SOURCE TERM FOR THE VARIABLE U :
C
C S = A * U + B
C
C WHICH APPEARS IN THE EQUATION TYPE :
C
C RHO VOLUME D(U)/Dt = VOLUME*S
CC
C THIS TERM HAS AN IMPLICIT PART : A
C AND A PART WHICH IS EXPLICIT : B
CC
C HERE FOR EXAMPLE :
C
C A = - RHO CKP
C B = QDM
C WITH
C CKP = 10.D0 [1/s ] (COEFFICIENT DE RAPPEL SUR U)
C QDM = 100.D0 [kg/(m2 s2)] (MOMENTUM PRODUCTION
C PER VOLUME UNITS AND PER TIME UNITS)
C
C SO,
C CRVIMP(IEL) = VOLUME(IEL)* A = - VOLUME(IEL) (RHO CKP )
C CRVEXP(IEL) = VOLUME(IEL)* B = VOLUME(IEL) (QDM )
C
C CRVIMP ET CRVEXP ARE DESCRIBED BELOW
C
C===================================================================

C BE CARREFUL, THIS EXAMPLE IS TOTALLY ARBITRARY
C ===========
C AND SHOULD BE REPLACED BY THE PROPER USER TERMS
C

C ----------------------------------------------

C CKP = 10.D0
C QDM = 100.D0
C
C DO IEL = 1, NCEL
C CRVIMP(IEL) = - VOLUME(IEL)*PROPCE(IEL,IPCROM)*CKP
C ENDDO
C
C DO IEL = 1, NCEL
C CRVEXP(IEL) = VOLUME(IEL)*QDM
C ENDDO
C
C--------
C FORMATS
C--------
C
C ********* INITIALISATION OF THE MODEL VARIABLES***
LTOT = 0.0102531914894D0 +0.0785319148936D0
LCARRE = 0.039D0
C radius of the cylinder
RAYON = 45.0D-4
C wetted perimeter of the section
PERINT = 2.0D0*(LTOT+LCARRE)+9*2.0D0*3.14D0*RAYON
C surface of the section
SINT = (LTOT*LCARRE)-9*3.14D0*RAYON*RAYON
C mean velocity wanted in the z-direction
UMOY = 5.D0
C volumic mass
RHO = 1000.0D0
C friction coefficient
COEFRI = 2.0D-2
C
C ****** CALCULATION OF THE AXIAL IMPULSE SOURCE*******
C if the iteration is in the z-direction

IF(IVAR.EQ.IW(IPHAS)) THEN
SOURCE = COEFRI*RHO*UMOY*UMOY/2.0D0*PERINT/4.0D0/SINT
WRITE(NFECRA,*) 'Source: ', SOURCE
DO IEL = 1, NCEL
CRVEXP(IEL) = VOLUME(IEL)*SOURCE
ENDDO
ENDIF

1000 FORMAT(' TERMES SOURCES UTILISATEURS POUR LA VARIABLE ',A8,/)
Appendix 2: article related to the master thesis
U-RANS SIMULATION OF FLUID FORCES EXERTED UPON AN OSCILLATING TUBE ARRAY IN AXIAL FLOW AT LARGE KEULEGAN-CARPENTER NUMBERS

L. Divaret\(^1\), P. Moussou\(^2\), O. Cadot\(^3\), J. Berland\(^4\), O. Doaré\(^3\)

\(^1\)EDF R&D, Analysis in Mechanics and Acoustics Dpt, Clamart, France
\(^2\)LaMSID UMR CNRS EDF 2832, Clamart, France
\(^3\)UME, ENSTA ParisTech, Palaiseau, France
\(^4\)EDF R&D, Fluid Dynamics, Power Generation and Environment Dpt, Chatou, France

lise.divaret@edf.fr

Abstract

This paper presents a CFD simulation of the fluid flow and of the fluid forces generated by large lateral displacements of cylinder arrays in the presence of an axial turbulent flow. A Morison-like expansion of the force consisting in an inertial term and a drag term is proposed, and the dependency of the dimensionless coefficients as functions of the Keulegan Carpenter number, of the Stokes number and of the axial Reynolds number are investigated.

Introduction

In the nuclear industry, the seismic design of reactor cores requires the structural analysis of fuel rod assemblies undergoing transient excitations. The fuel assemblies are classically described by vibrating beams, and the fluid is usually taken into account with the help of an added mass and an added damping coefficient incorporated into the structure model \([1]\). Such a representation is of current usage, but recent advances in CFD have made feasible thorough evaluation of the fluid forces nowadays.

In Power Water Reactor plants, fuel assemblies are constituted of stacks of 17 by 17 rods with a diameter \(D\) equal to about 9 mm, and a pitch to diameter ratio equal to about 1.33. The steady state axial fluid velocity reaches values equal to 5 m/s, which corresponds to an axial Reynolds number of the order of \(10^5\). In standard calculations, the amplitude of the lateral displacement of a fuel assembly during a seismic can reach a few diameters, and the corresponding frequencies range from 5 to 20 Hz. The velocity of a rod can hence reach values up to 2 m/s, which implies that the displacement-induced fluid flow is also turbulent. The dimensionless oscillation amplitude can be described by the so-called Keulegan-Carpenter (KG) number \([2]\), equal to \(2\pi X/D\), where \(X\) is the lateral rod displacement: in the framework of seismic analysis, the KG number varies from 0.1 to 20.

Fig. 1: Sketch of a fuel assembly, from the WNA website www.world-nuclear.org

The purpose of the present paper consists in evaluating the unsteady fluid forces exerted upon a rigid tube array moving laterally with KG numbers varying from 0.1 to 20. The system under study is kept as simple as possible, in order to facilitate the identification of the key parameters, and in order to better understand the physics of the fluid flow.

The choice of a controlled sine displacement is made to facilitate the analysis of the fluid dynamics. Such a representation cannot be used as it stands for seismic analysis, for non-linear terms in the fluid forces forbid to expand a transient displacement in Fourier terms.
However, this approach constitutes a first step in the understanding of fluid dynamics.

**Approach**

For the sake of efficiency, the simplest possible configuration is considered, namely, a 3 by 3 cylinder array inside a rectangular box in two dimensions. Such a simplified representation is to be used the following way: the central cylinder stands for the inner fuel rods, the four corner cylinders stand for the corner cylinders of a fuel assembly, and the four lateral cylinders stand for the outer rods of a fuel assembly. It is here assumed that the presence of a series of rods is screened by one single row of cylinders.

The cylinders have the same diameter as the actual fuel rods, i.e., 9 mm, and they are separated by gaps equal to 3 mm. A rectangular box is arranged around the cylinder array, with a gap equal to 2 mm in the y-direction, and a variable gap in the x-direction. The displacement of the cylinder array is a controlled sine oscillation with a frequency varying from 2 to 15 Hz and an amplitude varying from 0.05 to 4 diameters. An axial flow is imposed, with an average value inside the array varying from 2.5 to 7 m/s. The dimensionless values associated with these figures are given in Table 1: the case number 2 is chosen as reference.

As shown in Fig. 2, the fluid mesh around the cylinders is constituted of hexahedra only as required by the best practice in fluid dynamics, with a small width in the axial direction associated with one cell. The mesh generator GMSH [6] was used to generate it; it exhibits 19560 cells. The skew ratio of the cells is always lower than 0.5, and the maximum edge ratio is equal to 1.9.

Unsteady Reynolds Averaged Navier Stokes (URANS) simulation is performed with the help of the finite volume fluid dynamics solver Code_Saturne [7] using the Arbitrary Lagrangian Eulerian method (see for instance [8] for a detailed description of the ALE). The choice was made to use a k-ε model with a standard two-scale law.

A 2D simulation is performed by applying periodic conditions in the axial direction, i.e., by forcing the upper and lower velocity vectors to coincide. The net flow in the axial direction is generated by the addition of a momentum source term to the Navier-Stokes equations, which compensates the friction force along the cylinder walls and the box walls. Such a procedure ensures a two-dimensional solution of the fluid dynamics equations, without forcing the velocity nor the pressure at the mesh boundaries.

The ALE method is applied in its simplest form: the displacement of the boundaries of the fluid domain are prescribed, and the mesh is automatically compressed and expanded with the help of a Laplacian algorithm which keeps the mesh almost unchanged close to the cylinders.

The calculations are performed with an 8 processor Linux station: in the reference case, 20000 initial time steps of 0.2 ms are first dedicated to the stabilization of the axial flow pattern, followed by a smooth transition to a final series of controlled oscillations of the cylinder array during 15000 time steps. The Courant-Friedrichs Lewy (CFL) number obtained is lower than 4.

### Flow pattern and unsteady forces

Visualization of the fluid flow are achieved with the help of the free software Paraview [9]. The unsteady forces obtained during the calculations are illustrated in the reference case by the following figures, where the fluid velocity in the x-direction and the axial component of the vorticity are plotted. The x-direction goes from left to right in the Figs, the y-direction form bottom to top and the z-direction is the axial direction, perpendicular to the Figs.

<table>
<thead>
<tr>
<th>Case</th>
<th>Keulegan-Carpenter</th>
<th>Stokes</th>
<th>Axial Reynolds</th>
<th>Confinement</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12.56</td>
<td>162</td>
<td>37622</td>
<td>11.5</td>
</tr>
<tr>
<td>2</td>
<td>12.56</td>
<td>405</td>
<td>37622</td>
<td>11.5</td>
</tr>
<tr>
<td>3</td>
<td><strong>6.28</strong></td>
<td>405</td>
<td>37622</td>
<td>11.5</td>
</tr>
<tr>
<td>4</td>
<td>0.31</td>
<td>405</td>
<td>37622</td>
<td>11.5</td>
</tr>
<tr>
<td>5</td>
<td><strong>25.12</strong></td>
<td>405</td>
<td>37622</td>
<td>11.5</td>
</tr>
<tr>
<td>6</td>
<td>12.56</td>
<td><strong>810</strong></td>
<td>37622</td>
<td>11.5</td>
</tr>
<tr>
<td>7</td>
<td>12.56</td>
<td>405</td>
<td><strong>17442</strong></td>
<td>11.5</td>
</tr>
<tr>
<td>8</td>
<td>12.56</td>
<td>405</td>
<td><strong>54566</strong></td>
<td>11.5</td>
</tr>
<tr>
<td>9</td>
<td>12.56</td>
<td><strong>1215</strong></td>
<td>37622</td>
<td>11.5</td>
</tr>
<tr>
<td>10</td>
<td>12.56</td>
<td>405</td>
<td>37412</td>
<td>5</td>
</tr>
<tr>
<td>11</td>
<td>12.56</td>
<td>405</td>
<td>37320</td>
<td>3</td>
</tr>
<tr>
<td>12</td>
<td>12.56</td>
<td><strong>243</strong></td>
<td>37622</td>
<td>11.5</td>
</tr>
<tr>
<td>13</td>
<td>12.56</td>
<td><strong>324</strong></td>
<td>37622</td>
<td>11.5</td>
</tr>
<tr>
<td>14</td>
<td>12.56</td>
<td><strong>648</strong></td>
<td>37622</td>
<td>11.5</td>
</tr>
<tr>
<td>15</td>
<td><strong>0.69</strong></td>
<td>405</td>
<td>37622</td>
<td>11.5</td>
</tr>
<tr>
<td>16</td>
<td><strong>0.2</strong></td>
<td>405</td>
<td>37622</td>
<td>11.5</td>
</tr>
<tr>
<td>17</td>
<td>12.56</td>
<td><strong>972</strong></td>
<td>37622</td>
<td>11.5</td>
</tr>
<tr>
<td>18</td>
<td><strong>2.51</strong></td>
<td>405</td>
<td>37622</td>
<td>11.5</td>
</tr>
</tbody>
</table>

Table 1: flow and oscillation conditions
A simple flow pattern is observed, with flow separation downstream of each cylinder and the formation of a wake behind the last row of cylinders. The unsteady vortices in the wake appear to vanish quite quickly, most probably because of the turbulent axial flow. It can be also be observed that the leading cylinders at the beginning of the backwards displacement interact with the residual vortices as shown in the fourth subfigure, which corresponds to an intermediate time shortly after the maximum displacement.

The fluid forces exerted upon the cylinders exhibit a sine-like shape, with a phase difference with the array displacement equal to about $-0.95$ in the reference case illustrated in Fig. 5 and 6. This phase shift indicates that the drag force dominates in the range of displacement amplitude and of fluid velocities considered, and that inertial effect are also present. Such a description overlooks unsteady effects like the interaction of cylinders with the wake vortices, but it is meaningful as a first approximation.
Axial velocity dynamics

Fig. 5: evolution of the x-component of the force exerted upon the upper center cylinder during one cycle of oscillation, for a displacement of the array proportional to \( \sin 2\pi ft \). Black line: result of the computation of the reference case, gray line: Morison-like fit of the result of the computation (see next Section )

Fig. 6: evolution of the x-component of the force during one cycle of oscillation for the central cylinder for a displacement of the array proportional to \( \sin 2\pi ft \). Black line: result of the computation of the reference case, gray line: Morison-like fit of the result of the computation (see next Section )

Fig. 7: sketch of the axial velocity along one cycle of oscillation for the reference case, in \( s^{-1} \)

It is worth mentioning the fact that the axial velocity does not adapt instantaneously to the position of the tube array, as illustrated in Fig. 7: the axial velocity in the wake of the cylinders remains close to zero during a part of the oscillation cycle. This results probably from axial inertia combined to the vanishing velocity condition along the cylinder walls.
Inertia and drift coefficients

A simple attempt to fit the computed forces can be based on the Morison expansion originally derived for single cylinders in quiescent flow \[10\], namely:

\[
F_{x,\text{Morison}} = -\alpha_m \frac{\pi \rho D^2}{4} \dot{x} - \alpha_d \frac{\rho D}{2} \left| \ddot{x} \right| \dot{x}
\]

where \(\alpha_m\) is the dimensionless coefficient of the added mass term, assumed to be proportional to the acceleration of the cylinder. In the above expression, the drag term follows a quasi-steady law, and unsteady effects are overlooked. Nevertheless, such an expression has the merit of simplicity, and as the term \(X|\ddot{x}|\) is reasonably close to a sine, the identification of a couple of dimensionless coefficients \(\alpha_m\) and \(\alpha_d\) is straightforward.

Following this line of reasoning, it is proposed to expand the \(x\)-forces in Morison-like coefficients according to

\[
F_x = \omega^2 \left( \alpha_m \frac{\pi \rho D^2}{4} \sin \omega t A - \alpha_d \frac{\rho D}{2} \cos \omega t \cos \omega t A^2 \right)
\]

where \(\omega\) and \(A\) are the circular frequency and the amplitude of the oscillation, respectively. The coefficients obtained that way are still related to the drag and to the fluid inertia, but they may depend on the oscillation parameters, especially the frequency and the amplitude. Such an expansion cannot cover all the details of the fluid flow, and presumably not the interaction of the cylinder array with the axial flow. It is used in the present study because it the simpler expansion that can be proposed.

The coefficients obtained throughout least square estimations for four out of nine cylinders are plotted in the next figures as functions of parameters listed in Table 1, i.e., the dimensionless amplitude (KG), the dimensionless frequency \(fD^2/\nu\), a.k.a. Stokes number, and the axial Reynolds number. The width of the box was not found to have a significant influence on the coefficients in the range of widths tested (see Table 1).

It appears that the mass and the drag coefficients obtained that way are, as an order of magnitude, close to unity. These values are consistent with the classical values of the added mass of a single cylinder estimated by a potential flow model (\(\alpha_m = 1\), see for instance \([11]\)) and with the drag coefficient of a cylinder which is close to unity in a wide range of Reynolds numbers (\(\alpha_d \sim 1\), see for instance \([12]\)).

As can be seen in Fig. 8, a strong dependency of the drag coefficient with the Keulegan Carpenter number is observed. The strong increase of \(\alpha_d\) for small values of KC seems to indicate that the scale laws of the above equations may require some further elaboration.

- Fig. 8: mass and drag coefficients vs. Keulegan Carpenter number. •: center cylinder, <: left central cylinder, ▲: central upper cylinder, ●: upper left cylinder.
Fig. 9: mass and drag coefficients vs. Stokes number

- : center cylinder,
- : left central cylinder,
- : central upper cylinder,
- : upper left cylinder

Perspectives

A CFD calculation of the fluid forces exerted upon a 3 by 3 array of cylinders with large Keulegan Carpenter could be achieved with URANS and ALE. Further work is needed to experimentally validate the principle of this calculation, and special attention should be given to the two dimensional nature of the fluid flow. Another aspect of the approach deals with the nature of the confinement; one effects the alpha coefficients to significantly be altered if the surrounding box was opened on both ends. Further calculation are required to investigate it.

Nevertheless, the approach of the present paper constitutes a first attempt to grab the essential features of large lateral displacements of cylinder arrays in the presence of axial flow.

Fig. 10: mass and drag coefficients vs. axial Reynolds number

- : center cylinder,
- : left central cylinder,
- : central upper cylinder,
- : upper left cylinder
**Acknowledgements**

The authors express their gratitude to Miss Marie Clement for paving the way to the different CFD tools used in the present study.

**References**


