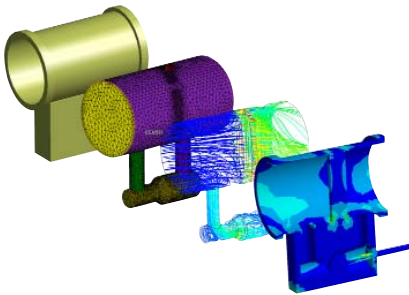


### Product Features

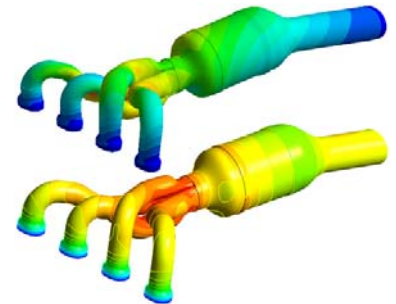
- ▶ File-based or Multi-field solver-based load transfer
- ▶ Surface or volume loads
  - Pressure
  - Force
  - Displacements
  - Temperature
  - Heat flux
  - Heat transient coefficient
- ▶ Applications
  - Steady-state analysis
  - Time transient analysis
  - Fixed geometry
  - Moving and deforming geometry



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

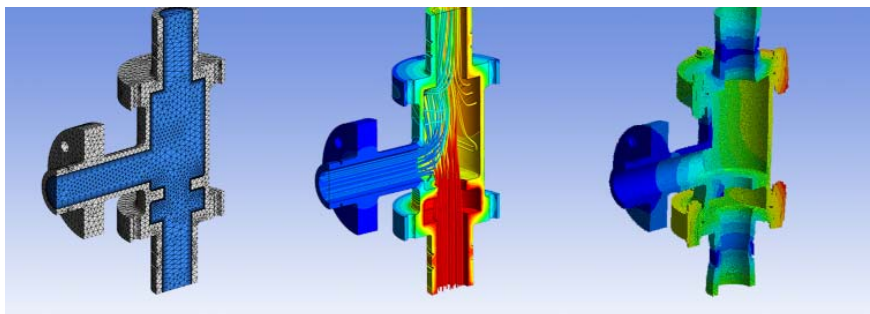
### ANSYS® Workbench™ 11.0 Makes Simulating Fluid Structure Interaction Easier

Fluid structure interaction (FSI) occurs when a fluid interacts with a solid structure, exerting pressure that may cause deformation in the structure and, thus, alter the flow of the fluid itself. The ANSYS FSI solution provides designers and analysts with the most flexible and advanced coupled structural-fluid physics analysis tool available. Fluid structure interaction is required for many industry applications, such as bio-medical (elastic artery modeling for stent design), aerospace (foil flutter) and civil engineering (wind loading of structures). There is a complete spectrum of fluid structure interactions that are important in industry. FSI simulation is steadily gaining importance as business drivers push designers to make their products lighter, more flexible, easier to manufacture, while maintaining or improving quality and reliability.



FSI thermal stress simulation of an automotive exhaust manifold

FSI simulations can be broadly categorized as one-way or two-way coupled. One-way coupling exists, for example, in the T-junction simulation model shown below. For this case, significant thermal stresses in the solid are induced by thermal gradients in the flow field; however, since the resulting deformation of the solid is small, the flow field is not greatly affected. This allows computational fluid dynamics (CFD) and finite element analysis (FEA) solutions to be run independently, with loads transferred in only one direction.



CFD Mesh

Conjugate heat transfer  
CFD simulation

ANSYS deformation results

One-way fluid structure interaction simulation of a T-junction is performed within a single simulation environment. The thermal loads are passed from the CFD simulation to the structural simulation to predict deformation. *Geometry courtesy CADFEM GmbH.*



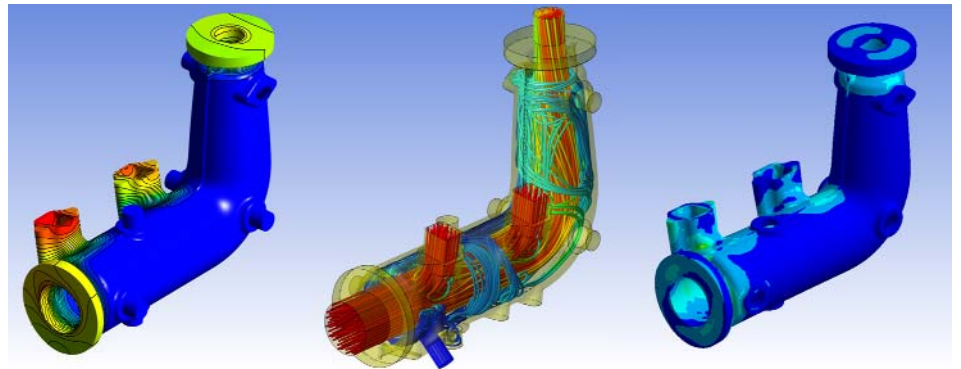
## Product Features

### FSI Key Features

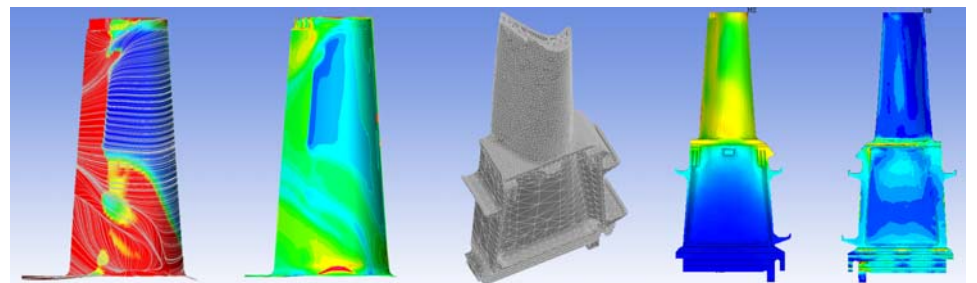
- ▶ Access to all features supported by ANSYS® Mechanical™ and ANSYS CFX products. As a result, users are not limited by “lightweight” CFD or FEA technology on either side of the FSI interface, allowing them to simulate challenging, real-world problems.
- ▶ Fluid and structural physics are treated as separate “fields” with an independent model and mesh.
- ▶ Mechanical and CFD physics are coupled by passing loads across field interfaces.
- ▶ Dissimilar mesh is 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 discretization
- ▶ Automated mesh morphing
- ▶ Material and geometric nonlinearity
- ▶ Independent results files for each physics “field”
- ▶ The ANSYS CFX solver manager can be used to launch and monitor the coupled CFX ANSYS simulation.
- ▶ Both CFD and FEA results can be simultaneously post-processed in the ANSYS CFX post-processor.

Perhaps the most common application of one-way FSI simulation is the solution of thermal-stress problems. The flow field is complex and a CFD solution is required to determine the thermal condition at the interface between the fluid and solid domains. Coupling CFD and FEA in this fashion is not new, as most commercial CFD and FEA codes support file-based transfer of heat transfer data. However, ensuring that the independent CFD and FEA models are correctly located and scaled historically had been left to the user, and data transfer errors were all too common. The use of a shared geometry model within the ANSYS Workbench environment eliminates problems of this nature, and ANSYS® 11.0 software makes load transfer trivial.

Using ANSYS FSI, a thermal-stress simulation can be performed within ANSYS Workbench. For this gas engine exhaust header (below), thermal loads were transferred from ANSYS® CFX® to ANSYS software in order to determine the heat transfer between the fluid and the solid body. From this information, stresses could be determined and ultimately a fatigue analysis performed, all within the ANSYS Workbench environment.



ANSYS FSI thermal-stress simulation of a gas turbine exhaust header  
 Left: temperature of the solid; center: computational fluid dynamics simulation including two separate fluids and heat transfer in the solid; right: stress simulation  
 Courtesy CADFEM.



Turbulence Intermittency with Wall Shear Stress Lines

ANSYS Workbench Mesh

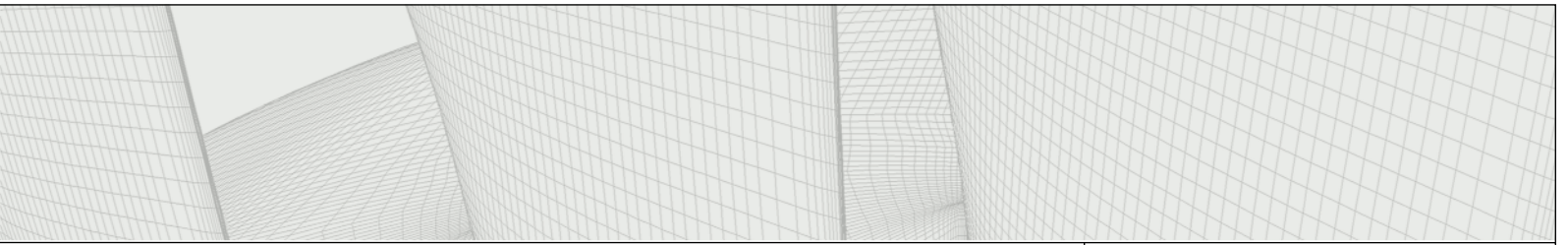
Temperature

Von Mises - Stresses

Heat Transfer Coefficient

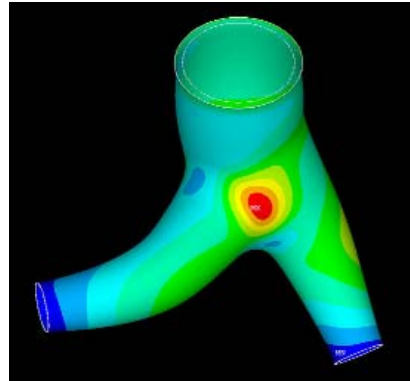
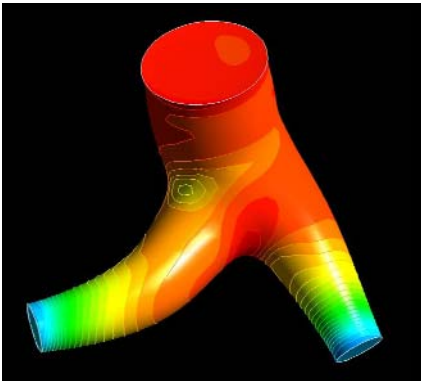
Thermal performance of a gas turbine blade can be determined early in the design process with fluid structure interaction.

Courtesy Wood Group Heavy Industrial Turbines.



With the release of ANSYS 11.0, usability has been greatly improved and both one-way and two-way FSI simulations now can be performed entirely within the ANSYS Workbench environment. The process of setting up, solving and post-processing has been dramatically streamlined.

For cases in which the structure deforms and significantly affects the flow field, two-way FSI is needed. Industrial examples include aerodynamic flutter of wings, buffeting of car hoods, transient wind loads on buildings and bridges, and biomedical flows involving compliant blood vessels and valves. For cases such as this, both ANSYS and ANSYS CFX must be run concurrently with loads transferred between the two solvers. The coupling between ANSYS and ANSYS CFX is unique in that socket-based inter-process communication is built in, and no third-party coupling software is required. Two-way coupled 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.



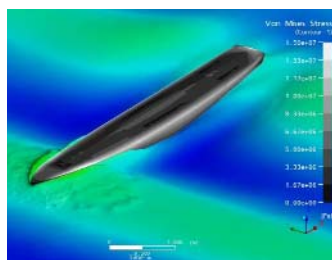
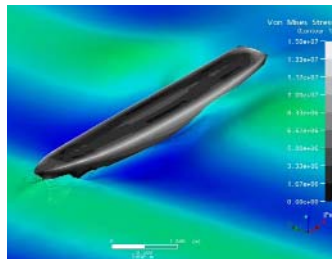
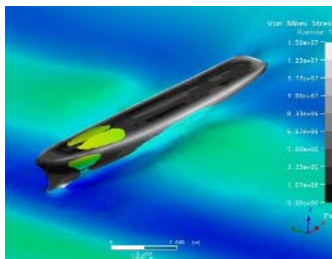
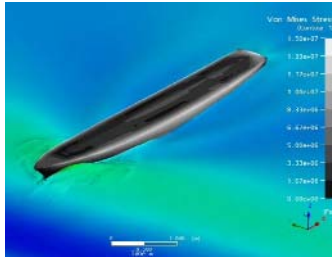
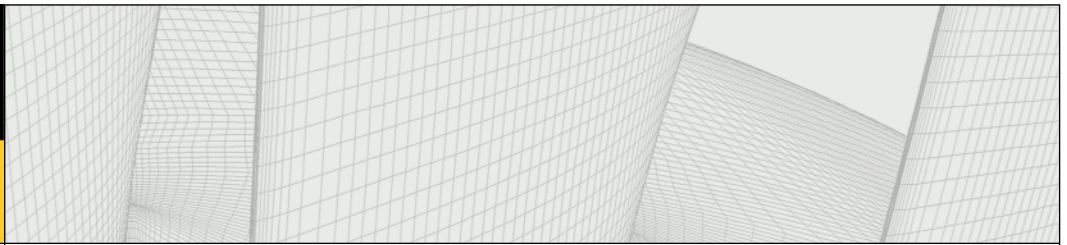
Two-way FSI simulation of pulsing blood flow in a deforming artery provides insight into the complex transient physics. Eventually, the clinical use of non-invasive, patient-specific simulation may provide better understanding of arterial hypertension and improved predictions of the potential outcomes of treatments. Left: CFD simulation of pressure within the pulmonary artery; right: structural deformation of the artery. *Courtesy University of Colorado Health Science Center.*

## Solution Benefits

### FSI Solution Benefits

The Advanced FSI solution provides many benefits inherent in a single-vendor solution:

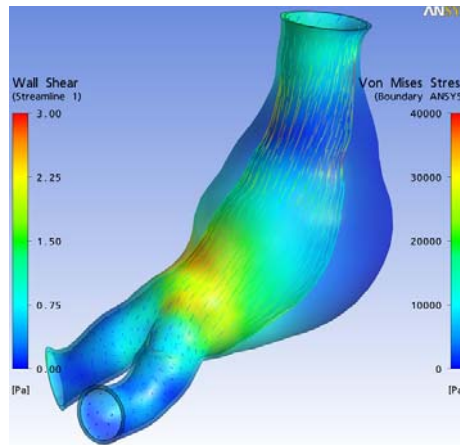
- ▶ A single environment with a consistent graphical user interface, making it easier to learn and use
- ▶ No third-party coupling scheme or post-processing products are needed. This reduces the solution deployment cost, since it results in fewer analysis software tools to purchase, learn and manage.
- ▶ Robust and reliable implicit coupling mean less time and effort is needed to achieve the desired simulation. Great care is taken to ensure that the coupling has converged (implicit) at the end of each time step by use of a stagger loop and intermediate data exchange as needed, even within a single time step.
- ▶ Load transfers between the FEA and CFD solvers are both profile-preserving and conservative.
- ▶ Large FSI problem sizes can be tackled since the solution can be spread across two machines, and the ANSYS CFX solution may make additional use of parallel processing (requires additional ANSYS CFX parallel modules).
- ▶ FSI physics coupling can make use of LAN, WAN or Internet connections.



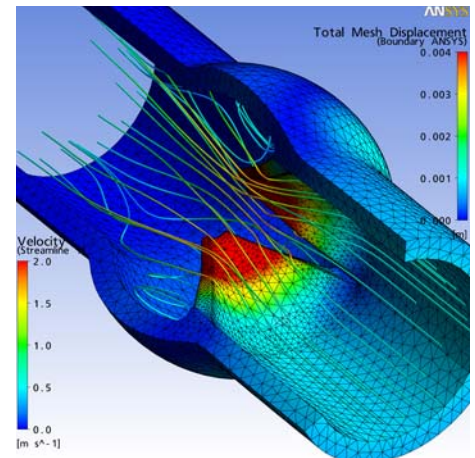
This dynamic sink and trim calculation for a ship combines the free surface and moving mesh capabilities in ANSYS CFX with ANSYS Mechanical software.

## ANSYS Multi-field Solver Technology

The ANSYS Multi-field solver provides an easy-to-use framework to solve coupled field problems in many new industries and applications in which solutions have not previously existed. The Multi-field solver is a general-purpose, automated iterative coupled physics solver applicable across all physics. It supports coupling with ANSYS CFX software for fluid structure interaction applications. The structural part of the analysis is solved using the ANSYS solver and the fluid part using the CFX solver. The enhanced Multi-field solver technology 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 multi-field coupling is based on customized 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 solver coupling. Therefore, the fluid and structural computation be run on different machines, and any number of computers can be applied to reduce the clock time of the fluid portion of the simulation. Multi-field solver technology enables ANSYS to provide effective and efficient tools for a broad range of industrial fluid structure interactions, recognizing that the coupling strength and physics modeling of the solid and fluid components are key factors.



Two-way ANSYS FSI simulation of flow in an aortic aneurysm. Vessel walls are rendered transparent and colored with Von Mises stress. Also shown are surface streamlines colored by the local wall shear stress.



Simulation of three-lobe biomedical valve that deforms in response to a pressure pulse at the inlet.

For a complete list of ANSYS products, including add-on modules, visit [www.ansys.com/products](http://www.ansys.com/products).