System Coupling 14.0 – Two-way FSI with ANSYS FLUENT and ANSYS Mechanical

ANSYS Regional Conference
Fluid-structure interaction problems encompass a wide range of applications in many different industries.

Aerospace, automotive, power generation, biomedical, etc.
Fluid-Structure Interaction

• The solution to two-way fluid-structure interaction requires co-simulation between computational fluid dynamics and structural mechanics.

• Applications such as air foil flutter, flow induced vibration from wind loading, membrane valves, pumps, elastic artery modeling and fuel tank sloshing require a two-way fluid-structure interaction solution to accurately predict the behavior of the design.
System Coupling 14.0

- 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 14.0 – A Broad Range of 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 Controls the Participant Solvers for Transient and Steady/Static Solutions

- Solution update can ONLY be done via System Coupling
- System Coupling ensures that the time duration and time step settings are consistent across all participant solvers
Setup Transient Structural Model

Setup transient structural solution, structural boundary conditions and Fluid Solid Interface
Setup Fluid Flow (FLUENT) Model

Setup transient fluid solution, fluid boundary conditions and specify System Coupling Dynamic Mesh Zone for fluid-structure interaction motion.
System Coupling Motion Type

- System Coupling motion identifies zones that may participate in System Coupling
- Allows user-defined motion to be combined with System Coupling motion
- Defaults to stationary motion type when not connected to System Coupling
Update Setup Cells for Transient Structural and Fluid Flow (FLUENT)

- State of System Coupling setup cell will be
  - Upstream data is now available for SC Setup
System Coupling Setup GUI

Outline
Chart Monitors
Details
Solution Information
Text Monitors
System Coupling Analysis Settings

- Coupling End Time
- Coupling Step Size
- Minimum Number of Iterations per Coupling Step
- Maximum Number of Iterations per Coupling Step

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Property</td>
<td>Value</td>
</tr>
<tr>
<td>2</td>
<td>Analysis Type</td>
<td>Transient</td>
</tr>
<tr>
<td>3</td>
<td>Initialization Controls</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>Coupling Initialization</td>
<td>Program Controlled</td>
</tr>
<tr>
<td>5</td>
<td>Duration Controls</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>Duration Defined By</td>
<td>End Time</td>
</tr>
<tr>
<td>7</td>
<td>End Time</td>
<td>10</td>
</tr>
<tr>
<td>8</td>
<td>Step Controls</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>Step Size</td>
<td>0.1</td>
</tr>
<tr>
<td>10</td>
<td>Minimum Iteration</td>
<td>1</td>
</tr>
<tr>
<td>11</td>
<td>Maximum Iteration</td>
<td>20</td>
</tr>
</tbody>
</table>
System Coupling Participants are Transient Structural and Fluid Flow (FLUENT)

- Region and variable information is generated automatically via Update when analysis systems are first connected to System Coupling.
- For FLUENT, all regions of type Wall are shown in SC Setup.
- For Mechanical, all regions of type Fluid Solid Interface are shown in SC Setup.
Recommended Way to Create Data Transfer Regions

- Use Ctrl key to select a FLUENT and Mechanical region pair and select Create Data Transfer from right-click pop-up menu
- Automatically fills in the details for the data transfer region
- Data transfers can be one-way (i.e. only transfer force or only transfer displacement) or two-way
Data Transfer Defines the Details for the Source, Target and Data Transfer Controls

- Participant
- Region
- Variable
- Transfer At
  - Start of Iteration only
- Under Relaxation Factor
- Convergence Target
Execution Control

- **Co-Simulation Sequence**
  - Transient or Static Structural will always be first in the co-simulation sequence

- **Debug Output**
  - Different levels of debug output for analysis and data transfers

- **Intermediate Results File Output**
  - Controls the intervals for writing restart file information

### Properties of Solution Control

<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>B</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Participant</td>
<td>Sequence</td>
</tr>
<tr>
<td>2</td>
<td>Transient Structural</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>Fluid Flow (FLUENT)</td>
<td>2</td>
</tr>
</tbody>
</table>

### Properties of Debug Output

<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>B</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Property</td>
<td>Value</td>
</tr>
<tr>
<td>2</td>
<td>Global Level</td>
<td>None</td>
</tr>
<tr>
<td>3</td>
<td>Startup</td>
<td>Use Global Level</td>
</tr>
<tr>
<td>4</td>
<td>Participant Connection</td>
<td>Use Global Level</td>
</tr>
<tr>
<td>5</td>
<td>Analysis Initialization</td>
<td>Use Global Level</td>
</tr>
<tr>
<td>6</td>
<td>Solution Initialization</td>
<td>Use Global Level</td>
</tr>
<tr>
<td>7</td>
<td>Data Transfers</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>Convergence Checks</td>
<td>Use Global Level</td>
</tr>
<tr>
<td>16</td>
<td>Shutdown</td>
<td>Use Global Level</td>
</tr>
</tbody>
</table>

### Properties of Intermediate Result Files Output

<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>B</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Property</td>
<td>Value</td>
</tr>
<tr>
<td>2</td>
<td>Output Frequency</td>
<td>At Step Interval</td>
</tr>
<tr>
<td>3</td>
<td>Step Interval</td>
<td>2</td>
</tr>
</tbody>
</table>
Executing System Coupling
Alternative Method for Executing System Coupling

- From schematic select Update using right-click menu on System Coupling solution cell
- Solution progress (% complete) can be monitored using View Progress menu
Solution Information

- Build information
- Complete summary of coupling service input file
- Analysis details
- Participant summaries
- Data transfer details
- Mapping diagnostics
- Time step and iteration summary
- Solver field equation convergence summary
- Data transfer convergence summary
- FLUENT/MAPDL solver output
Chart Monitors

Default chart monitors show convergence history for all data transfers.

X-axis can be coupling time, step or iteration.
Adding Charts and Variables

- Add charts by selecting Create Convergence Chart
- Variables can be added or removed from charts
  - Data transfers, CFD and structural convergence norms
- Chart properties are editable in same manner as other charts within ANSYS Workbench
Post Processing System Coupling

- Transient Structural or Fluid Flow (FLUENT) Results cell for solver-specific post-processing
- Add a Results System (ANSYS CFD-Post) for unified post-processing of structural and fluid results
• Oscillating Plate Verification
  – Excellent correlation between System Coupling, published data and MFX solver
System Coupling – Examples
Fuel Tank Sloshing

Transient free surface flow in a fuel tank with internal baffles.
Mitral Valve

Transient blood flow through a three leaf mitral valve, non-Newtonian fluid and anisotropic hyperelastic tissue. Solution includes re-meshing of the fluid domain and nonlinear contact.
Reed Valve

Transient response of reed valve opening and closing. Solution includes re-meshing of the fluid domain, large deformations and nonlinear contact.
Vibrating Rod

Transient response of vibrating rod including vortex shedding.
Deformation of a Leakage Path

Steady state solution for a narrow representative leakage path in a fuel injector assembly clearance gap.
Questions and Answers