ANSYS Fluid Structure Interaction for Thermal Management and Aeroelasticity

Phil Stopford

Duxford Air Museum
11th May 2011
Fluid Structure Interaction (FSI)

• What is Fluid Structure Interaction?
  – Occurs when fluid flow interacts with solid structures, exerts pressure and/or thermal loads that may cause structural deformations and thus affecting the fluid flow itself

• Why is FSI important?
  – Crucial in understanding many engineering problems
    • Material selection, fatigue, effect on fluid flow parameters etc.
  – For better designs!
  – Can be catastrophic if neglected
FSI Modeling Approaches

• Two-way Coupling
  – Coupling of FEA and CFD solvers
    • Implicit and Explicit Approaches
    • E.g. Vortex induced vibration, large time scale phenomenon

• One-way Coupling
  – One-way interaction
  – Fluid pressure/temperatures produces structural loads, but strains too small to affect fluid flow field
  – Superposition methods: modal analysis provides deformed shape to flow field
ANSYS Offerings for FSI

• Two-way Coupling
  – ANSYS Mechanical – CFX
    • Iteratively implicit coupling
    • Fully integrated environment
    • Two-way coupling with FLUENT in ANSYS 14

• One-way Coupling
  – ANSYS Mechanical – FLUENT or CFX
    • Transient 1-way coupling is best performed using the 2-way analysis approach
One-way Coupling – Overview

- Couple ANSYS Mechanical with FLUENT or CFX
  - Coupling to thermal and structural analysis in ANSYS

- Applications
  - Any application involving thermal-stresses or transfer of fluid pressure/viscous forces
  - Steady state and transient analysis
Integrated Process in Workbench

Project Schematic

Geometry
CHT Mesh
CFD CHT Solution
Thermal Loads
Pressure Loads
Thermal Stress Solution
1-way Structural

- Transfer forces from CFD to ANSYS
- Transfer displacements from ANSYS to CFD
- Steady state
  - Transfer occurs after-the-fact
- Transient
  - Can use scripting to create a series of load files from a completed run
    - Use APDL or CEL to read the loads in at the appropriate time
  - Implemented more easily within the 2-way framework by sending data in only one direction
1-way Time Averaged Data

• Time-averaged data is useful in a number of cases, e.g.
  – Averaged pressure loads from transient CFD simulations
    • LES, DES, SAS

• Time-averaged data can be generated and passed to ANSYS as a static load
Two-way Coupling: ANSYS – CFX

• Couples ANSYS Mechanical solver and ANSYS CFX
  – Retains advanced physics capabilities of both solvers
  – Available in FLUENT in Version 14

• Option of Steady and Transient Coupling

• Force and/or Heat Flux/Temperature data transfer
  – Any other field variable

• Unified and fully coupled environment in ANSYS WorkBench

• Semi-Implicit Matrix Coupling through Multi-field Solver
Two-way Coupling: ANSYS – CFX

- Coupling is achieved by transferring surface loads / displacements across physics interface
- An iterative coupling approach *within* each timestep provides implicit coupling at each timestep…

![Diagram showing coupling between Solid Mechanics and Fluid Dynamics](image)

- Solid Mechanics
- Structural
- Fluid Dynamics
- Mass
- Momentum
- Turbulence
- Heat Transfer
Semi-Implicit Matrix Coupling

- Physics fields calculated by separate solvers
  - Multiple data transfers within timestep
  - Implicit solution at end of timestep
Two-way Coupling: Key Features

• Easy to setup
• Total Forces and Heat Fluxes are conservative across FSI interface
• Non-conformal meshes
• Automatically morphs CFD mesh
• Large Models
  – Both sides can use parallel computing
• Third party coupling scheme not required
• Data transfer across TCP/IP internet sockets
  – Efficient; no intermediate files
  – Heterogeneous architectures (Linux, Windows)
  – Solvers can run on different machines (LAN, WAN, Internet)
Two-way FSI Workflow

- The workflow is built on the WB Project page

Streamlined process integration without leaving the Workbench environment
Two-way FSI Workflow – Geometry

- **Solid and Fluid geometry in ANSYS DesignModeler**
  - Create and modify CAD geometry
  - Bi-directional direct CAD connections
    - ProE, SolidWorks, UG, CATIA, etc
  - Parametric modeling capability
  - Easy fluid volume extraction

**Structural Part**

**Fluid Volume**
Two-way FSI Workflow – Meshing

- Single meshing application for structural and fluid meshes
  - Swept, Tet, Inflation, Hex Dominant, Hex Core, Multi-block
- Matching or non-matching meshes at the FSI interface
  - Fully conservative transfer across interface
- Can use other Fluid mesh generators
  - ICEM for full Hex mesh
Two-way FSI Workflow – Structural Setup

• Structural Problem setup in ANSYS Mechanical
  – Easy to use
  – Setup like any other Transient Structural simulation
    • Tag the FSI interface regions
  – Library of solid materials, advanced material properties
  – Also Modal, Random Vibration, Thermal Stress, Harmonic Response, …
Two-way FSI Workflow – Fluid and FSI Setup

• Coupled simulation set-up in ANSYS CFX-Pre
• 2-way data/load transfer specified
• Simple & intuitive FSI interface panels
  – User-friendly, easy to use

Transient controls (common)

Coupling controls

Interface transfer quantities
Advanced Turbulence Models

- Advanced Turbulence Models
  - SST, LES, DES, SAS

- Advanced Wall Functions
  - Automatic blending between low-Re and Wall Function approach

- Laminar to Turbulent Boundary Layer Transition
  - Unique ANSYS capability
  - Completely automatic prediction of transition onset
Turbulence Transition Model
Wind Turbine Blade

Tu Contour

Transition

Transition

Transition

Transition
Two-way FSI Workflow – Solving

- Both solvers automatically started from the CFX Solver Manager

![CFX-Solver Input]

![ANSYS Solver Input]
Two-way FSI Workflow – Solving

- Single environment for solution monitoring
  - Check interface quantities are converged within each timestep
  - Monitor forces, displacements, custom expressions
Two-way FSI Workflow – Post-processing

- Coupled simulation post-processing in ANSYS CFD-Post
  - User-friendly Graphical User Interface
  - Can analyse intermediate time step data
  - FFT

Wing Flutter analysis using 2-way FSI
FSI Examples
NREL Phase VI rotor

Rotor diameter 10.058 m
Blade are based on an aerofoil (S809)
Rotational speed 71.9 m/s
Measurements in NASA Ames wind tunnel
Cross section: 24.4 m x 36.6 m
Inlet speed 7 m/s
FSI Examples
NREL Phase VI rotor

DirectCAD interfaces can be used
Using a script a high quality mesh is generated in minutes

Blade region meshed in ICEM HEXA
Geometry imported Parasolid
Min angle > 20 deg
Nodes pr. passage 100,000
FSI Examples
NREL Phase VI rotor

Tower and nacelle parameterised in DesignModeler

Subtract solid from wind tunnel domain and meshed in Workbench

By using parameters a design change is implemented in a few minutes
FSI Examples
NREL Phase VI rotor

- Solution
  - Steady state
  - Frozen rotor interface
  - Timestep = 10/w
  - Convergence criteria (RMS): $10^{-5}$
  - Turbulence model: SST
  - Transition is important
FSI Examples
NREL Phase VI rotor
FSI Examples
NREL Phase VI rotor

One way FSI – Von Mises Stresses
FSI Examples
NREL Phase VI rotor

One way FSI – Deformations
Transient Simulation

- Steady state simulation as initial guess
- Temporal variation of fluid and structural variables
  - Temporal variation of wake
  - FSI between tower and blade (two-way coupling)
  - Noise (monopole, dipole, quadrupole)
- Time average quantities also generated
  - Expected to be similar to steady state
FSI Examples
NREL Phase VI rotor

Transient: Max deformation=0.0012
Steady: Max deformation=0.00007
FSI Examples – Leaf Valve

- Pressure pulse passing through a leaf valve
FSI Examples – “Singing” Hydrofoil

- Hydrofoil simulated at a free stream velocity that produces a resonating response
- 2 million cells for CFD
  - DES with y+ ~ 25
- 22,000 elements for FEA
- Time step = 1.63 X 10^{-4} s
FSI Examples – “Singing” Hydrofoil

Displacements Magnified 5000x
FSI Examples – Bore Choking

• Bore Choking in Solid Rocket Motors
  – Interaction between propellant grain and flow field results in the radially inward deformation of the propellant
  – Difference in pressure P1 and P2 results in deformation of the solid propellant
  – Result in artificial throat and choking of the flow, leading to pressure build up and case failure

• Self-Sustaining phenomenon
  – Deformation results in increase in difference in pressure which further increases the deformation

\[ \text{P1} \quad \text{P2} \]
FSI Examples – Bore Choking

- Results (no FSI)
  - Pressure differential around corner

Pressure Contours
FSI Examples – Bore Choking

FSI Solid Deformation (as function of time)
FSI Examples
Wing Flutter

- AGARD 445.6 test case
- Mahogany structure
- \( Ma = 0.50 \ldots 1.14 \)
- Zero angle of attack

![Diagram of wing flutter model with dimensions: 0.76 m, 0.56 m, 0.37 m, and an angle of 45°.]

© 2011 ANSYS, Inc. All rights reserved.
FSI Examples
Wing Flutter

• Modal analysis
  – Bending mode
  – Torsional mode

<table>
<thead>
<tr>
<th>Mode</th>
<th>Experiment</th>
<th>Simulation</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>9.59 Hz</td>
<td>9.37 Hz</td>
</tr>
<tr>
<td>2</td>
<td>38.16 Hz</td>
<td>39.07 Hz</td>
</tr>
</tbody>
</table>
FSI Examples
Wing Flutter

- Stagger loop: implicit each timestep
- Benefit: time-step set by physics, not code coupling

- Large timestep, more stagger iterations
- Small timestep, less stagger iterations
- Optimize physics, robustness, CPU time
Deformation increased by factor 200
FSI Examples
Static Aeroelastic Wing/Body Configuration

- 3D-simulation of HIRENASD wing
  - ‘High Re Aerostructural Dynamics’ Workshop
  - Transonic
  - Span = 1.3 m
  - Chord = 0.3445 m

Robert Selent, Technical University Dresden
Thorsten Hansen, ANSYS Germany

https://heinrich.lufmech.rwth-aachen.de
FSI Examples
Static Aeroelastic Wing/Body Configuration

1. Solve CFD
   Undeformed Grid

2. Transfer loads to CSM

3. Solve CFD
   Deformed Grid

4. Transfer deformations
FSI Examples
Static Aeroelastic Wing/Body Configuration

• Aeroelastic Deformations
• Alpha 0°, 2°, 4° with aerodynamic load
FSI Examples
Static Aeroelastic Wing/Body Configuration

C_p @ Sections 1, 4, 7, \( \alpha = 2^\circ \)

Section 1

Section 4

Section 7

© 2011 ANSYS, Inc. All rights reserved.
FSI Examples
Static Aeroelastic Wing/Body Configuration

Cp @ Sections 1, 4, 7, $\alpha = 2^\circ$

Section 1

Section 4

Section 7

Experiments

Simulation

Courtesy of RWTH Aachen
FSI Examples
Forced Vibration Analysis Using Mode Shapes

- Solve modal analysis in ANSYS
- Export the mode shape
- CFD: transient analysis with prescribed mesh motion

Apply as Mesh Deformations in CFD

CFD Results
FSI Examples
Forced Vibration Analysis Using Mode Shapes

- Can use superposition method to combine mode shapes

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>1\textsuperscript{st}</td>
<td>683 Hz</td>
</tr>
<tr>
<td>2\textsuperscript{nd}</td>
<td>1707 Hz</td>
</tr>
<tr>
<td>3\textsuperscript{rd}</td>
<td>2248 Hz</td>
</tr>
<tr>
<td>4\textsuperscript{th}</td>
<td>3707 Hz</td>
</tr>
</tbody>
</table>
FSI Examples
Forced Vibration Analysis Using Mode Shapes

- Can use superposition method to combine mode shapes

\[ x_{disp} = \sum A_i \sin(\omega_i t) \phi_i \]

- \( A_i \): constant amplitude for \( i \)th mode
- \( \omega_i \): frequency for \( i \)th mode
- \( \phi_i \): \( i \)th mode shape

Deformation, scaled by factor 200

Normal force on blade
FSI Examples
Forced Vibration Analysis Using Mode Shapes

- Workbench Project Schematic
  - 1-click project update for entire system!
Summary

• ANSYS Workbench simplifies FSI simulations with FEA and CFD
  – Single multiphysics environment
  – Streamlined workflow

• Can be combined with industry-leading turbulence and physical models

• Extensive experience in wind power and aeroelastic simulations