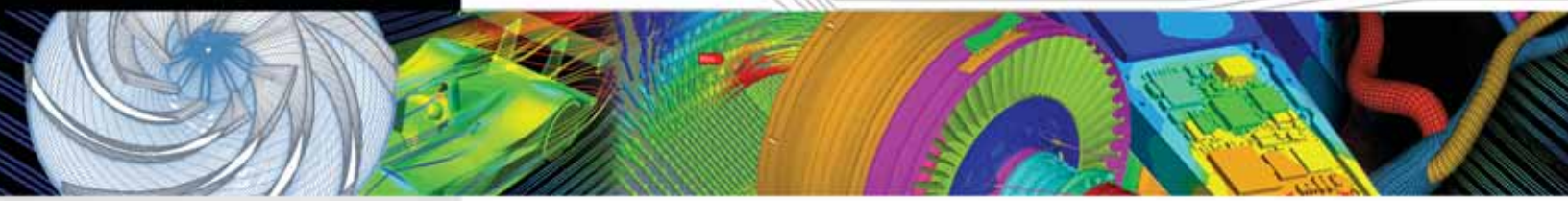
Fluid Structure Interaction Solution



Product Features

Coupled Analysis Classes

Basic FSI

- File based
- Sequential (load vector) coupling
- Unidirectional coupling
  - Fluid-to-structure
  - Fluid-thermal-to-structure
  - Structure-to-fluid
  - Thermal-to-fluid
  - Electromagnetic-to-fluid
- Surface or volume based loads
  - Pressure
  - Force
  - Displacements
  - Temperature
  - Heat flux
  - Heat transient coefficient
- Interpolation schemes
  - Non conservative
  - Conservative
  - Cloud of points
- Applications
  - Steady-state analysis
  - Fixed geometry

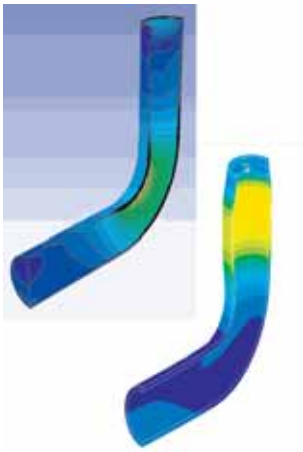
Advanced FSI

- Multi-field solver based
- Iterative (sequential & simultaneous) coupling
  - Bidirectional coupling
  - Fluid-structure
  - Fluid-thermal-structure
  - Structure-fluid
  - Thermal-fluid
  - Conjugate heat transfer (CHT)
- Surface or volume based loads
  - Pressure
  - Force
  - Displacements
  - Temperature
  - Heat flux
- Interpolation schemes
  - Nonconservative
  - Conservative
- Applications
  - Steady-state analysis
  - Time transient analysis
  - Fixed geometry
  - Moving & deforming geometry

www.ansys.com

The scalable FSI solution, from entry level to an advanced multi-user FSI environment!

Fluid Structure Interaction (FSI), is where fluid flow exerts pressure on a solid structure causing it to deform such that it perturbs 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.



Biomedical advanced FSI analysis of blood flowing through an elastic artery. Top left shows fluid velocity, bottom right show structural deformation and stress contours.

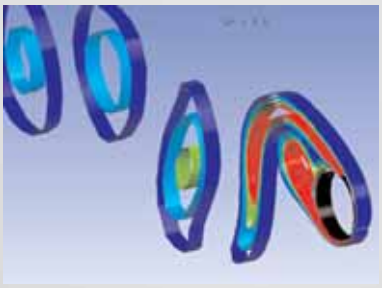
The ANSYS Fluid Structure Interaction solution provides the analysis industry's most flexible and advanced coupled structural-fluid physics analysis tool. Fluid-structure interaction is required for many industry applications such as biomedical (elastic artery modeling for stent design), aero foil flutter and civil engineering (wind loading of structures). The ANSYS Fluid Structure Interaction solution consists of two levels of fluid-structure coupling.

- **Basic FSI:** Entry level coupling between structural, electromagnetic and fluids suitable for steady-state analysis. The coupling is file based and unidirectional and involves using the result of one simulation as a load or boundary condition for a second simulation. This level of Computational Fluid Dynamics (CFD) coupling is available between all ANSYS (Professional and higher) and CFX® solver products.

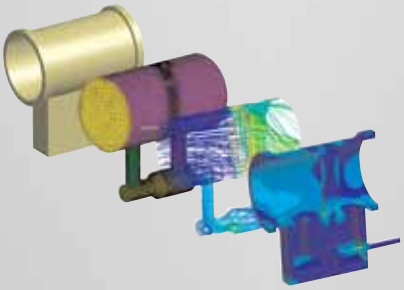
- **Advanced FSI:** Uses the ANSYS® Multi-field™ solver to provide a true bi-directional FSI capability for time transient or steady state analysis with moving/deforming geometry. This capability has been implemented for ANSYS® Mechanical™ and ANSYS® Multiphysics™ coupling to the CFX full capability solver, and also is available for coupling with ANSYS® FLOTRAN™ within ANSYS Multiphysics.

ANSYS Multi-field Solver

The ANSYS Multi-field solver provides an easy-to-use framework to solve coupled field problems in many new markets and applications where solutions have not previously existed. The Multi-field solver is a general purpose, automated iterative coupled physics solver applicable across all physics available in the ANSYS Multiphysics solution. The ANSYS Multi-field solver has been enhanced to support coupling with CFX for Fluid Structure Interaction applications. The structural part of the analysis is solved using ANSYS Multiphysics (or ANSYS Mechanical) and the fluid part using the CFX full capability solver. The enhanced Multi-field solver technology (called MFX-Multiple code) allows the structural and fluid solutions to run simultaneously on the same or different machines, thus accommodating much larger models more efficiently than a multi-field solver using a single machine environment. The MFX-Multiple code coupling is based on proprietary inter-process communication technology. This technology ensures that the CFX solver can be run in parallel using any of its built-in parallel communication methods without any interference or conflict with the MFX coupling. This means not only that the fluid and structural computation be run on different machines, but also that any number of computers can be applied to reduce the wall clock time of the fluid portion of the simulation. Further, the MFX-Multiple code solver can communicate across a local area network, wide area network or even via an Internet connection.



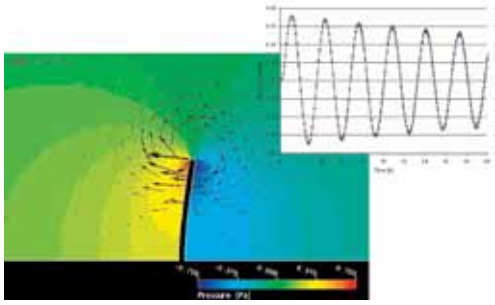
Advanced FSI. Vortex shedding from an oscillating cylinder at critical frequency.



Unidirectional FSI analysis of valve showing model, mesh, fluid flow streamlines and resultant structural deformation.

Key Features of Advanced FSI:

- Fluid and structural physics are treated as separate “fields” with an independent model and mesh
- Features within each field are supported by ANSYS Mechanical and the CFX full capability solver
- Mechanical and CFD Physics are coupled by passing loads across field interfaces
- Dissimilar mesh allowed for each field
- Surface and volumetric loads transfer across physics fields
- Fluid and structural field solutions can be divided between two separate computers
- Iterative (load vector) coupling is between fields and each field may have different:
  - Analysis types (transient, static or harmonic)
  - Solvers and analysis options
  - Mesh descretization
- Automated mesh morphing
- Material and geometric nonlinearity
- Independent results files for each physics “field”



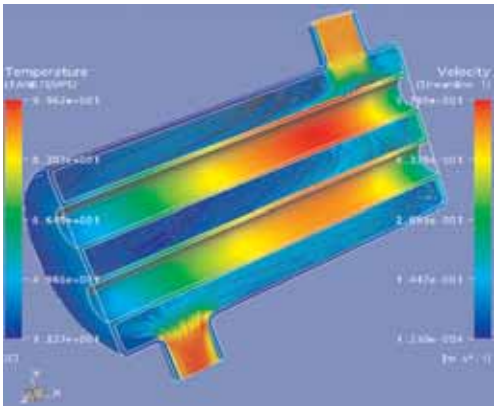
Advanced FSI analysis of wall oscillating in fluid flow.

Advanced FSI Solution Benefits

The Advanced FSI solution provides many benefits inherent from a single vendor solution.

- No third-party coupling scheme is needed. This reduces the solution deployment cost since you have fewer analysis software tools to purchase, learn and manage.
- Large FSI problem sizes can be tackled since the solution can be spread across two machines, and the CFX solution may make additional use of parallel processing (requires additional CFX parallel modules).
- Robustness and reliable implicit coupling mean less time and effort to achieve the desired simulation. Great care is taken to ensure that the coupling has converged (implicit) at the end of each timestep by use of a stagger loop and intimate data exchange as needed even within a single timestep.
- FSI physics coupling can make use of your LAN, WAN or Internet connection.

For a complete list of ANSYS products, including add-on modules, please visit www.ansys.com/products.



Unidirectional FSI. Thermal-stress analysis of a heat exchanger.

www.ansys.com

GSA Contract Holder
FSS Contract 0123456789

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

PBR05MAY010