Fluid-structure interaction with OpenFSITM and MD NastranTM structural solver

Robert Sandboge*
MSC Software Corporation
2300 Traverwood Drive
Ann Arbor, MI 48105
USA
March 5, 2010

Contents

1 Introduction 1
2 The basic FSI problem setup 2
2.1 Problem definition 2
2.2 Simulation 2
3 Analysis method 2
3.1 Coupling procedure 2
3.2 Fluid-structure Interface 2
3.3 Fluid-structure dynamics formulation 3
4 FSI example demonstrating self-induced structural oscillations 3
4.1 Problem formulation 3
4.2 Computational results 4
4.2.1 FSI2 simulation 5
4.2.2 FSI3 simulation 6
4.2.3 Comparison 7
5 FSI example demonstrating flutter prediction 7
5.1 Problem formulation 7
5.2 Computational results 7
6 Conclusion 8

1 Introduction

Mechanical structures don’t normally exist and operate in a vacuum. A structure is normally surrounded by a fluid like air or water, and generally the design of the mechanical system must take the potentially moving fluid into consideration. The

OpenFSI interface is the mechanism that allows for the analysis of interaction between fluid and structure with the MD NastranTM solver. Examples of Fluid-Structure Interaction (FSI) systems are for instance the flexing of airplane wings of commercial jets in flight, dynamics of oceangoing ships, bending and durability of wind-turbine blades, and the movement of a flag in a breeze, see Figure 1. To simulate FSI problems, the analysis of the fluid flow is performed by a Computational Fluid Dynamics (CFD) solver, the structural behavior is computed by the MD Nastran solver, and the dynamic exchange of the fluid forces and structural displacements is done using the OpenFSI interface.

Figure 1: Animation of flag in a breeze, based on OpenFSI simulation.
2 The basic FSI problem setup

2.1 Problem definition

A narrow duct with fluid flow exerting forces on a flexible baffle is used to illustrate the setup of a simple FSI problem, see Figure 2. The flow profile is specified at the inlet, and the floor and baffle has no slip boundary conditions. The flexible baffle is pinned to the floor, but has a small clearance to the side walls.

2.2 Simulation

The CFD code (AcuSolve® CFD solver [1] is used in the examples in this note) computes the forces exerted on the baffle, and MD Nastran computes the resulting displacement of the baffle. The force–displacement data is communicated for each time step in the simulation, and results in the dynamic behavior shown in Figure 3.

3 Analysis method

3.1 Coupling procedure

Solution data is exchanged between MD Nastran and the CFD code during the simulation via the OpenFSI interface dynamically, see Figure 4.

Force data from the CFD code is received by MD Nastran, and the computed structural displacement and velocity data is sent back to the CFD code. The data is communicated on nodes that belong to wetted surfaces, which are defined in the MD Nastran input deck as surface meshes of triangles and quadrilaterals. In addition, the data may be communicated on nodes that belong to 1D elements, such as beams. The data is transferred once or several times (depending on the type of FSI analysis) for each time step in a nonlinear MD Nastran solution sequence.

The procedure for the data exchange is handled by one or more simulation architecture components, which are defined specifically for each external CFD solver. The user may chose among existing components matching certain commercial external codes, or build one from scratch using the MD Nastran solver software development kit.

3.2 Fluid-structure Interface

The coupling interface is provided at two different levels within the structural solver:

i. The force vectors are received at the beginning of a time step and displacement and velocity vectors are sent at the end of the time step.

ii. Optionally the force vectors and displacement and velocity vectors are exchanged within the iterative (Newton) loop.

Essentially, the coupling at the first level implies an explicit coupling and by adding a coupling at
the second level allows for an implicit (and tighter) coupling, see Figure 5.

The most common explicit coupling scheme is the Conventional Serial Staggered (CSS) approach, see e.g. [2, 3]. To achieve better stability properties, e.g. for problems where the fluid and structural densities are close, an implicit scheme may be appropriate.

### 3.3 Fluid-structure dynamics formulation

Different structural dynamics formulations can be chosen in MD Nastran, where the choice depends on e.g. the material properties, load characteristics and if large rotations and deformations need to be modeled. Using the standard material model and considering large displacements, the structural model is formulated in a Lagrangian corotational framework, which allows for arbitrarily large displacements and rotations but the strain must be small. In the corotational framework, the motion of the structure is composed of a rigid body motion defining the co-rotated configuration and a deformation motion that references the co-rotated configuration.

The fluid dynamics formulation of the external code is independent on the formulation used in MD Nastran OpenFSI simulations, except that the data communication must match the chosen scheme in the OpenFSI simulation: The explicit FSI scheme or an implicit FSI scheme.

### 4 FSI example demonstrating self-induced structural oscillations

#### 4.1 Problem formulation

To demonstrate strongly coupled FSI systems, a benchmark model proposed by S. Turek and J. Hron [10] is analyzed. The model has the simple geometrical configuration shown in Figures 6–7 and the parameter configuration defining the FSI1, FSI2 and FSI3 tests are shown in Tables 1–2.

![Figure 6: Computational domain. The fluid flows from left to right around the structural part. Origin (0, 0) is located in the lower left corner.](image-url)
Figure 7: Solid structure part.

<table>
<thead>
<tr>
<th>Feature</th>
<th>Parameter [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Channel length</td>
<td>( L = 2.5 )</td>
</tr>
<tr>
<td>Channel width</td>
<td>( H = 0.41 )</td>
</tr>
<tr>
<td>Cylinder center</td>
<td>( C = (0.2, 0.2) )</td>
</tr>
<tr>
<td>Cylinder radius</td>
<td>( r = 0.05 )</td>
</tr>
<tr>
<td>Bar structure length</td>
<td>( l = 0.35 )</td>
</tr>
<tr>
<td>Bar structure thickness</td>
<td>( h = 0.02 )</td>
</tr>
<tr>
<td>Bar structure reference point at ( t = 0 )</td>
<td>( A = (0.6, 0.2) )</td>
</tr>
</tbody>
</table>

Table 1: Geometrical parameters for bar structure behind cylinder model.

The fluid velocity profile is prescribed at the inlet \( x = 0 \) as

\[
    u_{\text{inlet}}(y) \equiv u(x = 0, y) = 1.5 \cdot \bar{U} \frac{y(H - y)}{(H/2)^2}, \tag{1}
\]

with a mean inflow velocity \( \bar{U} \). At the outlet, \( x = L = 2.5m \), a pressure boundary condition is used

\[
    p_{\text{outlet}} \equiv p(x = L, y) = 0, \tag{2}
\]

where \( H = 0.41m \) is the height of the channel.

The initial condition of the flow field is computed from a fluid dynamics simulation where the structure is assumed to be rigid.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>FSI1</th>
<th>FSI2</th>
<th>FSI3</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \frac{\rho_s}{\rho_f} )</td>
<td>1</td>
<td>10</td>
<td>1</td>
</tr>
<tr>
<td>( \nu_s )</td>
<td>0.4</td>
<td>0.4</td>
<td>0.4</td>
</tr>
<tr>
<td>( \mu_s )</td>
<td>10^6 kg/m^2</td>
<td>0.5</td>
<td>0.5</td>
</tr>
<tr>
<td>( \beta )</td>
<td>( \frac{\rho_s}{\rho_f} )</td>
<td>1</td>
<td>10</td>
</tr>
<tr>
<td>( \rho_f )</td>
<td>10^3 kg/m^3</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>( \nu_f )</td>
<td>( 10^{-3} \text{m}^2 \text{s}^{-1} )</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>( \bar{U} )</td>
<td>( \text{m/s} )</td>
<td>0.2</td>
<td>1</td>
</tr>
<tr>
<td>( Re = \frac{\bar{U}d}{\nu_f} )</td>
<td>20</td>
<td>100</td>
<td>200</td>
</tr>
</tbody>
</table>

Table 2: FSI parameters for bar structure behind cylinder model.

As above, the AcuSolve® CFD solver [1] is coupled with MD Nastran, and an Arbitrary Lagrangian Eulerian (ALE) (see e.g. [4,5,6]) formulation is used to match the computational CFD mesh with the displaced structural mesh. An implicit FSI scheme is chosen as the density ratio \( \beta \) between the structure and fluid is equal or close to one (see Table 2). The forces supplied via the OpenFSI interface are ramped up during the first second. The model size presented here is \( \sim 1.5 \cdot 10^5 \) degrees of freedom (including mesh displacement).

4.2 Computational results

Dynamic FSI simulations solving the FSI2 and FSI3 configurations are presented (FSI1 is a steady state configuration). The solutions show good stability for the chosen numerical scheme, and fully convergent residuals were reached for the fluids solver, the structures solver and the coupled implicit OpenFSI scheme.

Note that the geometry is slightly non-symmetric in the \( y \)-direction, so that fully symmetric solutions are not expected.
4.2.1 FSI2 simulation

The solution from the fluid dynamics simulation is transient with an oscillatory flow field behavior, see Figures 8–9. The initial condition to the FSI simulation is evaluated at a stage where the oscillatory cycles of the integrated traction are stable. As the FSI forces are ramped up, the bar behind the cylinder starts to slowly fluctuate up and down. After \( \sim 7 \) s the oscillatory cycles maintain their appearance.

Figure 8: FSI2 animation of flow around cylinder with bar structure. The cutting plane displays the magnitude of velocity \(|v|\).

Figure 9: FSI2 animation of flow around cylinder with bar structure. The cutting plane displays the pressure \( p \).

The displacement of point \( A \) for FSI2 is shown in Figures 10–11 for a simulation using time step \( \Delta t = 0.002 \) s.

Figure 10: FSI2: Plot of \( x \)-displacement of point \( A \).

Figure 11: FSI2: Plot of \( y \)-displacement of point \( A \).
4.2.2  FSI3 simulation

Like the FSI2 simulation, the initial condition is evaluated from a fluid dynamics transient solution. With the density ratio $\beta = 1$, the solid structure is strongly coupled with the fluid flow, and the bar quickly reaches a stage where the cycles are steady, see Figures 12–13. Within a time step, several FSI iterations are necessary to reach convergence, which makes this problem computationally intensive.

Figure 12: FSI3 animation of flow around cylinder with bar structure. The cutting plane displays the magnitude of velocity $|v|$.

The displacement of point $A$ for FSI3 is shown in Figures 14–15 for a simulation using time step $\Delta t = 0.001s$.

Figure 13: FSI3 animation of flow around cylinder with bar structure. The cutting plane displays the pressure $p$.

Figure 14: FSI3: Plot of $x$–displacement of point $A$.

Figure 15: FSI3: Plot of $y$–displacement of point $A$. 
4.2.3 Comparison

Comparative test results for the simulations are summarized in Table 3, and agree with published computational results in [10].

<table>
<thead>
<tr>
<th>Quantity</th>
<th>FSI2</th>
<th>FSI3</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d_x$ [mm]</td>
<td>$-14.33 \pm 12.27$</td>
<td>$-2.83 \pm 2.78$</td>
</tr>
<tr>
<td>$f$ [Hz]</td>
<td>3.85</td>
<td>10.76</td>
</tr>
<tr>
<td>$d_y$ [mm]</td>
<td>$1.28 \pm 79.63$</td>
<td>$1.35 \pm 34.75$</td>
</tr>
<tr>
<td>$f$ [Hz]</td>
<td>1.96</td>
<td>5.38</td>
</tr>
<tr>
<td>$t_x$ [N]</td>
<td>$210.0 \pm 73.5$</td>
<td>$458.5 \pm 24.0$</td>
</tr>
<tr>
<td>$f$ [Hz]</td>
<td>3.95</td>
<td>10.76</td>
</tr>
<tr>
<td>$t_y$ [N]</td>
<td>$1.5 \pm 233.5$</td>
<td>$2.5 \pm 147.5$</td>
</tr>
<tr>
<td>$f$ [Hz]</td>
<td>1.93</td>
<td>5.38</td>
</tr>
</tbody>
</table>

Table 3: FSI displacement ($d_x, d_y$) and integrated traction ($t_x, t_y$) (of the entire solid structure) comparison for bar structure behind cylinder model.

5 FSI example demonstrating flutter prediction

5.1 Problem formulation

This example is based on the ha145 model from [9]. Air flows over a swept-back plate-like wing, and the question is at what air speed a critical flutter condition is observed. In a series of tests, the inlet velocity is gradually increased until the flutter condition reaches a stage where a steady flutter amplitude is obtained. To study the dynamics of a super critical flutter condition, where the risk for catastrophic failure is imminent, the inlet air speed is increased further.

5.2 Computational results

Results from three different inlet air speeds are presented, where sub-critical, critical and super critical flutter conditions are obtained. The control volume for the analysis is shown in Figure 16.

Figure 16: The flutter control volume. The wing is attached to the yellow wall. The arrow indicates the inflow surface.

The dynamic FSI simulations start from initial conditions where the structure is held rigid, and a steady air flow with a small perturbation in dynamic pressure above and below the wing is specified. To confirm consistency, additional simulations are performed where the dynamic pressure perturbation of the initial condition is large enough.
to cause the wing to deflect at a higher amplitude than the critical flutter amplitude. Table 4 shows an overview of the simulations presented here.

<table>
<thead>
<tr>
<th>Mach</th>
<th>Perturbation</th>
<th>Behavior</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.455</td>
<td>small</td>
<td>sub-critical</td>
</tr>
<tr>
<td></td>
<td>large</td>
<td>sub-critical</td>
</tr>
<tr>
<td>0.47</td>
<td>small</td>
<td>critical</td>
</tr>
<tr>
<td></td>
<td>large</td>
<td>critical</td>
</tr>
<tr>
<td>0.5</td>
<td>small</td>
<td>super-critical</td>
</tr>
<tr>
<td></td>
<td>large</td>
<td>super-critical</td>
</tr>
</tbody>
</table>

Table 4: Flutter test simulations to predict flutter.

The vertical displacement at the rear tip of the wing is monitored to show the flutter behavior in Figure 17. A steady amplitude is reached for inlet speed 0.47 Mach at either perturbation level indicating a critical flutter behavior. Figures 18–19 show the dynamics of the critical flutter behavior at speed 0.47 Mach.

6 Conclusion

The objective of the OpenFSI interface is to provide an interface between the MD Nastran SOL 400 solution sequence with an external CFD code to solve FSI problems. The technology allows for a looser explicit coupling, and a tighter iterative implicit coupling, that can be used to solve a wide variety of problems with different complexity levels, efficiently. The MD Nastran user, or a CFD vendor, can program a tailored service component using the solver SDK that handles the data conversion between the OpenFSI interface and the external CFD code.
References


