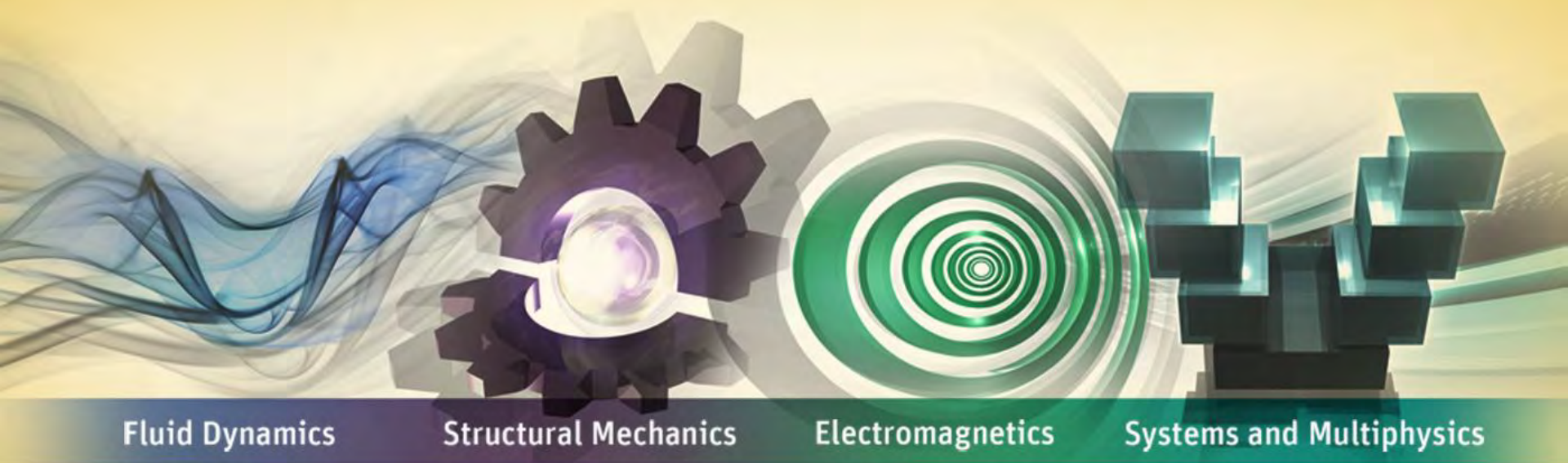


Modeling Fluid Structure Interactions

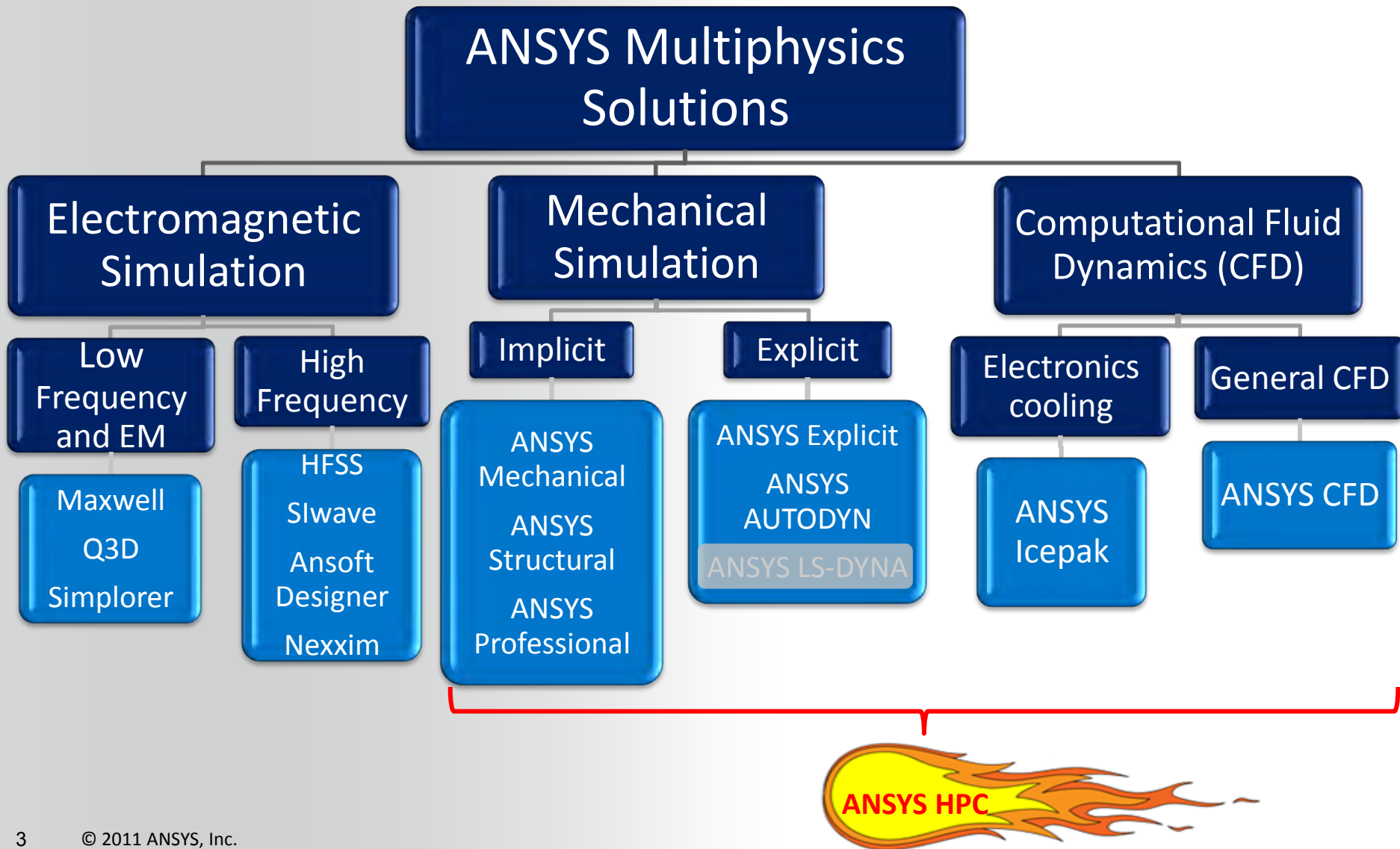


YY. Perng

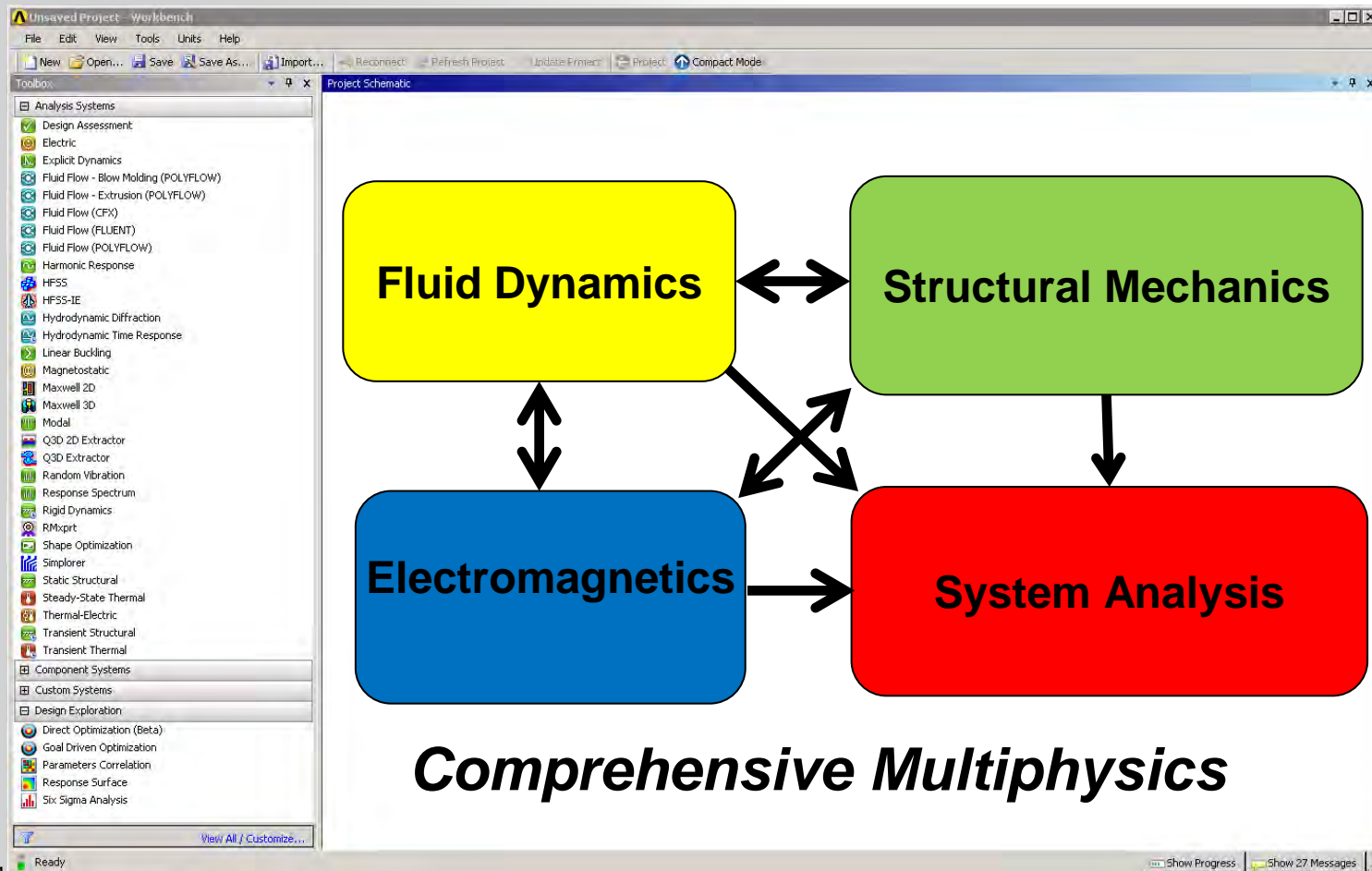
Lead Application Engineer

ANSYS, Inc.

- **What is FSI (Fluid Structure Interaction)**
- **Modes of FSI**
- **Solution Procedure of FSI**
 - **1-Way FSI**
 - **2-Way FSI**
- **System Coupling at R14.0**
- **Summary**



Seamlessly couple solver technology to simulate real-world multiphysic applications



Multiphysics Products in Workbench

ANSYS Mechanical

ANSYS FLUENT

ANSYS CFX

ANSYS CFD

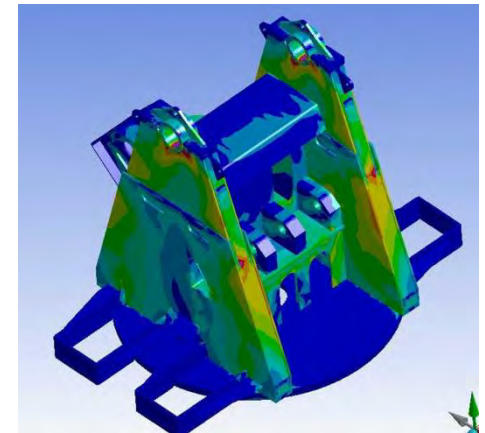
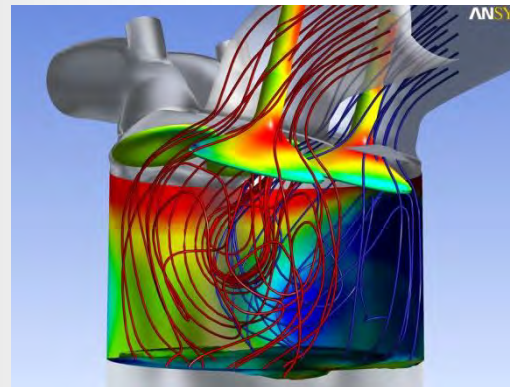
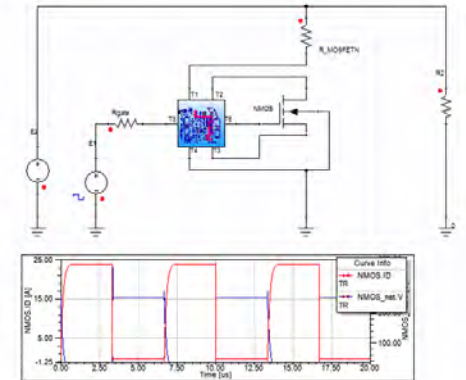
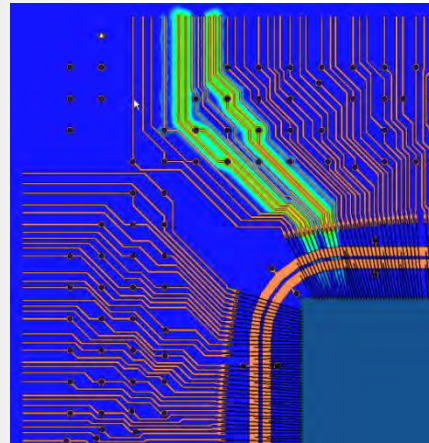
ANSYS Icepak

HFSS

Maxwell

Simplorer

SIwave



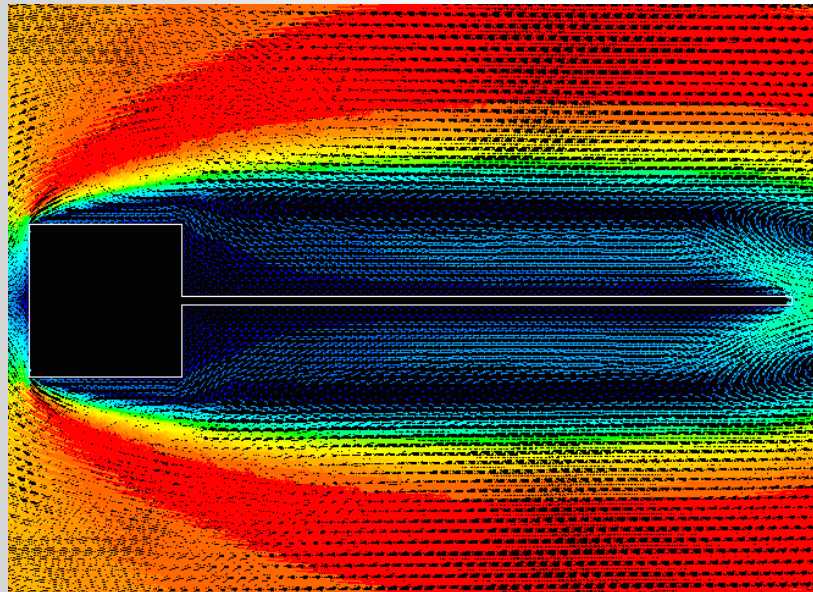
Multiphysics Integration in Workbench

	CFD	Mechanical	HFSS	Maxwell	Simplorer	Icepak	SIwave
CFD		•	•	•	•		
Mechanical	•		•	•	•	•	
HFSS	•	•			•	•	•
Maxwell	•	•			•	•	
Simplorer	•	•	•	•		•	•
Icepak		•			•		•
SIwave			•		•	•	

- **What is FSI (Fluid Structure Interaction)**
- Modes of FSI
- Solution Procedure of FSI
 - 1-Way FSI
 - 2-Way FSI
- System Coupling at R14.0
- Summary

FSI applications involve coupling of fluid dynamics and structure mechanics disciplines

- Fluid flow exerts hydrodynamic forces on a structure and deforms and/or translates the structure
- Fluid flow can also modify thermal stresses within the structure
- Deformed or translated structure imparts velocity to the fluid domain and changes its shape and thus changes the fluid flow



- What is FSI (Fluid Structure Interaction)
- **Modes of FSI**
- Solution Procedure of FSI
 - 1-Way FSI
 - 2-Way FSI
- System Coupling at R14.0
- Summary

Modes of FSI Modeling

Rigid body FSI

- Assume no deformation in the solid structures
- Only motions of solid structure in the fluid are considered
- Can be done in CFD alone

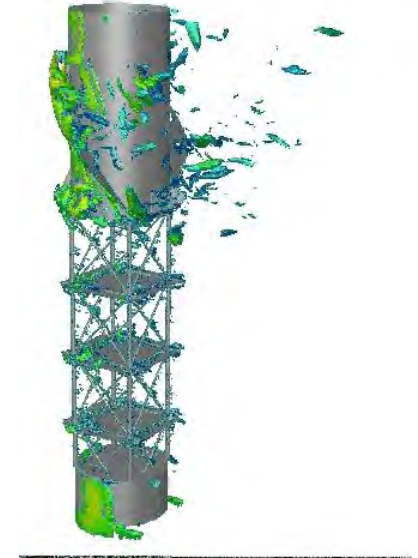
1-way FSI

- Very small deformations in the structure
- Calculate and pass flow and thermal fields from CFD to the structural analysis FEA code
- No need to update and recalculate flow

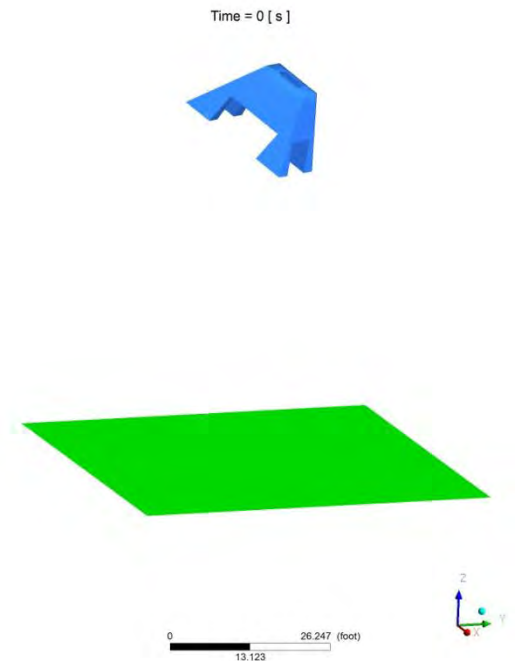
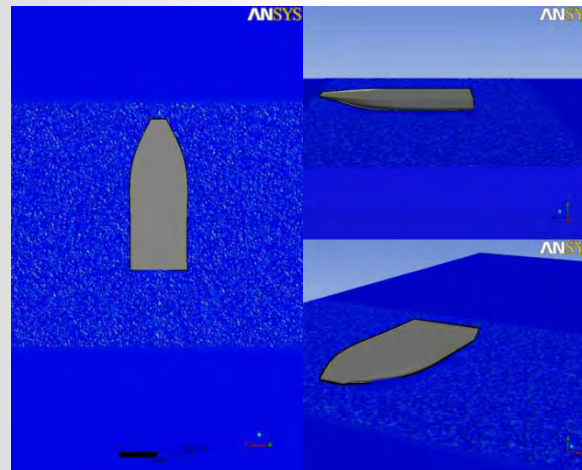
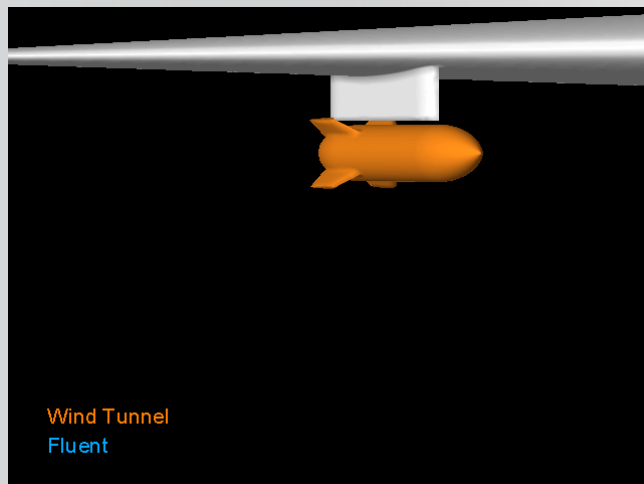
2-way FSI

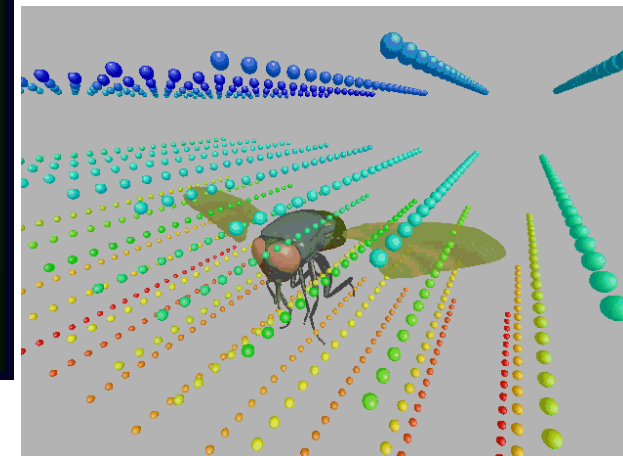
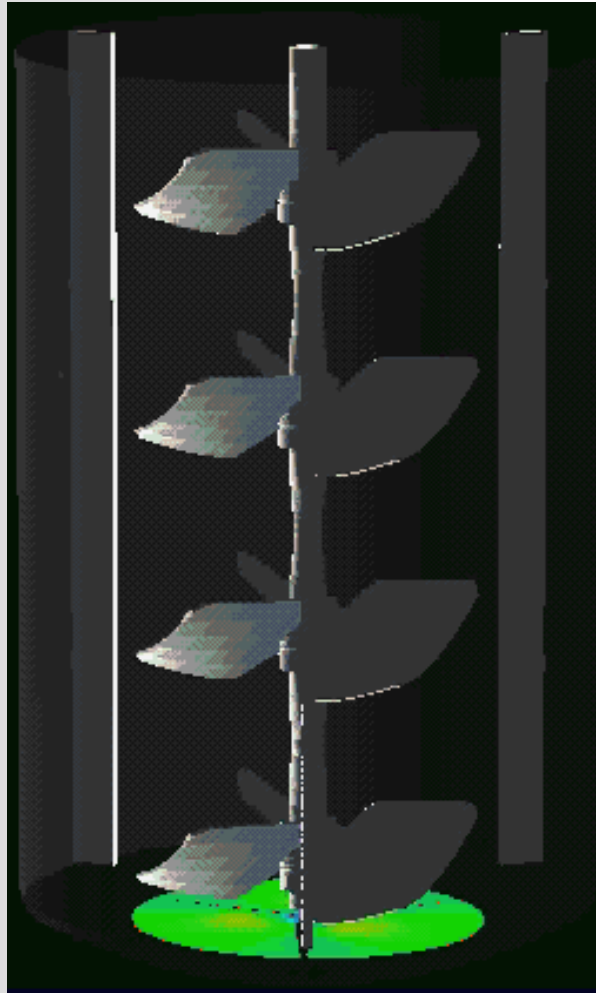
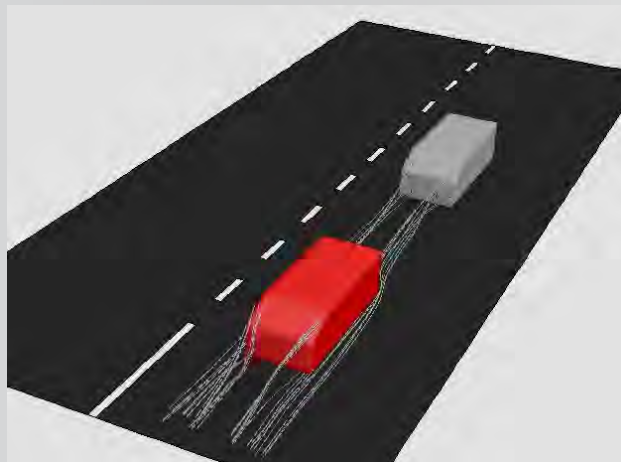
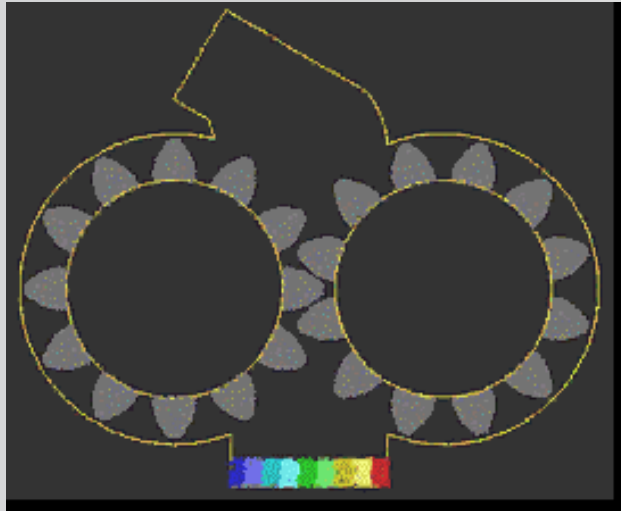
- Large structural deformations
- Iterate between CFD and FEA codes

- Solid structures are considered as rigid bodies
 - Motions of solid structures are the primary interest
 - Deformation not critical
- Can be done within CFD alone



Courtesy of Technip USA

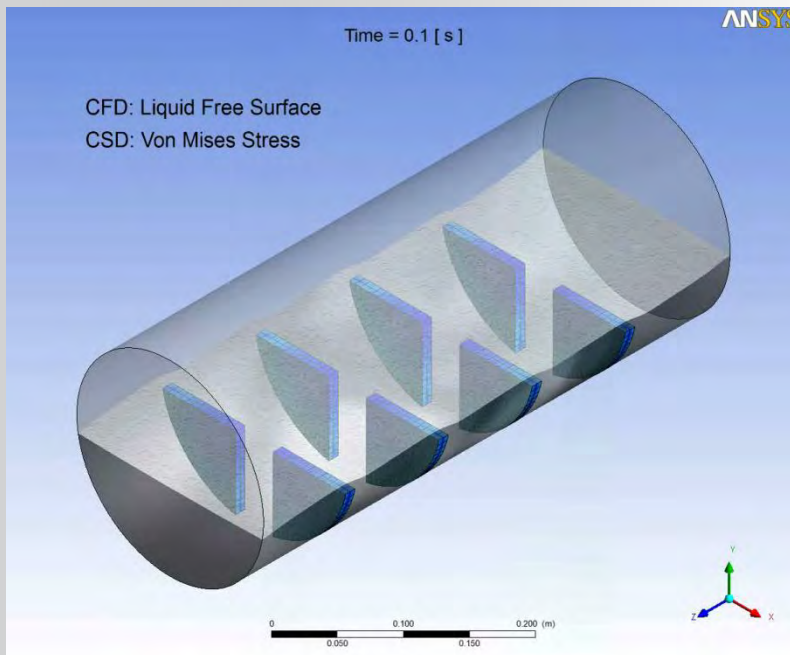




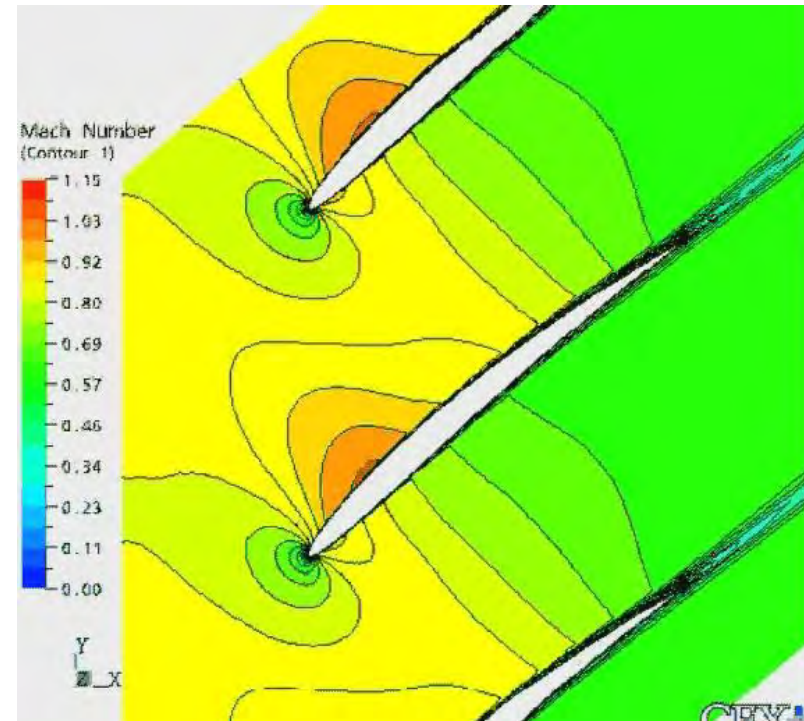
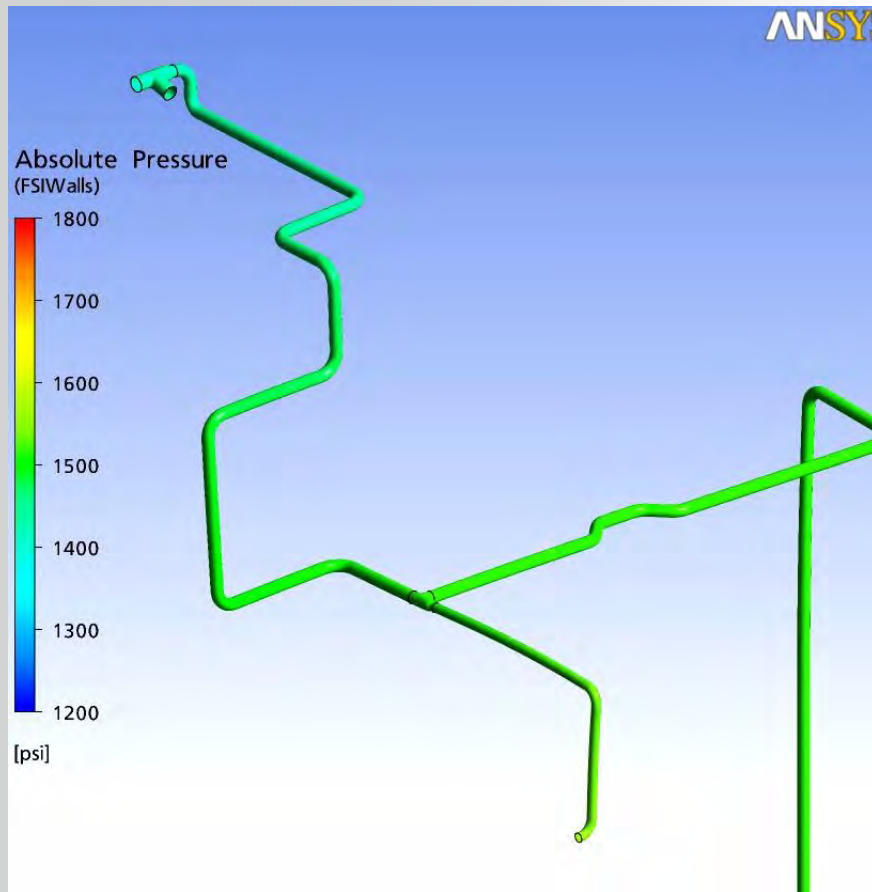
1-way FSI

Very small deformations in the structure

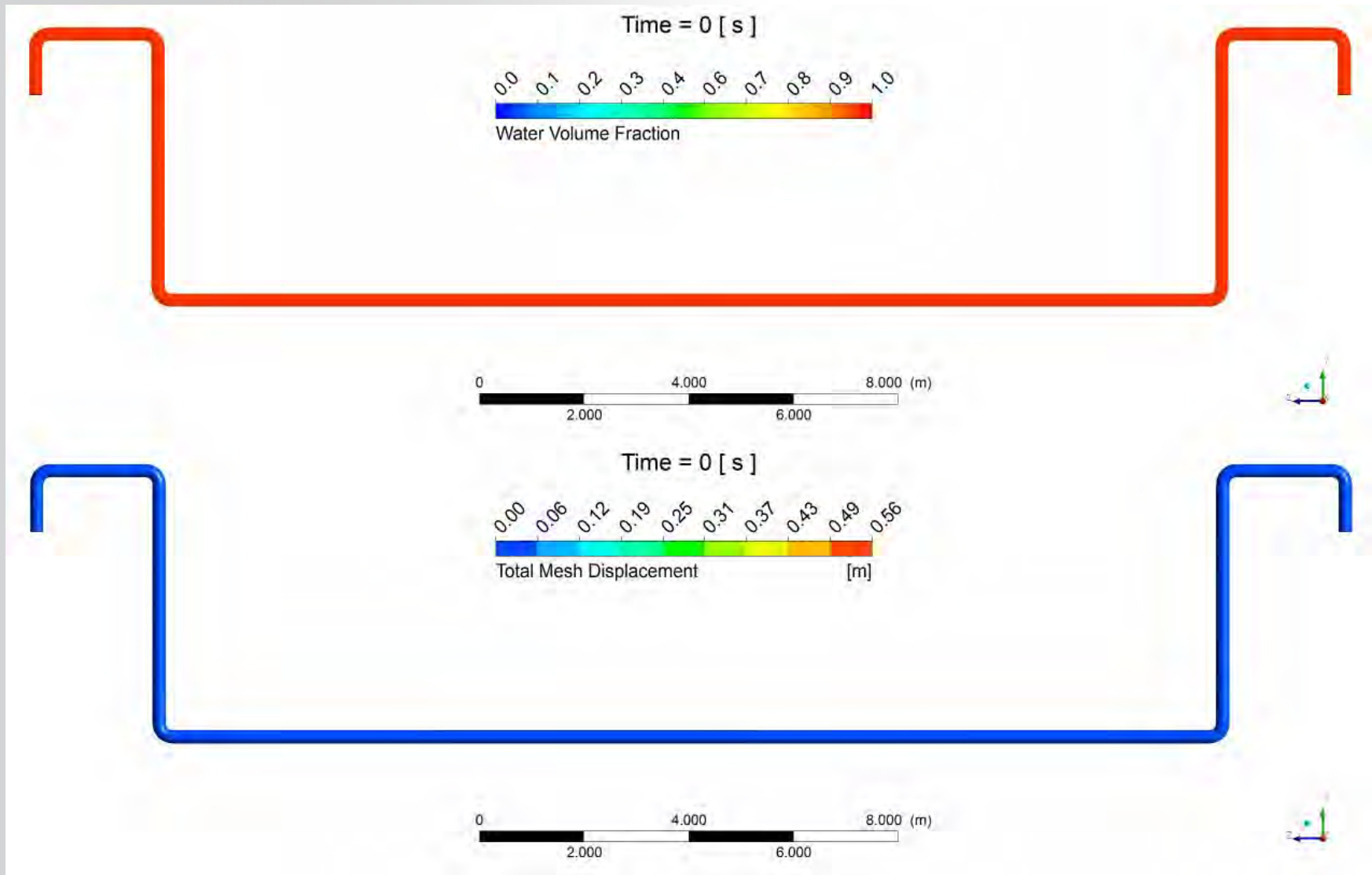
- Calculate and pass flow and thermal fields from CFD to the FEA code
- No need to update and recalculate flow



- Large deformations or transient simulations
- Iterate between CFD and FEA codes

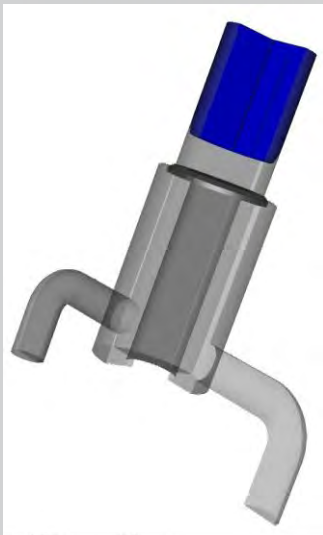
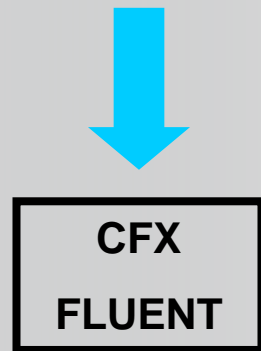


Jumper Pipe 2-way FSI - Multiphase



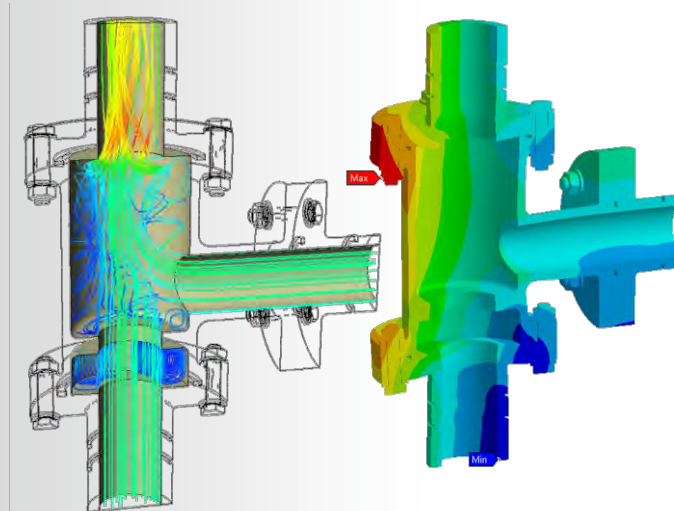
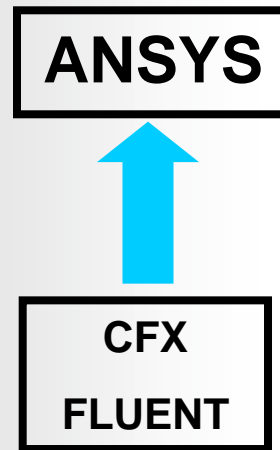
ANSYS[®] FSI in ANSYS WorkBench14.0

Rigid Body FSI



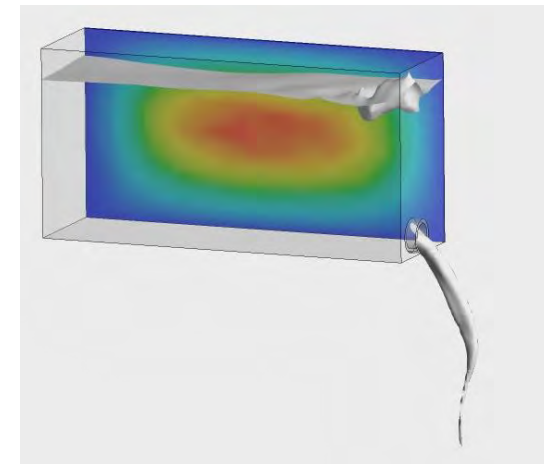
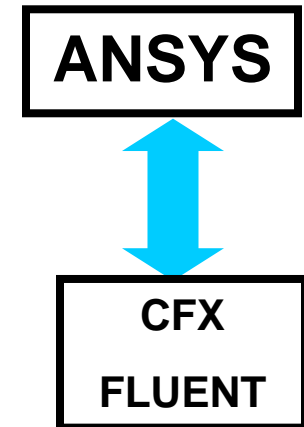
Rigid body motion

1-way FSI



One-way, common practice

2-way FSI

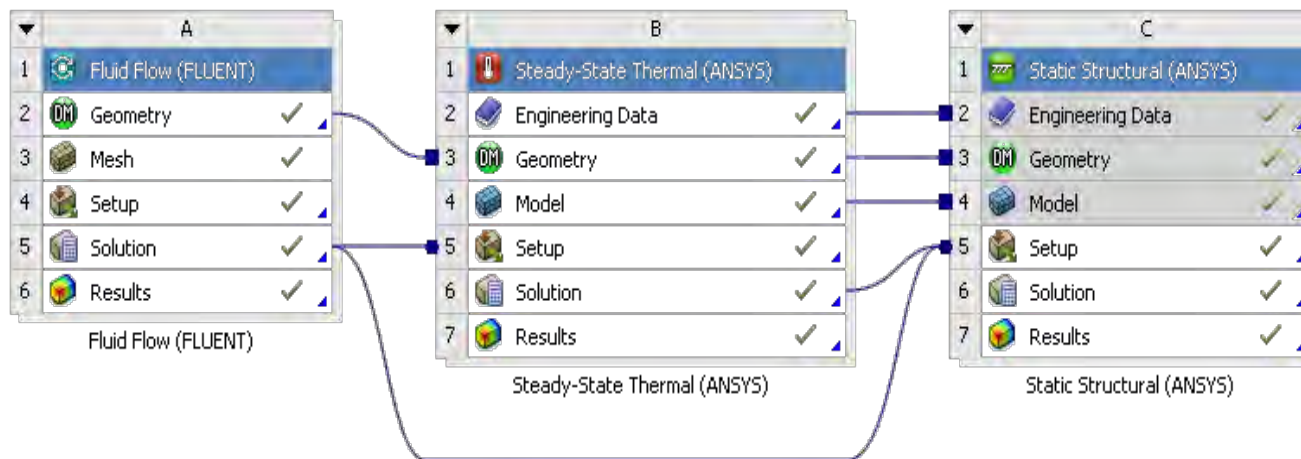
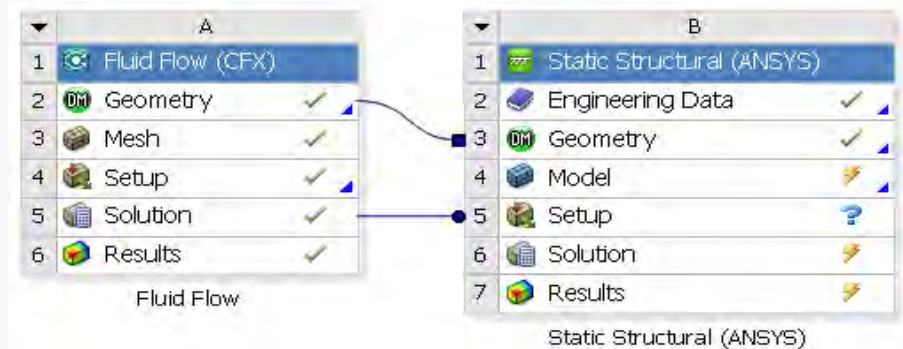


Two-way, high fidelity

- What is FSI (Fluid Structure Interaction)
- Modes of FSI
- **Solution Procedure of FSI**
 - **1-Way FSI**
 - 2-Way FSI
- System Coupling at R14.0
- Summary

1-way FSI in Workbench

- Straightforward and easy to do in Workbench
- Supports both thermal and structural loads
 - Surface Temperature
 - Volumetric Temperature
 - Heat Transfer Coefficient
 - Pressure (i.e. Stress Vector)



Set up and Solve in Mechanical

Import Pressure Load from CFD

The screenshot displays the ANSYS Mechanical interface for a Static Structural analysis. The main window shows a 3D model of a butterfly valve with an imported pressure load applied to its internal surfaces. The load is represented by a vector field of arrows. A red box highlights the 'Imported Load (Solution)' and 'Imported Pressure' entries in the Outline tree, with an arrow pointing to the 'Imported Pressure' entry in the Details panel.

Static Structural (ANSYS) Outline:

- Model (B4)
 - Geometry
 - Coordinate Systems
 - Symmetry
 - Connections
 - Mesh
 - Named Selections
 - Static Structural (B5)
 - Analysis Settings
 - Fixed Support
 - Frictionless Support
 - Imported Load (Solution)
 - Imported Pressure
 - Solution (B6)
 - Solution Information
 - Total Deformation
 - Equivalent Stress
 - Moment Reaction
 - Force Reaction

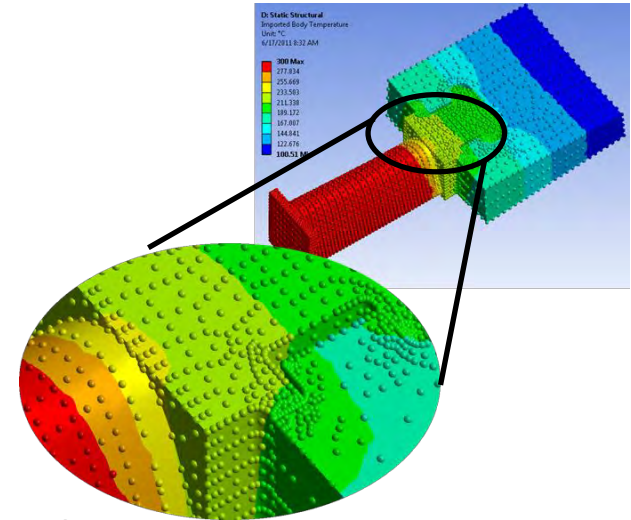
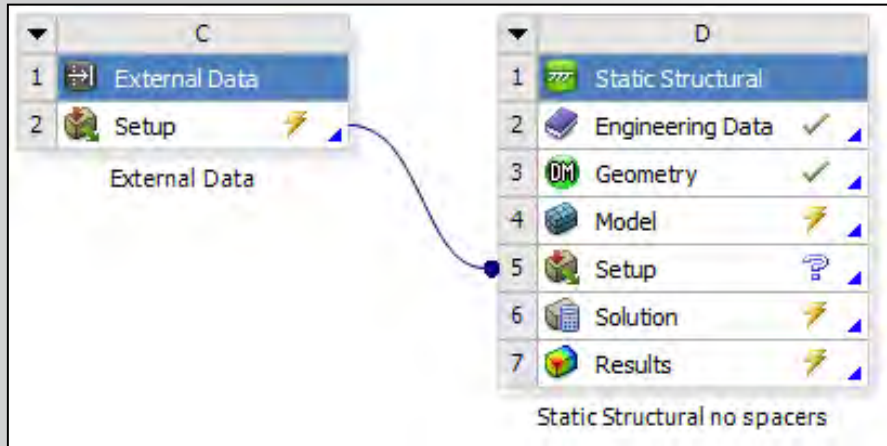
Details of "Imported Pressure"

- Scope**
 - Scoping Method: Geometry Selection
 - Geometry: 7 Faces
- Definition**
 - Type: Imported Pressure
 - Suppressed: No
- Transfer Definition**
 - CFD Surface: valve
- CFD Data**
 - CFD Results File: C:\users\yyp\demo\butterflyvalve\bfv-param1...

Worksheet: Imported Pressure

Source Time (s)	Source Time Step	Analysis Time (s)	Scale	Offset (MPa)
1	0.	0.	1	0

Static Data Transfers via External Data

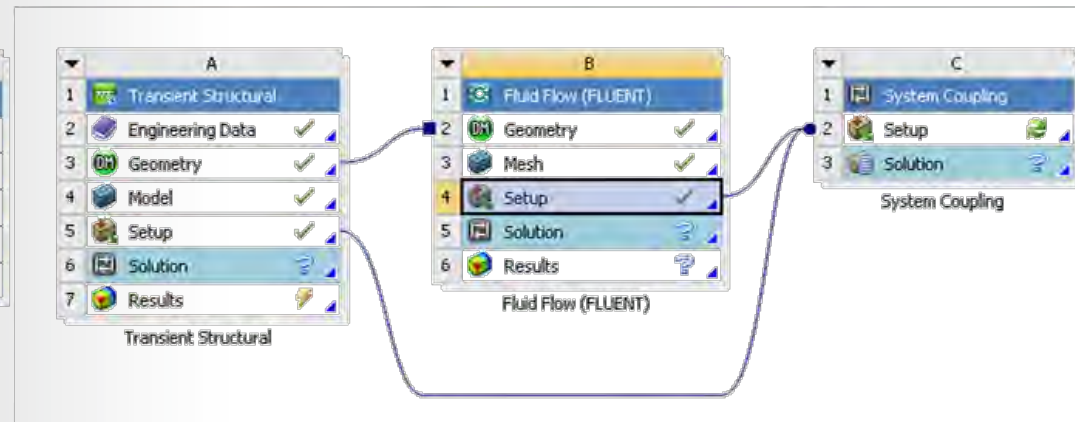
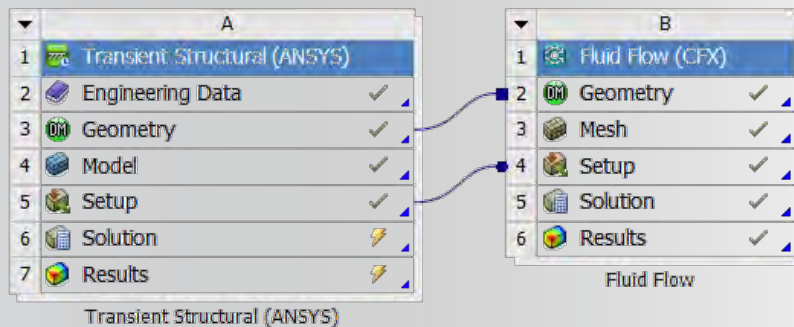


- 1-way connection to Mechanical systems only
- Supports volumetric/surface Temperature, Pressure (force vector), HTC, Heat Flux, volumetric Heat Generation, Shell Thickness and Displacement^β
- Source files for External Data can be exported from Fluent or any other package (or text editor/spreadsheet)
- Easy to use via the Workbench schematic
 - Must re-export and re-read files in External Data after making a change

- What is FSI (Fluid Structure Interaction)
- Modes of FSI
- **Solution Procedure of FSI**
 - 1-Way FSI
 - **2-Way FSI**
- System Coupling at R14.0
- Summary

ANSYS 2-way FSI Solutions in Workbench

- Multi-field solver – multiple code coupling (MFX) technology
 - **ONLY** for **CFX-Mechanical** coupling
 - Requires **ANSYS Mechanical License**
 - Mapping through CFD-post
 - No support for MPC
 - Need command snippets
- System Coupling
 - **ONLY** for **Fluent-Mechanical** coupling
 - Requires **ANSYS Structural License**
 - New coupling/mapping infrastructure
 - Easy restart
 - Supports MPC



General 2-way FSI Setup Procedure

Step 1:

- Set up project and share geometry

Step 2:

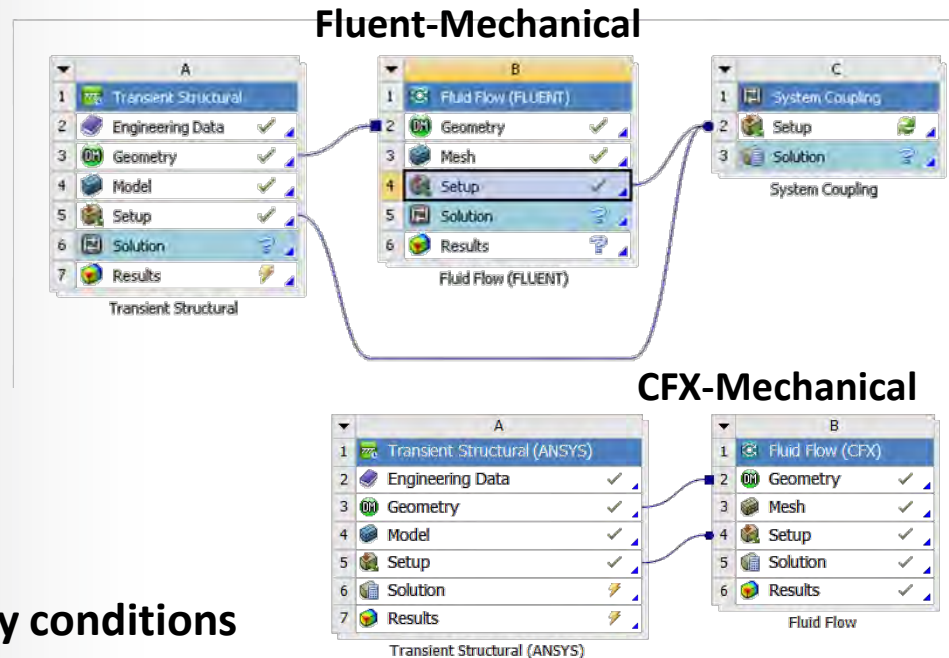
- Set up Mechanical model
 - Material model
 - Loads and constraints
 - Identify the Fluid-Solid Interface

Step 3:

- Set up CFD model
 - Flow properties and flow boundary conditions
 - Identify the Fluid-Solid Interface and specify mesh motion model

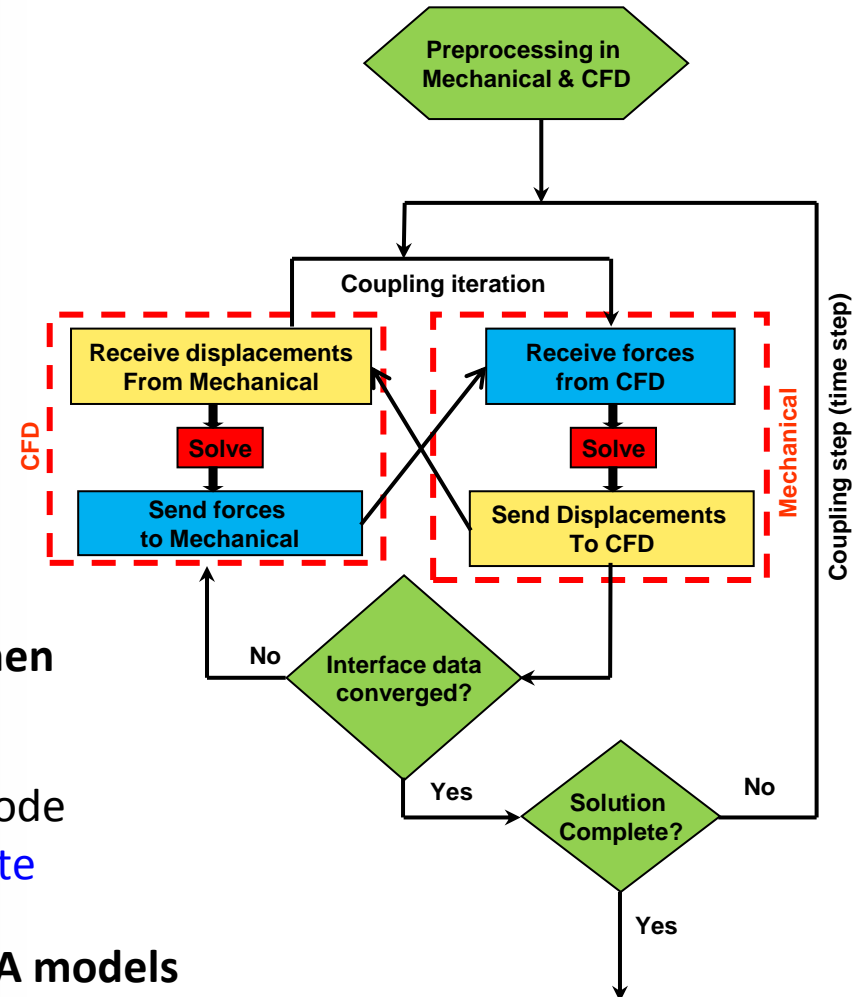
Step 4:

- Set up execution controls and solve (in CFX-pre or SC settings)
 - Time duration and time steps
 - Coupling sequence
 - Number of coupling iterations per time step
 - Interface exchange under-relaxation and convergence criteria



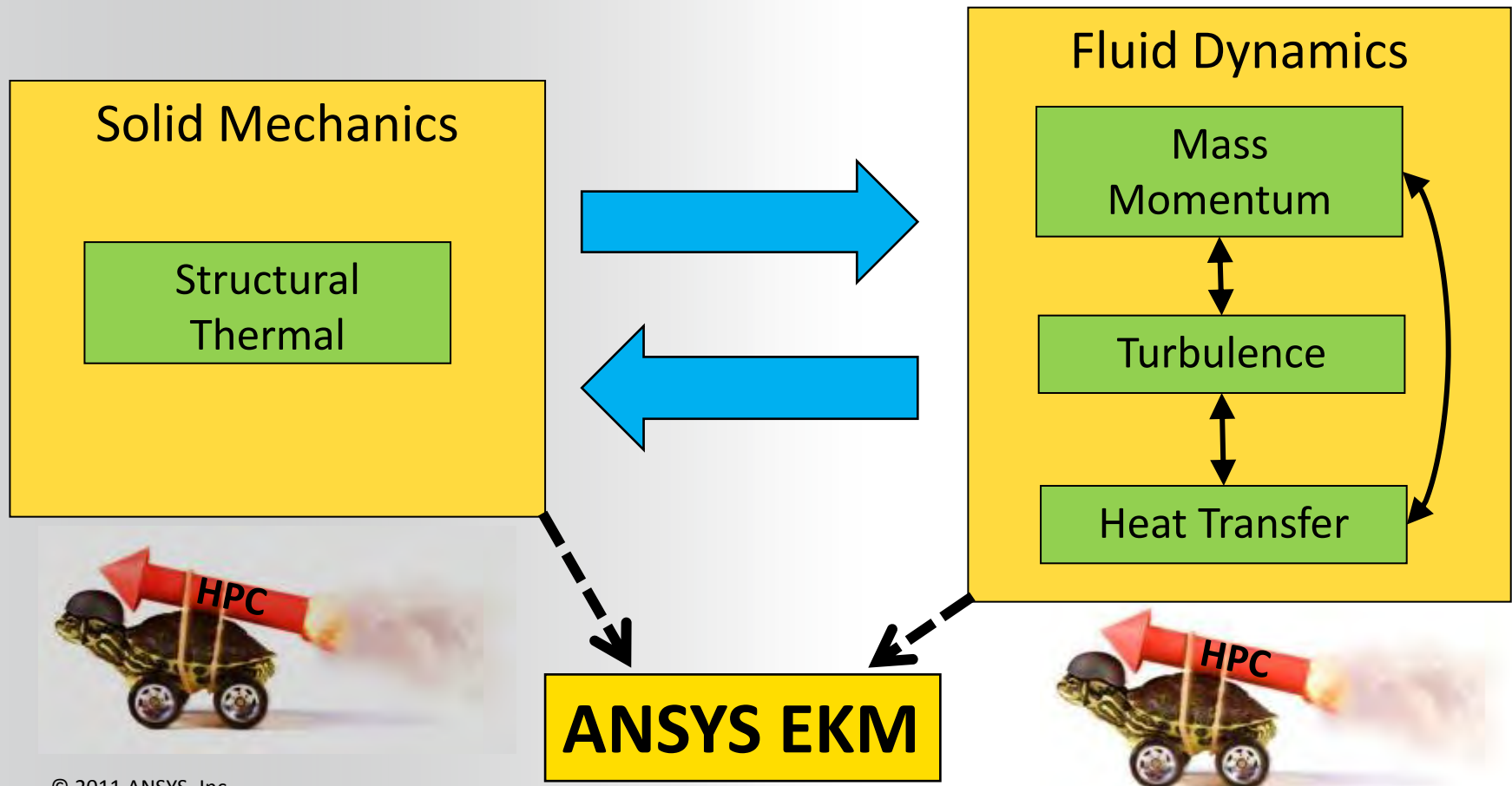
2-way FSI Solution Procedure

- **Ideal situation: structural and fluid equations are assembled and solved simultaneously**
 - Very limited capability in commercial software to date!
 - Need to make FEA only and CFD only software to talk to each other
- **Physics are coupled by passing loads across fluid-structure interfaces**
 - CFD → FEA: forces on structure surfaces
 - FEA → CFD: displacements of solid structures
- **Individual physics are solved separately and then coupled *sequentially* or *simultaneously* until equilibrium is reached**
 - Solve each physics in serial or in parallel mode
 - *No need to fully converge each intermediate solution*
- **ANSYS streamlines the coupling of CFD and FEA models**



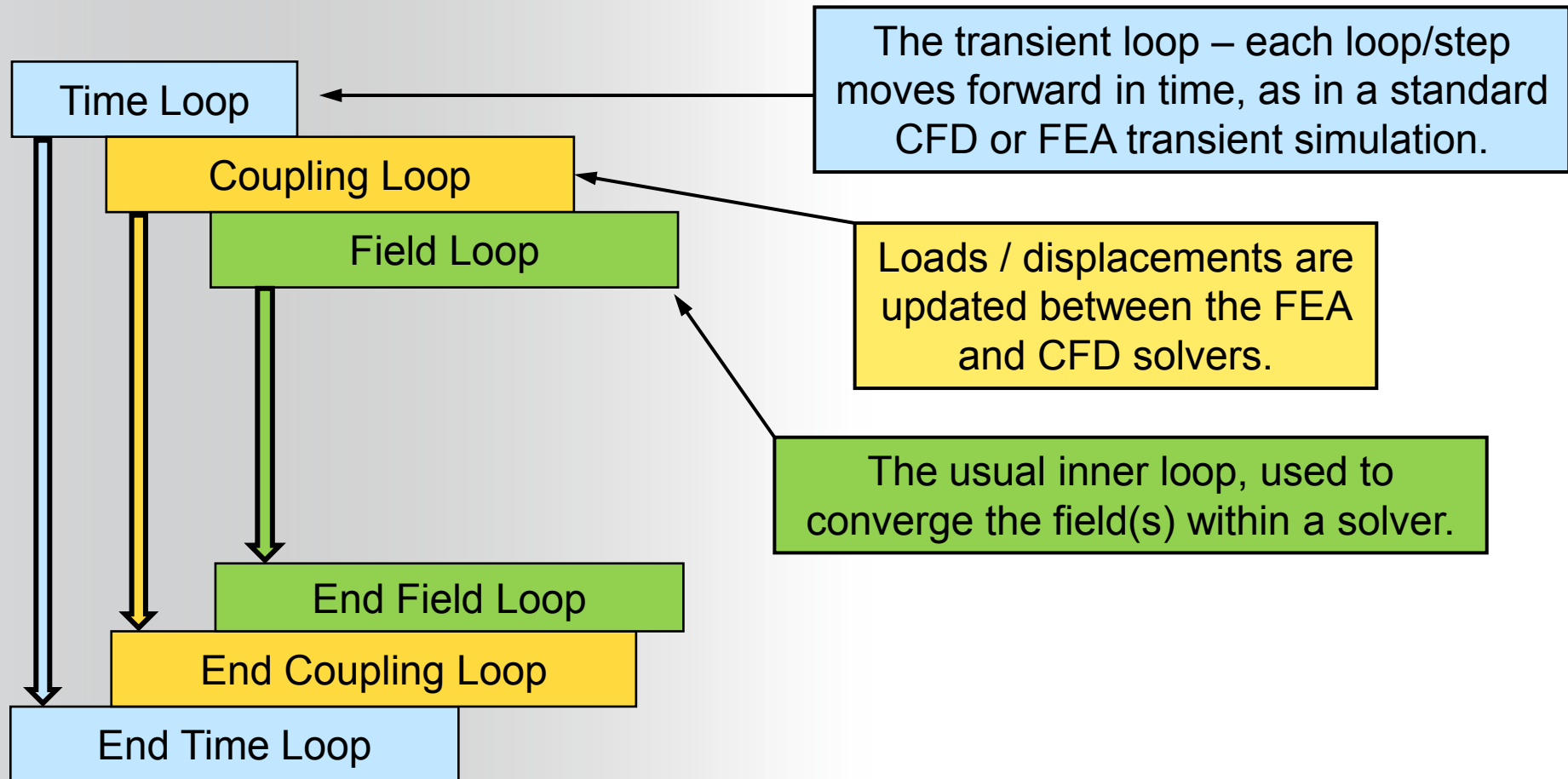
Iterative Coupling

- Iterations are required to converge the quantities transferred between the Mechanical and CFD solvers
 - Just like iterations are required to converge segregated fields *within* the CFD or FEA solvers



Iterative Coupling

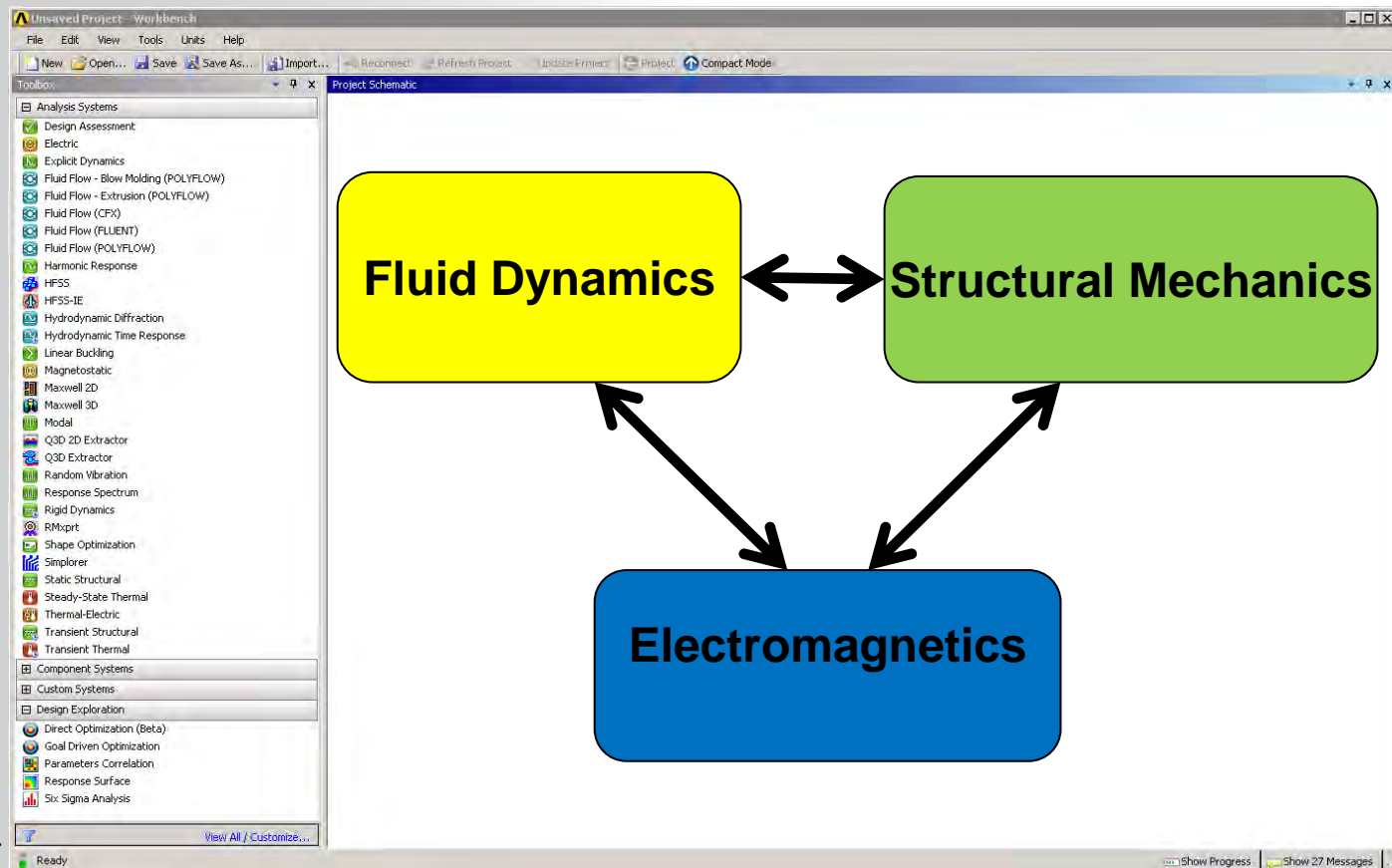
A transient 2-way FSI simulation has three levels of iterations:



- What is FSI (Fluid Structure Interaction)
- Modes of FSI
- Solution Procedure of FSI
 - 1-Way FSI
 - 2-Way FSI
- **System Coupling at R14.0**
- Summary

System Coupling: General Motivation

- Customers require coupling of various solvers and systems in the solution of coupled, multi-physics problems
- Customers don't want to learn how to do couplings differently using various physics solvers

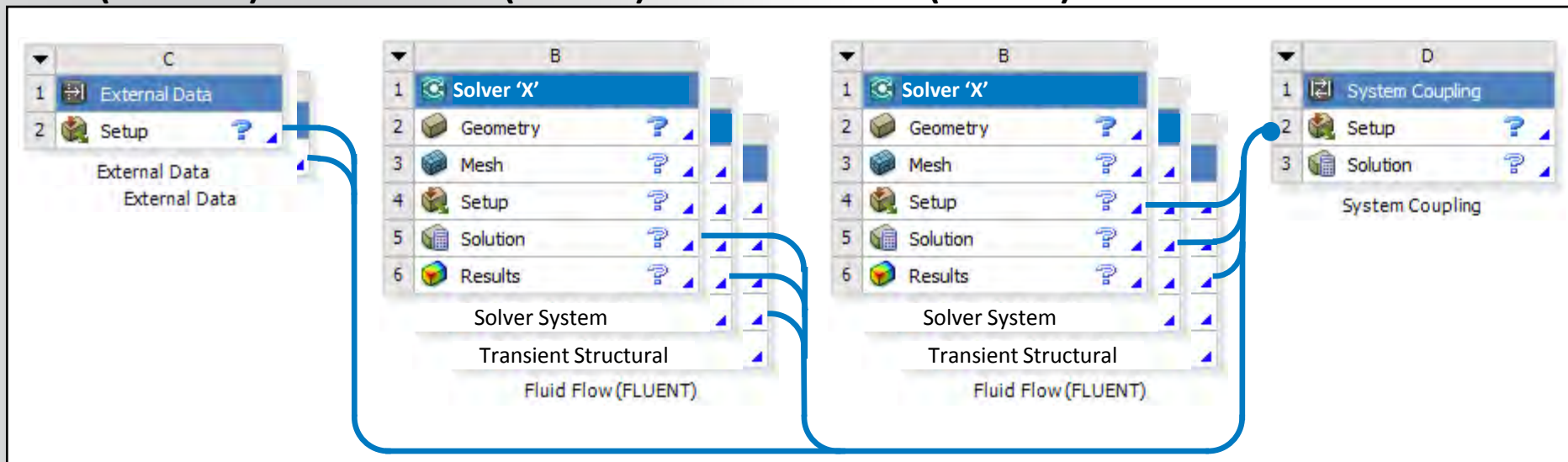


Generalized Vision for System Coupling in Workbench

Coupling Source Data (External)

Coupling Source Data (Solvers)

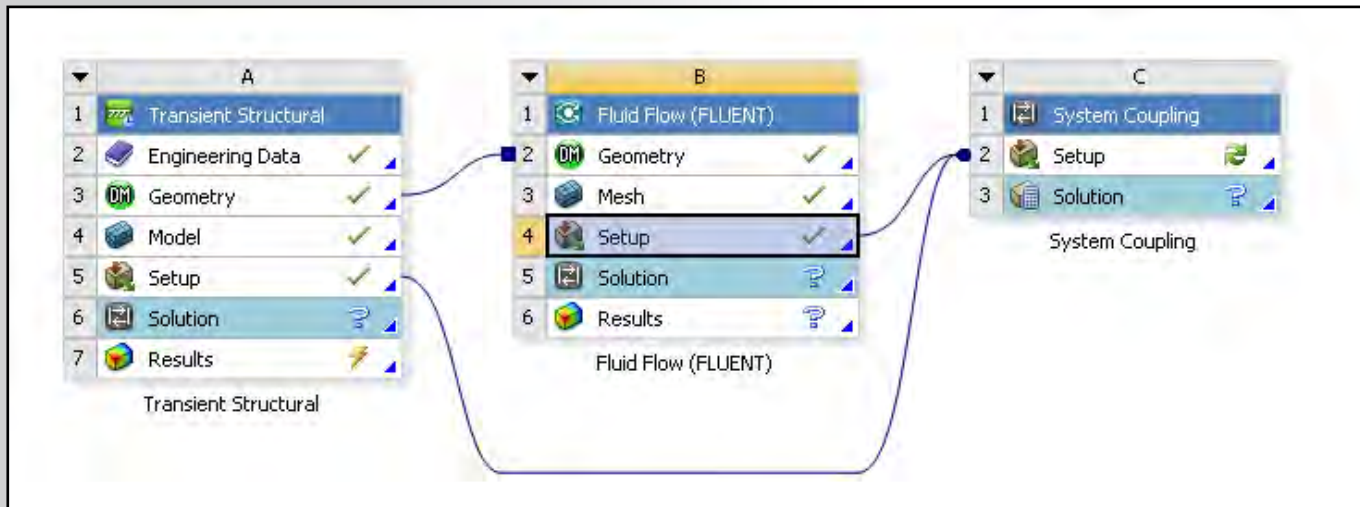
Co-Simulation (Solvers)



Future Releases

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
- Stand-alone coupling service delivers coupling management and mapping/interpolation
- Service and solvers communicate using proprietary TCP/IP client-server Remote Procedure Call (RPC) library and Standard Interfaces



- Two-way surface (3D) **force/displacement** coupling between ANSYS Fluent and ANSYS Mechanical
 - Steady/static and transient two-way FSI (co-simulation)
 - Note Fluent sends **gauged pressure** + viscous forces by default
- Workbench based setup and execution
 - Windows (32/64-bit) and Linux (64-bit)
- Execution from command line outside of Workbench including cross-platform
- Integrated post-processing with ANSYS CFD-Post
- Parallel processing for both Fluent and Structural/Mechanical solutions with ANSYS HPC
 - Compatible with DANSYS
- Easy restarts for fluid-structure interaction runs
- Parameterization, design exploration and optimization

System Coupling 14.0: Supported Capabilities

- Minimum ANSYS license must be **ANSYS Structural**
- Low and high order solid/shell elements in MAPDL
 - SOLID185/186/187, SHELL181/281, SOLSH190
- Compatible with multi-point constraints (MPC)
 - Springs, joints etc. can be created in WB Mechanical UI
- All triangular and quad boundary meshes in ANSYS Fluent
- Compatible with all mesh motion types in ANSYS Fluent
- Mapping is fully conservative
 - General Grid Interface (GGI) algorithm for force and profile preserving algorithm for displacement

System Coupling 14.0: Unsupported Capabilities

- No static data transfers (only co-simulation)
- No support for thermal data transfers
- No support for Fluent polyhedral or cut-cell meshes
- No Remote Solver Manager (RSM) support even though RSM is supported with individual solvers
- No support for 2D or 2D-Axisymmetric analysis
- No re-meshing of FSI interface in ANSYS Fluent (i.e. local face re-meshing)
 - Data mapping is fixed, not dynamic
- Cannot specify multiple load steps in ANSYS Mechanical even though MAPDL solver treats each coupling step as a load step
- **At Release 14.0, except ANSYS Fluent and ANSYS Mechanical, no other solvers can participate in System Coupling**

Setup Transient/Static Structural Model

A: Transient Structural - Mechanical [ANSYS Multiphysics]

File Edit View Units Tools Help

Show Vertices Wireframe Edge Coloring Thicken Annotations Show Mesh Show Coordinate Systems

Environment Inertial Loads Supports Conditions Direct FE

Outline

Project

- Model (A4)
 - Geometry
 - solid
 - Part
 - fluid
 - fluid
 - fluid
 - fluid
 - fluid
 - Coordinate Systems
 - Connections
 - Mesh
 - Edge Sizing
 - Edge Sizing 2
 - Edge Sizing 3
 - Named Selections
 - Transient (A5)
 - Initial Conditions
 - Modal (None)
 - Analysis Settings
 - Fixed Support
 - Pressure
 - Solution (A6)
 - Solution Information
 - Directional Deformation

Details of "Transient (A5)"

Definition

Physics Type	Structural
Analysis Type	Transient
Solver Target	Mechanical APDL

Options

Environment Temperature 22 °C

Generate Input Only

Setup structural solution, structural boundary conditions and Fluid-Solid Interface

Insert

- Go To
- Suppress Body
- Suppress All Other Bodies
- Unsuppress All Bodies
- Invert Suppressed Body Set
- Hide Body
- Hide All Other Bodies
- Hide Face(s)
- Isometric View
- Set
- Restore Default
- Zoom To Fit
- Cursor Mode
- View
- Look At
- Create Coordinate System
- Create Named Selection
- Select All

Acceleration

Standard Earth Gravity

Rotational Velocity

Pressure

Hydrostatic Pressure

Force

Remote Force

Moment

Joint Load

Fluid Solid Interface

Fixed Support

Displacement

Remote Displacement

Velocity

Frictionless Support

Compression Only Support

Elastic Support

Constraint Equation

Commands

Geometry Print Preview Report Preview

Graph

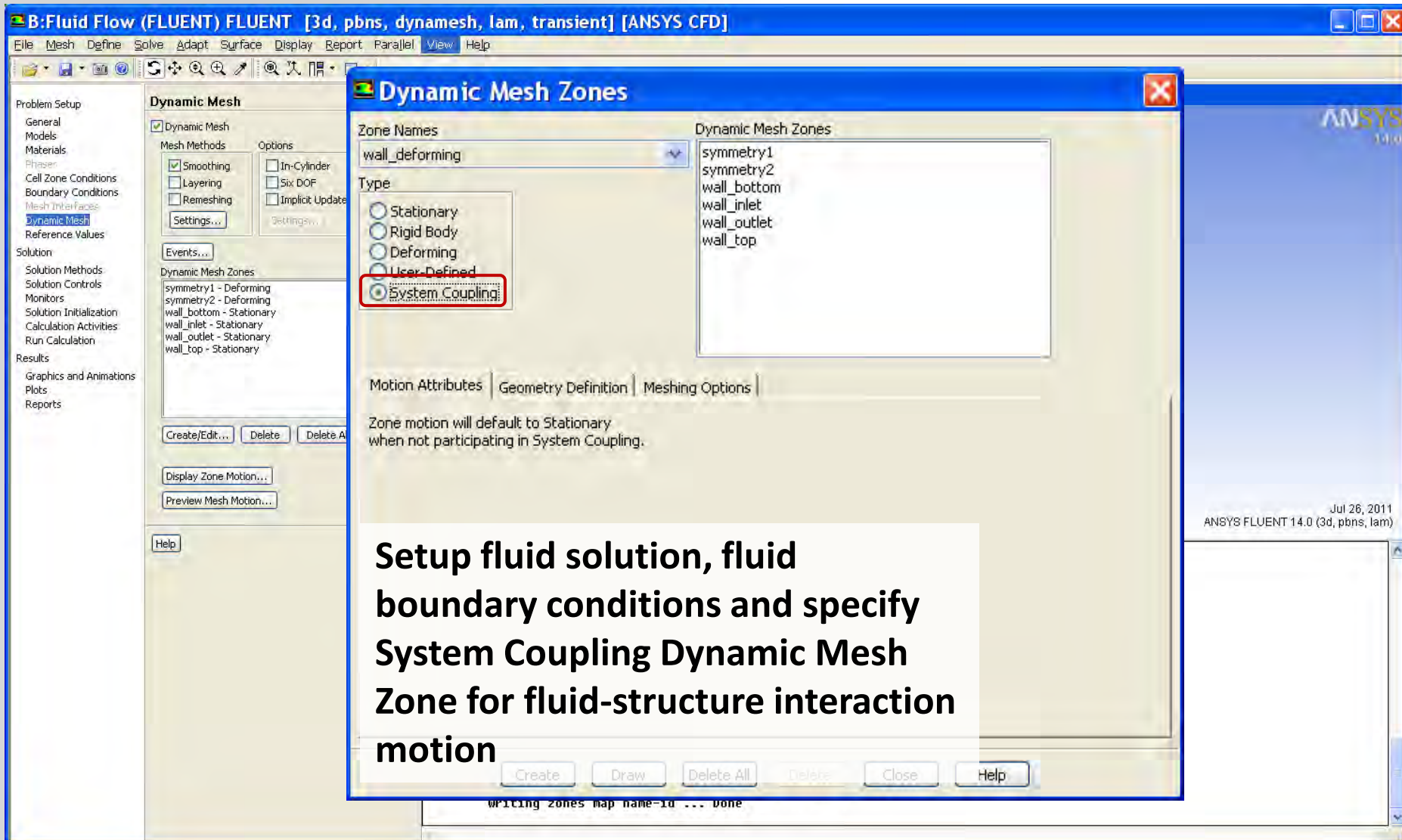
Press F1 for Help

No Messages

3 Faces Selected: Surface Area(approx.) = 0.824 m²

Metric (m, kg, N, s, V, A) Degrees rad/s Celsius

Setup Fluid Flow (FLUENT) Model



System Coupling GUI

Fluid Flow (FLUENT)

1	Fluid Flow (FLUENT)
2	Geometry
3	Mesh
4	Setup
5	Solution
6	Results

Transient Structural

1	Transient Structural
2	Engineering Data
3	Geometry
4	Model
5	Setup
6	Solution
7	Results

System Coupling

1	System Coupling
2	Setup
3	Solution

Context Menu for System Coupling Setup:

- Edit...
- Duplicate
- Transfer Data From New
- Update
- Clear Generated Data
- Refresh
- Reset
- Rename
- Properties
- Quick Help

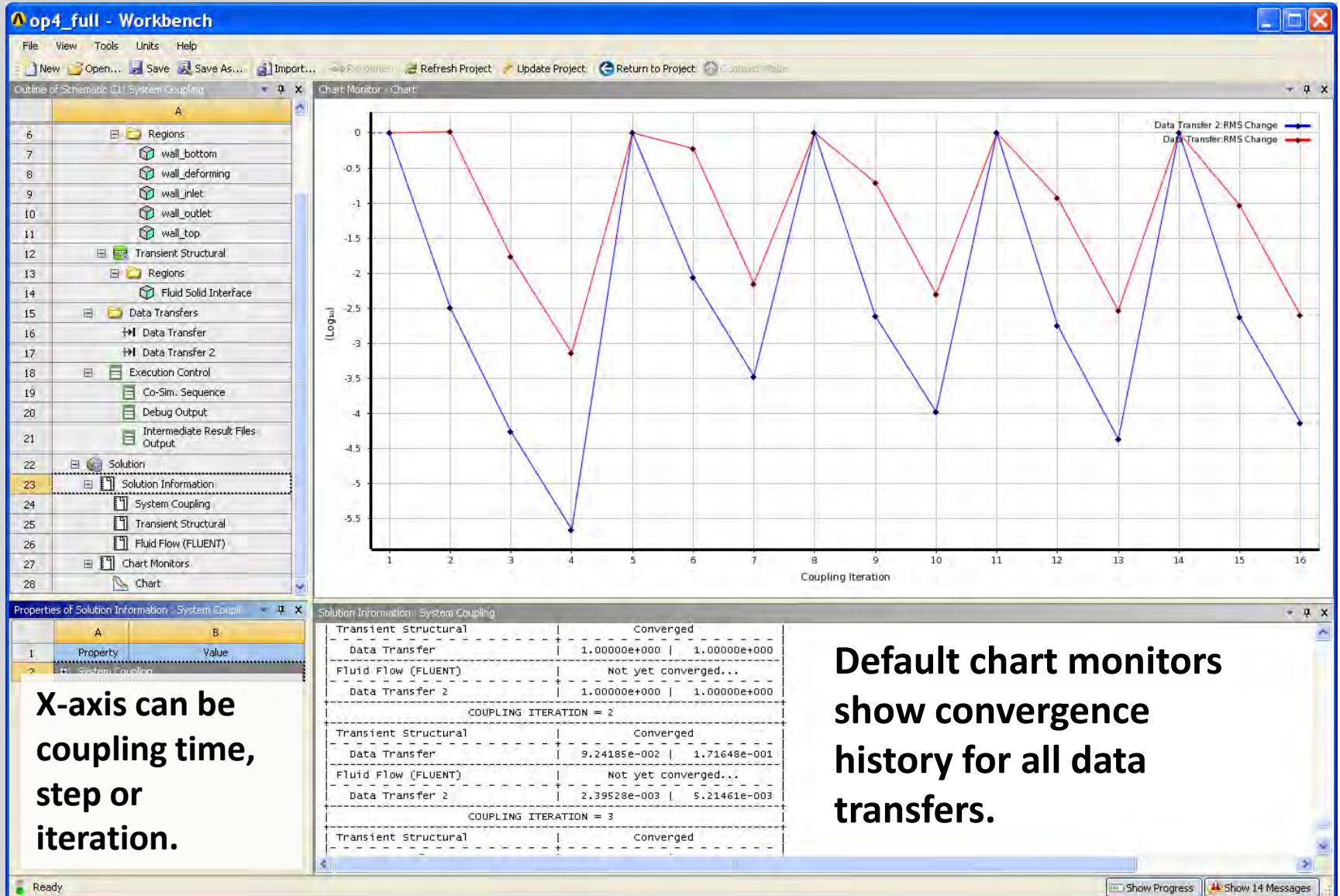
Properties of Analysis Settings

	A	B
1	Property	Value
2	Analysis Type	Transient
3	Initialization Controls	
4	Coupled	Controlled
5	Duration	
6	Duration	
7	End Time	
8	Step Controls	
9	Step Size	0.1
10	Minimum Iteration	1
11	Maximum Iteration	20

Solution Information

Details

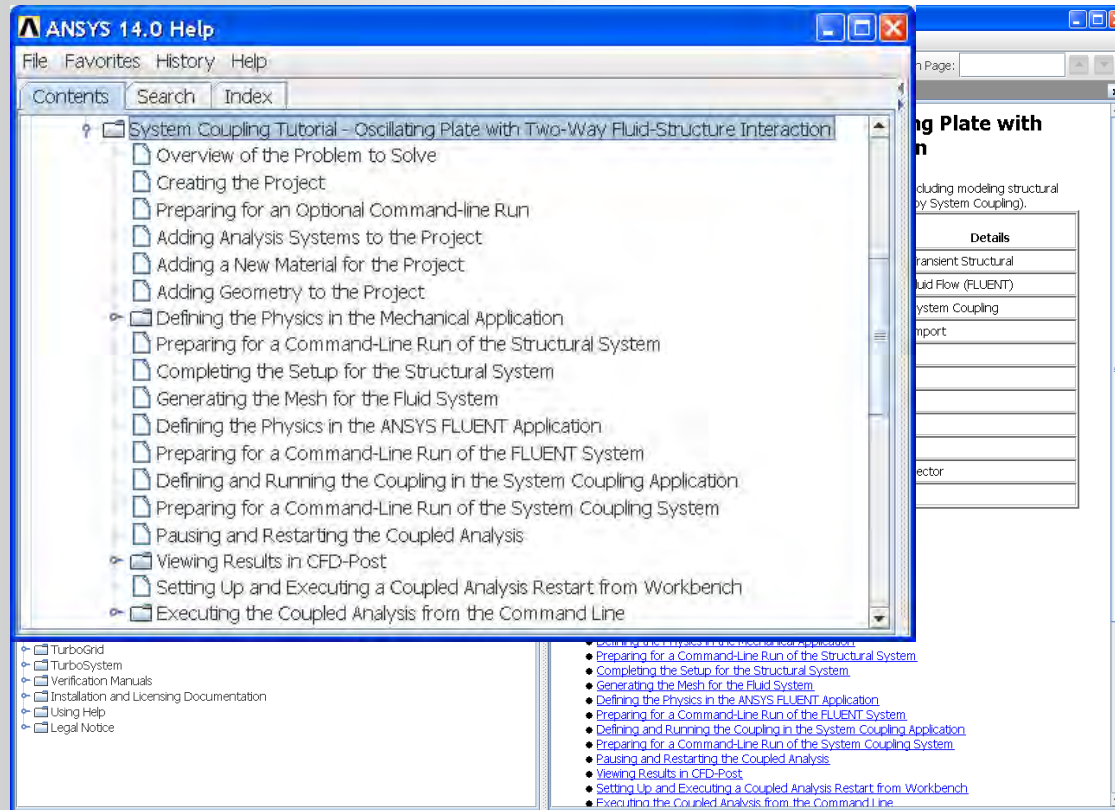
Solution Information Text Monitors



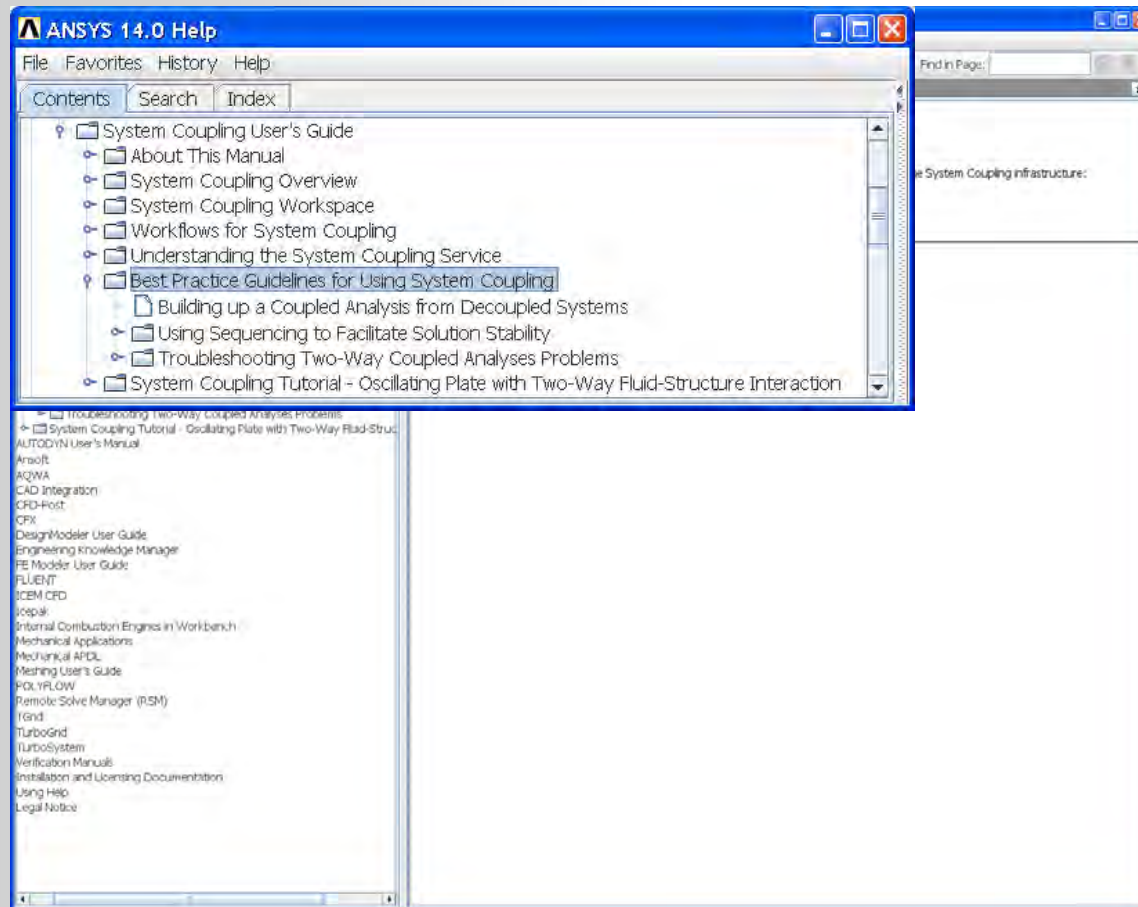
- Build information
- Complete summary of coupling service input file
- Analysis details
- Participant summaries
- Data transfer details
- Mapping diagnostics
- Time step and iteration summary
- Solver field equation convergence summary
- Data transfer convergence summary
- Fluent/MAPDL solver output

Solution			
MAPPING SUMMARY			
Data Transfer	Percent Target Nodes	(Source) Mapped Area	
Data Transfer	100 (100)	100 (85)	
Data Transfer 2	100		
TIME STEP = 1 SIMULATION TIME = 1.00000E-001			
Solver	Solution Status		
Data Transfer	RMS Change	Maximum Change	
COUPLING ITERATION = 1			
Transient Structural	Converged		
Data Transfer	1.00000e+000	1.00000e+000	
Fluid Flow (FLUENT)	Not yet converged...		
Data Transfer 2	1.00000e+000	1.00000e+000	
COUPLING ITERATION = 2			
Transient Structural	Converged		
Data Transfer	1.03656e+000	1.44434e+000	
Fluid Flow (FLUENT)	Not yet converged...		
Data Transfer 2	3.22436e-003	4.20818e-003	
COUPLING ITERATION = 3			
Transient Structural	Converged		
Data Transfer	1.73125e-002	2.74638e-002	
Fluid Flow (FLUENT)	Converged		
Data Transfer 2	5.51803e-005	1.07798e-004	
COUPLING ITERATION = 4			
Transient Structural	Converged		
Data Transfer	7.35560e-004	1.24824e-003	
Fluid Flow (FLUENT)	Converged		
Data Transfer 2	2.23482e-006	3.99582e-006	

System Coupling Tutorial



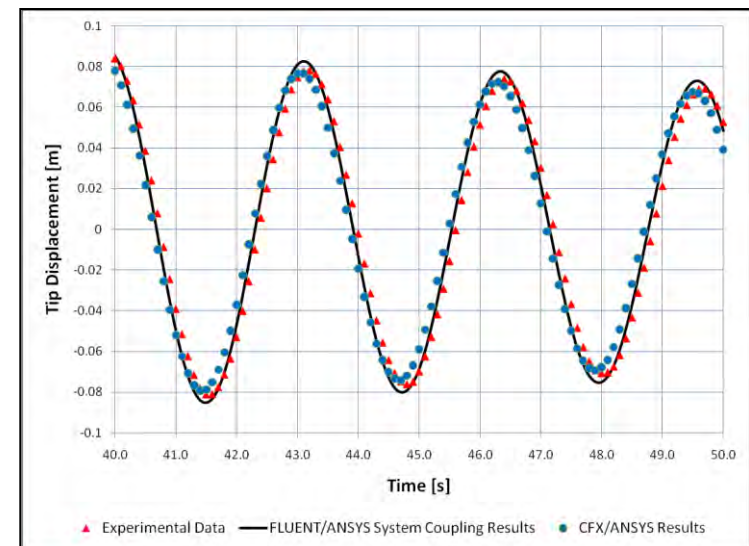
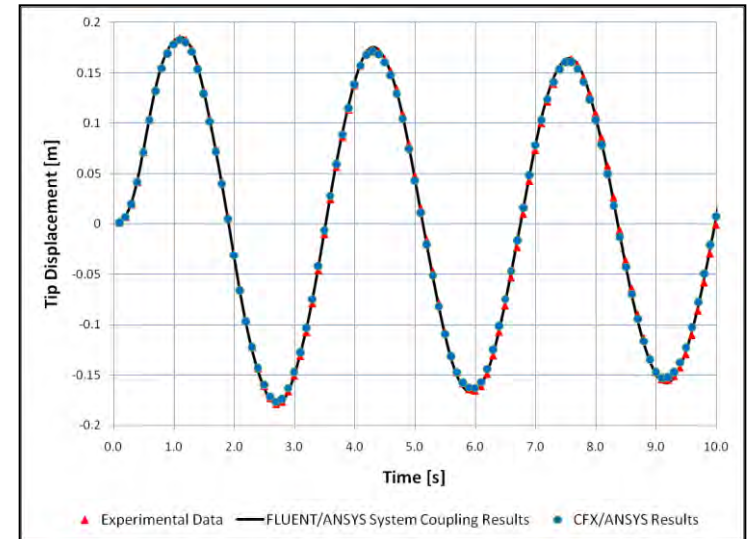
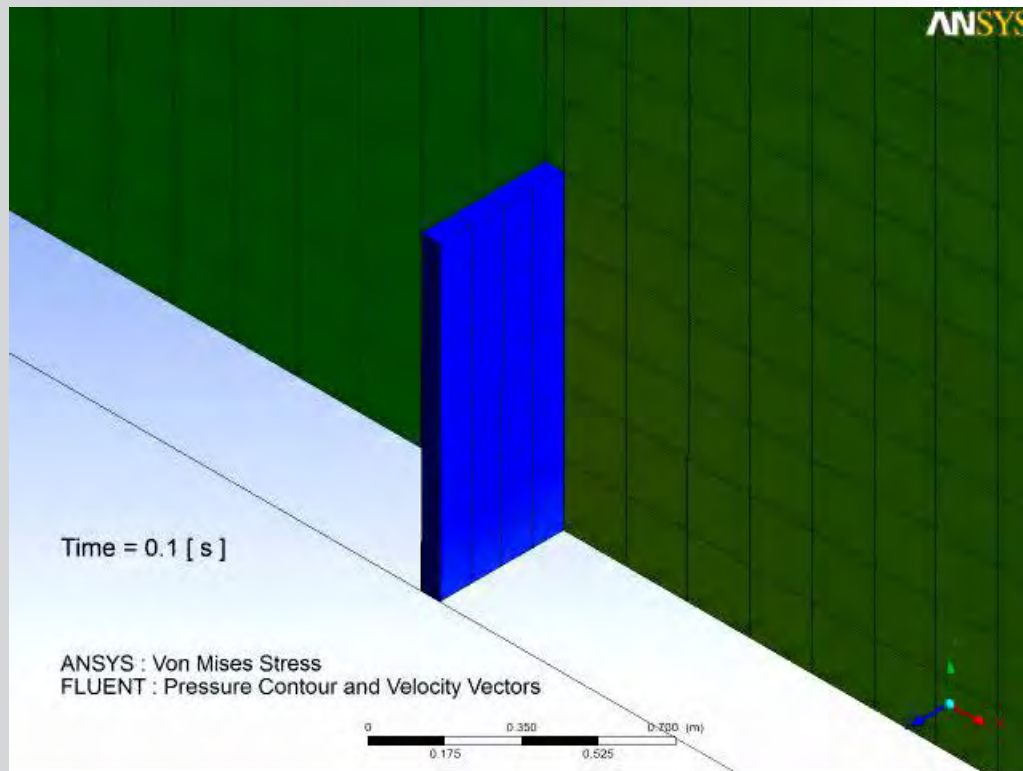
- Tutorial input file available on ANSYS Customer Portal by using the Download Wizard to download **ANSYS_Fluid_Dynamics_Tutorial_Inputs.zip**



- **User documentation contains a dedicated section on Best Practice Guidelines for Using System Coupling**

System Coupling: Oscillating Plate

- Oscillating Plate Verification
 - Excellent correlation between System Coupling, published data and MFX solver



Summary

- **ANSYS provides rich tool set for real world multiphysic solutions integrated in Workbench environment**
- **Streamlined FSI workflow in Workbench**
- **1-way FSI solutions**
 - **CFX, Fluent, Icepak, External data to Mechanical**
 - **Temperature, HTC, surface loads, heat flux, volumetric heat generation, displacement**
- **2-way FSI solutions at R14.0**
 - **CFX-Mechanical coupling through MFX technology**
 - **Thermal, forces and displacement**
 - **Fluent-Mechanical coupling through System Coupling**
 - **Forces and displacement**