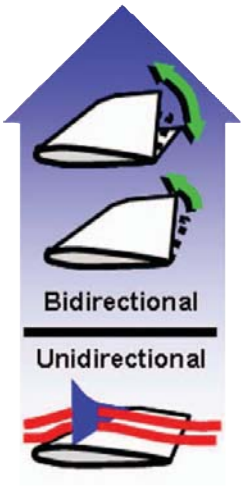
Advanced FSI Technology: ANSYS® MFX Multi-field™ Solver

Technical Brief



Implicit method is required for tightly-coupled transient simulations.

Explicit method is adequate for steady-state simulations.

One-way or uni-directional method is adequate for limited physical couplings.

Suitability of coupling methods for FSI problems with one-way and two-way (bi-directional) couplings.

The MFX Multi-field solver is available in the ANSYS® Multiphysics™ and ANSYS® Mechanical™ products.

Tightly integrated fluid and solid physics delivered by the ANSYS MFX Multi-field solver is one aspect of performing advanced FSI simulation. For more details on other FSI-related features and the ANSYS® CFX® solver technology, please to the *ANSYS CFX Technical Briefs* on:

- Flexible Moving Mesh
- Fast, Reliable, Robust, Accurate Numerics
- Flexible CFD Solution
- Conjugate Heat Transfer Analysis

www.ansys.com

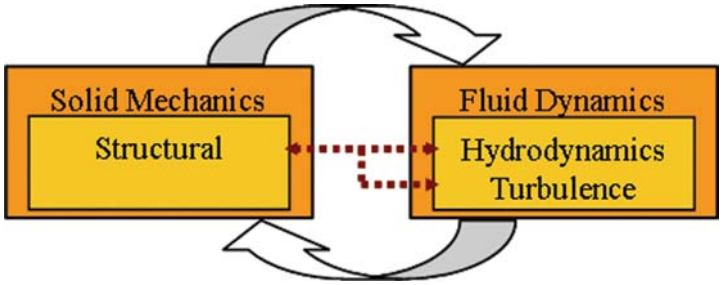
Fluid structure interaction (FSI) occurs when fluid flow generates forces on a solid structure, causing it to deform and potentially perturb the initial fluid flow. This type of interaction causes the deformation of an aircraft wing during flight, for example, or the vibration of a civil engineering structure due to airflow. Fluid structure interaction also commonly includes other interactions, such as heat transfer, between fluids and solids.

Optimal coupling methods for the execution of FSI simulations are determined by the complexity of the fluid and solid models and their physical coupling. A one-way coupling method (for example, data transfer) is adequate when the physical coupling between the fluid and solid models is steady and predominantly from one model to the other. For example, an accurate structural analysis that is based upon static loads from a completed fluid flow simulation is possible if the resulting structural deformations do not feed back to the fluid flow field. Several one-way coupling methods (including surface and volumetric load transfer) are available as part of the ANSYS FSI solution.

A two-way coupling method is required whenever the physical coupling between the fluid and solid models involves significant feedback from one model to the other. For example, this method is required to simulate the potentially transient deformation of an aircraft wing as air flows around it. Tight integration of advanced fluid and structural analysis tools is required for the efficient and accurate solution of large and physically complex FSI problems. The ANSYS MFX Multi-field solver delivers this integration.

FSI Solutions with Tightly Integrated Fluid and Solid Physics

FSI simulations require the solution of multiple coupled fields. For example, solution data from the solid model structural field is required for the fluid model hydrodynamics field. Two approaches, direct and iterative, exist for addressing this coupling.



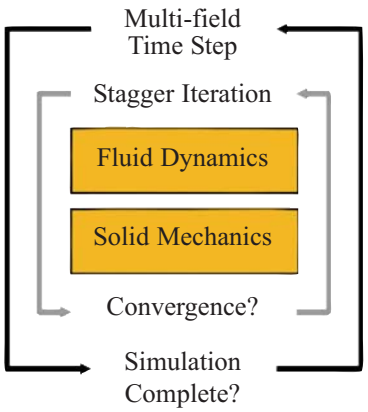
Iterative coupling approach

The *direct* approach involves assembling a single monolithic equation set, with coefficients to actively couple the individual field equations.

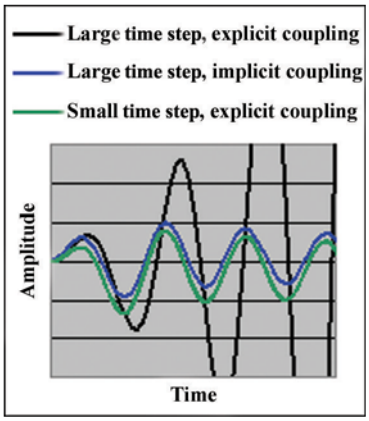
In spite of solving all field equations together, the direct approach still requires iteration to resolve the nonlinearities that exist in all FSI simulations. The *iterative* or *load transfer* approach involves assembling and solving an equation set for each field, with coupling data transferred at field interfaces. At least one coupling iteration, consisting of data transfer and equation set solution, is needed for each field to achieve a coupled response.

ANSYS®

Technical Brief



Multiple stagger iterations are performed during each time step to ensure implicit coupling.



AGARD 445.6 Wing Flutter: Implicit coupling allows a larger multi-field time step.

www.ansys.com

Iterative Coupling Between World-Class CFD and FEA Solvers

The MFX Multi-field solver uses an iterative approach to couple the ANSYS® CFX® and ANSYS® field solvers. In addition to accommodating pre-processing, solution, and post-processing methods optimized for fluid and structural analysis, the MFX solver delivers:

- Use of existing, well-proven solvers for both fluid and structural analysis, thereby eliminating the need for re-validation
- Significantly lower run-time memory requirements for large simulations than those for a direct coupling approach
- Easy access to a comprehensive range of existing state-of-the-art fluid and structural physical models
- Immediate availability of new physical models added to the fluid and structural analysis solutions

Robust, Reliable and Efficient Implicit Coupling

MFX simulations are executed as a series of multi-field load or time steps. Within each multi-field step, multiple stagger iterations are performed until each of the fluid and solid field equations and the coupling data transferred at the field interfaces have converged. This implicit coupling ensures that fluid and solid solution fields are consistent with each other at the end of each multi-field step, which leads to improved numerical solution stability. Overall efficiency improves since time step sizes are limited only by the physical processes being modeled. Conversely, an explicit coupling does not ensure solution field consistency, and time step sizes can be severely limited by the numerical solution process.

Additional controls over the ordering of field solver execution and the under-relaxation of coupling data transferred at field interfaces allow fine-tuning of the solution process. The MFX Multi-field solver uses fully automatic methods for treating dissimilar meshes at solid fluid interfaces, with both profile-preserving and conservative interpolation options available at each interface.

Native Communication Infrastructure

Two-way FSI applications require data transfer between the fluid and solid models throughout the simulation. The MFX solver achieves this through a native communication infrastructure that is based on a client/server protocol using standard internet sockets. This infrastructure offers:

- Data transfer between ANSYS CFX and ANSYS field solvers running on different physical architectures and operating systems
- Data transfer between field solvers on a single machine, and across LAN, WAN and internet connections
- Independence from third-party coupling software
- Independence from parallel processing libraries (like PVM and MPI or MPICH), hence all CFX parallel processing options are fully supported

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
USA
724.746.3304
ansysinfo@ansys.com

Toll Free USA/Canada:
1.866.267.9724
Toll Free Mexico:
001.866.267.9724
Europe:
44.870.010.4456
eu.sales@ansys.com

cfx-fsi_multifield_ Dec 06

GSA Contract Holder