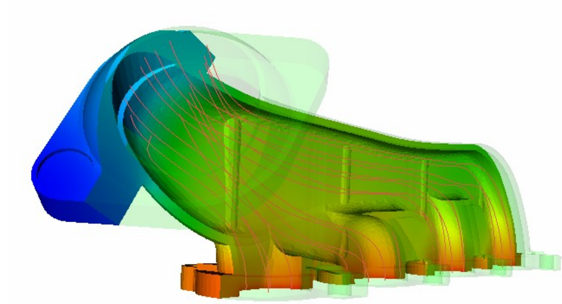
Fluid Structure Interaction Analysis with Abaqus and FLUENT

Summary

Engineering problems that involve the coupled response of a flowing fluid and a deforming structure constitute a broad class referred to as fluid-structure interaction (FSI). The interaction can be mechanical, thermal, or both. Many important problems involve some form of FSI, but the coupling effect is often ignored because of a lack of readily available solution technology. To address this limitation, Dassault Systèmes SIMULIA Corp. and Fluent, Inc. have partnered to provide a coupled solution capability.

The co-simulation capability, available beginning in Abaqus Version 6.5, can be used in a wide variety of applications such as thermal-stress prediction in exhaust manifolds, design of valves and rubber diaphragms that restrict fluid flow, micro-fluidics, in-vivo vascular flows, consumer goods design, and packaging applications. In this technology brief we present several problems from different industries that have been analyzed with the Abaqus and FLUENT FSI capability.



** Plotted in EnSight Gold from CEI*

Key Abaqus Features and Benefits

- Co-simulation capability in Abaqus/Standard and Abaqus/Explicit for fluid-structure interaction applications.
  - Uses dedicated tools (Abaqus and FLUENT) to solve the respective subdomain problems.
  - Coupled analyses can be steady-state or transient, two-dimensional or three-dimensional, with unidirectional or bidirectional coupling of the subdomains.
  - Thermo-mechanical coupling of the structural and fluid subdomains through a general code coupling interface (MpCCI from Fraunhofer SCAI).
- A rich set of Abaqus features are available in coupled problems including contact, geometric and material nonlinearity, and other capabilities.

Background

A fluid-structure interaction (FSI) problem can be broadly defined as one in which the responses of a moving fluid and a structure are coupled. Well established analytical techniques, collectively referred to as the Finite Element Method (FEM) for structures and Computational Fluid Dynamics (CFD) for fluids, exist for simulating the individual domains of a coupled problem. In the co-simulation approach the two techniques are used together in a synchronized fashion to exchange solution information between the two analysis tools.

FSI problems fall under two broad categories. In the first category the interaction between the fluid and the structural fields is small. These problems can be solved sequentially by performing the CFD analysis first and then mapping the results to the FEM analysis. The second category involves problems in which the fluid-structure interaction is not negligible and the overall solution for both the structural and fluid domains must be determined simultaneously.

FSI Solution Approach

The Abaqus-FLUENT FSI capability uses a staggered approach to solve fluid-structure interaction problems. In this technique Abaqus determines the structural portion of the solution using FEM, and FLUENT determines the fluid portion of the solution using a finite volume method. Information is exchanged between the codes at the common domain boundaries as each individual analysis moves forward in time.

The advantage of this methodology is that it allows relatively independent setup of the fluid and structural subproblems using familiar general-purpose solution tools and it allows for the solution of a wide variety of FSI problems.

The interface coupling between Abaqus and FLUENT is based on industry-standard scalable interprocess communication and mapping tools (MpCCI) and is provided by Fraunhofer SCAI. MpCCI provides for the transfer of film coefficients/heat fluxes/pressures from FLUENT to Abaqus and the transfer of temperature and structural position from Abaqus to FLUENT.

** All images presented in this technology brief were generated in EnSight Gold from CEI.*

FSI Examples

The capabilities of the co-simulation feature are discussed within the context of some representative analyses.

Flexible Flap in an Airflow

The following example illustrates the dynamic effects of airflow over a deformable structure. In this analysis a flexible flap is placed in an air stream. The flap deforms due to the normal and viscous fluid forces of the moving air. This deformation, in turn, affects the flow field.

Figure 1 shows the deformation of the flap in a sequence of images. The orange arrows represent fluid pathlines, and the contours of the fluid domain represent the fluid velocity.

Although this problem is academic in nature, it illustrates that the Abaqus-FLUENT FSI capability can be used to solve problems with strong fluid-structure interactions. The simulation was performed using Abaqus/Explicit and the segregated implicit solver in FLUENT. A turbulent flow is modeled with the standard k-ε model. FLUENT transfers the normal and viscous forces acting on the flap to Abaqus via the MpCCI interface. Abaqus computes the current structural geometry and transfers the structural positions to FLUENT via MpCCI. The dynamic meshing capability in FLUENT is used to move the CFD mesh based on this interface coordinate information.

Bimetallic Strip

This example illustrates fluid-structure interaction that involves coupled thermal-displacement interaction. A bimetallic strip employs a mismatch of thermal expansion coefficients to induce structural deformation with temperature and is used in industrial applications such as anti-scald shower heads, temperature sensors, flow limiters, and MEMS sensors.

The bimetallic strip deforms as it heats up under the influence of an upstream hot fluid. The deforming strip acts as a guide, channeling the hot fluid through an ancillary exit in the flow stream. This limits the flow of hot fluids downstream where high temperatures may be undesirable.

The fluid-structure interaction arises from the deforming structure affecting the fluid flow and heat transfer rate. The film conditions at the structure-fluid interface are changing with time and can be obtained accurately with a coupled analysis.

Figure 2 shows the response of the bimetallic strip to varying flow rates. Specifically, three flow rates are considered, while all other design parameters, such as upstream fluid temperatures and structural properties, remain unchanged. The contours in the fluid domain show the velocity field, and the contours in the structural domain show the displacements. The top plot corresponds to the lowest flow rate, and the bottom plot corresponds to the highest flow rate.

The rate of structural heating is proportional to the rate of the flow. Hence, at any given instant of time the strip in the low flow rate exhibits smaller deformations than in the higher flow rates. Most of the flow has been cut off for the highest flow rate case, while for the same time duration, the bimetallic strip has not deformed significantly to limit the flow in the slower moving fluid.

The problem was completed using the transient coupled temperature-displacement procedure in Abaqus/Standard and the implicit solver in FLUENT. A turbulent flow regime was used, providing Abaqus with the normal and viscous fluid forces, an updated heat transfer coefficient, and the ambient (fluid) temperature necessary to drive the Abaqus simulation. Abaqus, in turn, computes the wall temperature and the interface geometry.

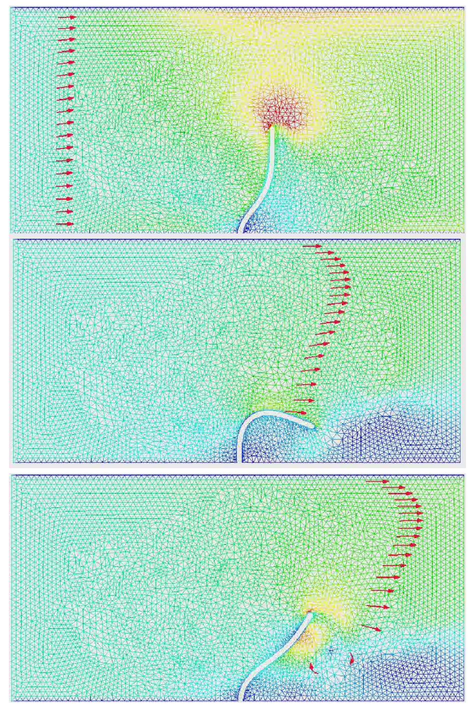


Figure 1: Velocity contour plots of flexible flap in an air stream

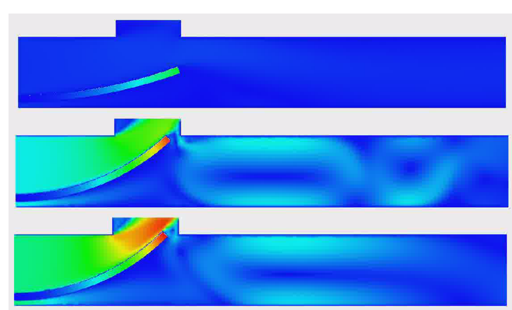


Figure 2: Velocity and displacement contour plots of bimetallic strip in a hot upstream fluid

Exhaust Manifold

Thermo-mechanical FSI coupling is illustrated in the transient heating of an automotive exhaust manifold due to flow of the internal hot exhaust gas. Figure 3 shows a representative result from this analysis.

The contours on the manifold show temperature distribution at a given time; the pathlines show the flow field. FLUENT computes the internal flow and the heat flux, and Abaqus/Standard computes the structural heating and the thermal stresses. In this case MpCCI provides the transfer of heat fluxes from FLUENT to Abaqus, while Abaqus provides the wall temperature to FLUENT. Abaqus further calculates the thermal stress and deformation of the exhaust manifold due to heating. Such displacements may also be passed back to FLUENT for cases where the thermal distortion of the structure significantly changes the shape of the flow domain. In this case this coupling effect was insignificant and the wall geometry was not updated.

The long-standing approach for these problems has been to determine a steady-state temperature profile of the interface wall in a CFD code. This interface temperature, along with assumed film conditions, is then used to predict stresses in the manifold. The current approach allows not only for steady-state analyses but for transient simulations as well.

Aortic Heart Valve

The next example illustrates pulsatile flow through the aortic heart valve. The structural deformation of the heart valve is controlled by the blood pressure. The valve opens in response to a high pressure (systolic) pulse, thus allowing blood to be pumped to the arteries. During the diastolic phase the blood pressure reduces; subsequently, the valve will close.

Figure 4 shows the position of the heart valve at a systolic pressure. The upper half of the figure shows contours of the fluid pressures, and the lower half shows contours of flow velocity. The embedded displacement versus time plot shows valve tip displacements in response to the pulsatile flow. This simulation uses the implicit dynamic procedure in Abaqus/Standard with the implicit solvers in FLUENT. The interface fluid pressures and the resultant positions are exchanged.

Antilock Braking System

This example considers a representative ball valve used in an antilock braking system (ABS). Ball valves, as the name implies, are stop valves that use a spring-loaded ball to stop or to start the flow of a fluid. Depending on the fluid pressure, the ball valve will open and stay open until the excess fluid pressure is released.

Figure 5 shows the ball valve in operation under three different braking pressures. The leftmost plot corresponds to nominal braking, and the rightmost plot corresponds to the maximum braking pressure applied. With an increase in the braking pressure, the valve opens more frequently to allow for the rapid release of the excess pressure. The analysis scheme used is similar to that of the heart valve example.

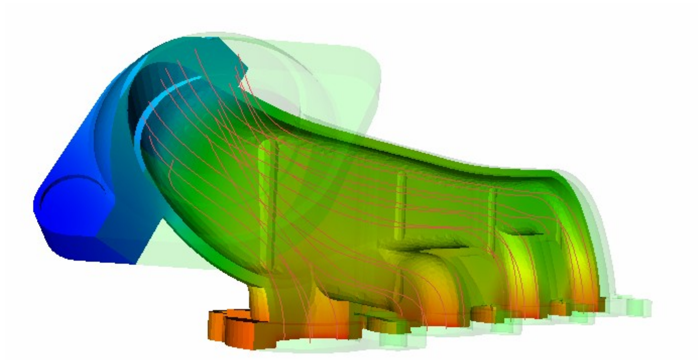


Figure 3: Temperature contour plot of exhaust manifold

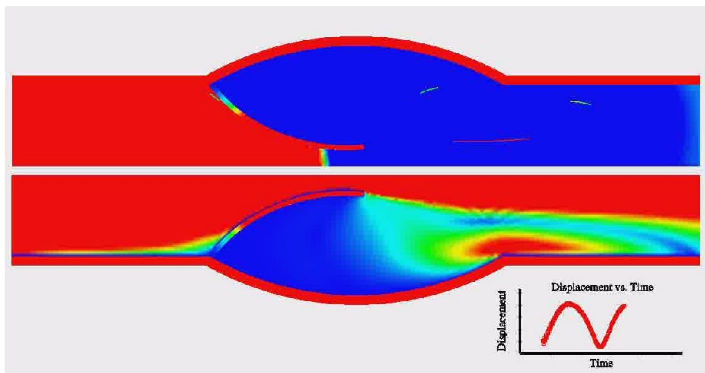


Figure 4: Pressure, velocity, and displacement contour plot of aortic heart valve

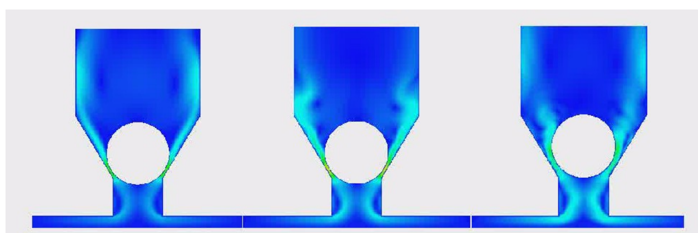


Figure 5: Velocity contour plot of an antilock braking system under different braking pressures

Reference

Chakrabarti, S., “Fluid Structure Interactions II,” Computational Mechanics, May 2003.

Abaqus Reference

For additional information on the Abaqus capabilities referred to in this brief, see the following Abaqus 6.11 documentation reference:

- Analysis User’s Manual
  - “Co-simulation: overview,” Section 16.1.1

FLUENT References

See http://www.fluent.com for additional information on Fluent, Inc., and the FLUENT capabilities shown in this brief.

MpCCI References

See http://www.scai.fraunhofer.de for more information on MpCCI and Fraunhofer SCAI.

EnSight References

See http://www.ceintl.com for additional information on EnSight Gold, a unified postprocessing tool for FSI applications.