Benchmark
on the numerical simulation
of a tube bundle vibration under cross flow

Fabien Huvelin (University of Lille & EDF R&D)

Marcus Vinicius Girao de Morais (University of Cergy-Pontoise & CEA)
Franck Baj (CEA)
Jean-Paul Magnaud (CEA)
Elisabeth Longatte (EDF R&D)
M’hamed Souli (University of Lille)
INTRODUCTION

CREATIF Program :

Comprehension of the vibratory Response of an Array of Tubes in Interaction with a Fluid.
Experiment AMOVI (CEA)

- 24 fixed tubes (Plexiglas, f>300Hz)
- 1 moving tube (steel, f=14.3Hz)
- P/D = 1.44
- Single phase flow (flow rate from 0 to 5 m³.h⁻¹)

contact: fbaj@cea.fr
<table>
<thead>
<tr>
<th>Contents</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 – Methodology</td>
</tr>
<tr>
<td>2 – Real case (2D model)</td>
</tr>
<tr>
<td>3 – Numerical case (2D model)</td>
</tr>
<tr>
<td>4 – Real case (3D model)</td>
</tr>
</tbody>
</table>
Methodology
1.1 Boundary conditions

Boundary conditions at fluid-structure interface

\[
\begin{align*}
\mathbf{u}_s &= \mathbf{u}_f \\
\sigma_s \cdot \mathbf{n} &= \sigma_f \cdot \mathbf{n}
\end{align*}
\]

Partitioned coupling

Coupling scheme

\[
\begin{align*}
\text{LOAD} & \quad \text{FLUID SOLVER} \\
\text{DISPLACEMENT/VELOCITY} & \quad \text{STRUCTURE SOLVER}
\end{align*}
\]
1.2 Tools

Structure solver (rigid body)

- 1 degree of freedom oscillator for EDF tool
- Finite element method with Lagrangian formulation for CEA tool

Fluid solver

- Finite volume method with Eulerian formulation (EDF tool)
- Finite element method with Eulerian formulation (CEA tool)
  ⇒ Use of an ALE technique to follow the structure interface (both cases)

Partitioned procedure

- Fluid solver needs structure data at time $t^{n+1}$
- Structure solver needs fluid data at time $t^{n+1}$
  ⇒ Prediction: the position of the structure interface at time $t^{n+1}$
1.3 Improved serial staggered procedure

**METHODOLOGY | REAL CASE (2D MODEL) | NUMERICAL CASE (2D MODEL) | REAL CASE (3D MODEL)**

Initialization

Prediction of the fluid structure interface

Fluid solver
Mesh updating

Prediction of the fluid force acting on the structure

Structure solver

\[ X_{fluid}^{pred,n+1} = X_{structure}^n + \frac{\Delta t}{2} V_{structure}^n \]

\[ F_{structure}^{pred,n+1} = 2F_{fluid} - F_{structure}^{pred,n} \]

*(Piperno et al., 2001)*
Real case (2D model)
2.1 Geometry (experiment AMOVI, CEA)

INLET FLOW | WALL | WALL | FREE OUTLET

METHODOLOGY | REAL CASE (2D MODEL) | NUMERICAL CASE (2D MODEL) | REAL CASE (3D MODEL)

MOVING TUBE
2.1 Geometry (experiment AMOVI, CEA)

METHODOLOGY | REAL CASE (2D MODEL) | NUMERICAL CASE (2D MODEL) | REAL CASE (3D MODEL)
### 2.2 Parameters

#### METHODOLOGY | REAL CASE (2D MODEL) | NUMERICAL CASE (2D MODEL) | REAL CASE (3D MODEL)

<table>
<thead>
<tr>
<th>Structure properties</th>
<th>Fluid properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Natural frequency (Hz)</td>
<td>Density (kg·m⁻³)</td>
</tr>
<tr>
<td>Natural damping (%)</td>
<td>Dynamic viscosity (kg·m⁻¹·s⁻¹)</td>
</tr>
<tr>
<td>Mass (kg·m⁻¹)</td>
<td></td>
</tr>
<tr>
<td>Diameter (mm)</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>P/D (-)</th>
<th>1.44</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet velocity (m·s⁻¹)</td>
<td>[0.03 ; 0.15]</td>
</tr>
<tr>
<td>Reduced velocity (-)</td>
<td>[0.5 ; 2.8]</td>
</tr>
<tr>
<td>Reynolds number (-)</td>
<td>[1200;6000]</td>
</tr>
</tbody>
</table>
### 2.3 Quiescent fluid: water frequency

**METHODOLOGY | REAL CASE (2D MODEL) | NUMERICAL CASE (2D MODEL) | REAL CASE (3D MODEL)**

<table>
<thead>
<tr>
<th>Water frequency (Hz)</th>
<th>CEA</th>
<th>EDF</th>
<th>Analytical</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>11.63</td>
<td>11.55</td>
<td>11.8</td>
</tr>
</tbody>
</table>
2.4 Quiescent fluid: water damping

### METHODOLOGY | REAL CASE (2D MODEL) | NUMERICAL CASE (2D MODEL) | REAL CASE (3D MODEL)

<table>
<thead>
<tr>
<th>Water damping (%)</th>
<th>CEA</th>
<th>EDF</th>
<th>Analytical</th>
</tr>
</thead>
<tbody>
<tr>
<td>extrapolation</td>
<td>1.216</td>
<td>1.215</td>
<td>1.18</td>
</tr>
</tbody>
</table>
2.5 Vibration under cross-flow

METHODOLOGY | REAL CASE (2D MODEL) | NUMERICAL CASE (2D MODEL) | REAL CASE (3D MODEL)

Re = 1200
Re = 3000
Re = 4800

• Numerical problem ?
• Physical problem ?
Numerical case (2D model)
### Structure properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Natural frequency</td>
<td>Hz</td>
<td>2.5</td>
</tr>
<tr>
<td>Natural damping</td>
<td>%</td>
<td>0.0437</td>
</tr>
<tr>
<td>Mass</td>
<td>kg</td>
<td>0.298</td>
</tr>
<tr>
<td>Diameter</td>
<td>mm</td>
<td>10.00</td>
</tr>
</tbody>
</table>

### Fluid properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>kg.m(^{-3})</td>
<td>10^3</td>
</tr>
<tr>
<td>Dynamic viscosity</td>
<td>kg.m(^{-1}.s^{-1})</td>
<td>10^{-3}</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Property</th>
<th>Unit</th>
<th>Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>P/D</td>
<td>(-)</td>
<td>1.44</td>
</tr>
<tr>
<td>Inlet velocity</td>
<td>m.s(^{-1})</td>
<td>[0.001 ; 0.035]</td>
</tr>
<tr>
<td>Reduced velocity</td>
<td>(-)</td>
<td>[0.1 ; 4.5]</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>(-)</td>
<td>[30 ; 1200]</td>
</tr>
</tbody>
</table>
3.2 Displacement

METHODOLOGY | REAL CASE (2D MODEL) | NUMERICAL CASE (2D MODEL) | REAL CASE (3D MODEL)

\[ v_{inlet} = 0.001 \text{ m.s}^{-1} \]
\[ v_{red} = 0.13 (-) \]

\[ v_{inlet} = 0.02 \text{ m.s}^{-1} \]
\[ v_{red} = 2.62 (-) \]

\[ v_{inlet} = 0.03 \text{ m.s}^{-1} \]
\[ v_{red} = 3.93 (-) \]
3.3 Frequency and damping

METHODOLOGY | REAL CASE (2D MODEL) | NUMERICAL CASE (2D MODEL) | REAL CASE (3D MODEL)
Real case (3D model)
4.1 Geometry

- 7.5 Millions of elements
- 2048 processors
4.2 Preliminary results

METHODOLOGY | REAL CASE (2D MODEL) | NUMERICAL CASE (2D MODEL) | REAL CASE (3D MODEL)

Reynolds = 3000 and DNS

Displacement with 2D simulation

Displacement with 3D simulation
4.3 3D effect

Pressure

Velocity

3D effects
CONCLUSION

✓ Catch the 3 behaviors of the structure
✓ Used of an extrapolation method for the damping
✓ Over-estimation of force fluid with 2D simulation (3D simulation is required)

➢ 3D simulation for numerical case
➢ 3D simulation with turbulent model for real case
➢ Comparison of the real case to the experiment AMOVI (CEA)