Automotive Fluid-Structure Interaction (FSI) Concepts, Solutions and Applications

Laz Foley, ANSYS Inc.
Outline

• FSI Classifications
• FSI Solutions
• FSI Modeling Approaches
• ANSYS Workbench for FSI
• System Coupling
• Examples
FSI Classifications

• Fluid-structure interaction problems encompass a wide range of applications in many different industries
  – Aerospace, automotive, power generation, bio-medical etc.
• FSI problems fall into two general classifications; one-way and two-way. The solution to two-way fluid-structure interaction requires co-simulation between computational fluid dynamics and structural mechanics
FSI Solutions

Physical Coupling

Very Strong

Strong

Weak

Numerical Coupling

1-way (Uncoupled)

2-way

Explicit

Implicit

Iterative

Fully Coupled

Biomedical, membranes, highly deformable solids, ...

Vortex induced vibrations, ...

Blade deformations, rigid bodies, ...

CHT, small deformations (excluding turbulence induced), ...

Thursday, September 06, 2012
1-Way Thermal, Structural

• This usually means transferring CFD thermal data to a structural model for a thermal stress analysis
  – No point in transferring data for a thermal analysis since a CHT solution in CFD is easier and inherently 2-way
  – Required if CFD and FEA are solved independently (iterative)
• Volume and surface transfer
  – Surface temperature/heat transfer coefficient/heat flow from ANSYS CFD to ANSYS Thermal
  – Volume heat generation from ANSYS CFD to ANSYS Structural
• Transient or steady state
2-Way Structural, Thermal, Both

- **2-Way Structural**
  - Both solvers run, exchange forces and displacements

- **2-Way Thermal**
  - Standard CHT simulations in CFD
  - Can also couple ANSYS Thermal and ANSYS CFD, but simpler and more efficient to solve in a single solver
  - Assumption: No structural deformations

- **2-Way Thermal and Structural**
  - Both solvers run, exchange temperatures, heat fluxes, forces and displacements
  - Coupled field elements in ANSYS Mechanical

Thin Flexible Filament
Rigid Bodies

• Simpler FSI approaches are possible when simplifying assumptions can be made
• A 6-DOF rigid body solver is available in ANSYS CFD
  – Explicit or implicit solution
  – Examples: Boats in waves, falling objects etc.

• For very simple motion (e.g. 1-DOF linear/angular motion) the rigid body motion can be calculated through CEL or UDF
  – Explicit or implicit solution
  – Examples: Control valves, pressure regulators etc.
ANSYS Workbench for FSI

• The core principles of ANSYS Workbench offer significant benefits for FSI based solutions
• Automated workflows and integration of multiple physics
• Parameter Manager
  – CAD Integration
  – Design Points
• Optimization
• And much more ….
System Coupling Overview

- Facilitates simulations that require tightly integrated couplings of analysis systems in the ANSYS portfolio
- Extensible architecture for range of coupling scenarios (one-, two- & n-way, static data, co-simulation...)
- ANSYS Workbench user environment and workflow
- Standard execution management and data interfaces
System Coupling Features

• Two-way surface force/displacement coupling with ANSYS Fluent and ANSYS Mechanical
  – Steady/static and transient two-way FSI
• Workbench based setup and execution
  – Windows and Linux
• Execution from command line outside of Workbench including cross-platform execution
• Integrated post-processing with ANSYS CFD-Post
• Parallel processing for both CFD and structural solutions with ANSYS HPC
  – RSM currently not supported
• Restarts for fluid-structure interaction
• Parameterization, design exploration and optimization
System Coupling Schematic Setup
System Coupling Setup GUI
Create Data Transfers
Executing System Coupling
Post-Processing System Coupling

• Oscillating Plate Verification
  – Charts show comparisons with MFX and experimental data
Exhaust Manifold

- Full conjugate heat transfer solution in ANSYS CFD
- Cast iron solid material properties
- Free stream external convection
Cylinder Head

- Thermal Stress Analysis
- Cylinder head model containing one solid domain and three fluid domains
  - Intake/Exhaust ports and cooling cavity
  - Heat flux from ANSYS CFD combustion analysis applied to fire deck
- Surface and/or volumetric thermal mapping
Axial Turbine Blade

• First stage blade for a 40MW industrial gas turbine
• Bladed deforms due to thermal loading
• Only fluid domain is solved in ANSYS CFD with transitional turbulence
• One way transfer of thermal loads (i.e. heat flux or HTC)
• Transfer procedure is typically iterative since fluid and solid are decoupled (i.e. not a conjugate heat transfer solution)
Tank Sloshing (Rigid Tank)

• Tank is excited by a time varying gravitational load for a duration of 10s
• Baffles are “non-metallic” and fixed to the tank with an adhesive
• Forces are transferred (one-way only) via co-simulation to determine the stresses acting on the baffles
  – Determine the integrity and viability of the adhesive bonding
Tank Sloshing (Non-Rigid Tank)

- Determine response of two containers under acceleration

Structural Model
- Containers modeled with solid-shell elements
- Multi-linear isotropic hardening plasticity
- Nonlinear contact

CFD Model
- Two domains modeled with fluid and air
- Fluid is assumed to take up half of tank
Reciprocating Compressor

• Transient response of a reed valves opening/closing
• Layering and Smoothing used in Fluent
  – Piston motion is defined using built-in IC panel
• Non-linear contact between reed valve and chamber head
• Boundary zone type automatically changed for open/close valve scenarios
Reed Valve

- Transient response of a reed valve opening/closing
- Closed domain with moving piston profile
- Re-meshing, smoothing and layering in Fluent
  - Piston motion uses layered mesh
  - Remeshing adds additional cells to fluid domain as valve opens (large deformation)
- Nonlinear contact in Transient Structural to cater for valve closure/bounce
Fuel Injector Leakage

- Steady State CFD and Static Structural analysis of leakage in an assembly clearance gap (~2.5 microns)
- Fuel pressure in excess of 2500 bar causes gap to deform
- Diffusion based smoothing used in CFD to cater for the mesh deformation
- Throttling effect along length of leakage path causes fuel temperature to increase by as much as 100 degrees due to viscous heating
Pressure Relief Valve

• Transient FSI simulation of a spring loaded ball valve releasing excess pressure in an ABS
  – (a) Low inlet pressure which results in a constant output pressure and no valve bounce
  – (b) Moderate inlet pressure which results in a variation of outlet pressure but minimal valve bounce
  – (c) High inlet pressure which causes outlet pressure to “chatter” and results in significant valve bounce
Ball Valve

• Simple 1DOF using CEL

EXPRESSIONS:
- $t_{\text{Step}} = 5.0 \times 10^{-5} \text{ [s]}$
- $t_{\text{Total}} = 7.5 \times 10^{-3} \text{ [s]}$
- $k_{\text{Spring}} = 300 \text{ [N m}^{-1}]$
- $\text{denBall} = 7800 \text{ [kg m}^{-3}]$
- $\text{volBall} = \pi \times (2.0 \text{ [mm]})^2 \times 10^{-4} \text{ [m]}$
- $m_{\text{Ball}} = \text{denBall} \times \text{volBall}$
- $F_{\text{Flow}} = \text{force}_y@\text{Ball}$
- $F_{\text{Grav}} = m_{\text{Ball}} \times 9.81 \text{ [m s}^{-2}]$
- $\text{velBall} = \text{areaAve(Mesh Velocity v)}@\text{Ball}$
- $d_{\text{Ball}} = \text{areaAve(Total Mesh Displacement y)}@\text{Ball}$
- $d_{\text{BallNumer}} = F_{\text{Flow}} - F_{\text{Grav}} + m_{\text{Ball}} \times \text{velBall}/t_{\text{Step}} + m_{\text{Ball}} \times d_{\text{Ball}}/t_{\text{Step}}^2$
- $d_{\text{BallDenom}} = k_{\text{Spring}} + m_{\text{Ball}}/t_{\text{Step}}^2$
- $d_{\text{BallNew}} = d_{\text{BallNumer}}/d_{\text{BallDenom}}$

END

• Assumes body is rigid and does not deform
• Can introduce springs/restoring forces
Springs, Joints, Contact ...
Many Applications ...

Engine Mount

Flexible Hose

Vortex Induced Vibration

Reed Valve

Liquid Pouring

Flexible Membrane
Automotive Fluid-Structure Interaction (FSI) Concepts, Solutions and Applications

Laz Foley, ANSYS Inc.