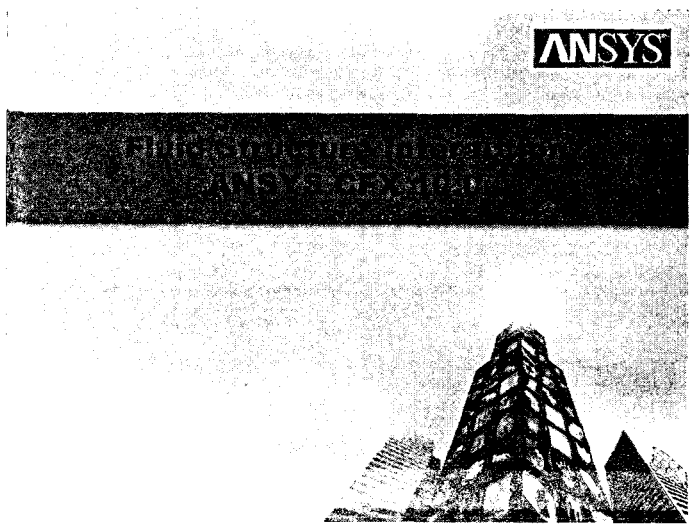


PART I

ANSYS

- The Multi-field solver

 2

Introduction

ANSYS

At release 10.0 The ANSYS Multi-field solver has been enhanced to support coupling with CFX for Fluid Structure Interaction (FSI) applications.

This new advanced FSI capability uses the ANSYS Inc. Multi-field solver™ to provide true bi-directional FSI capability for time transient or steady state FSI analysis with moving and/ deforming geometry. This capability has been implemented for ANSYS Mechanical & Multiphysics coupling to CFX-FCS.

Advanced FSI is required for many industry applications such as:

- Biomedical - elastic artery modeling for stent design,
- Aerofoil flutter
- Civil engineering – wind loading of structures.

 3

Type of Physics Coupling

ANSYS

There exist two fundamental methods to couple physics, regardless of physics or product:

- **Direct**
  - Solves coupled physics equations directly.
  - Solves each physics simultaneously
- **Iterative**
  - Physics are coupled by passing loads across physics field interfaces
  - Solves each physics simultaneously or sequentially
  - At least two iterations, one for each physics, in sequence, are needed to achieve a coupled response.

 4

The Multi-field solver

There are now two flavors of the ANSYS Multi-field solver.

•**MFS - single code**: The basic/core Multi-field solver used if the simulation involves small models with all physics field contained within a single executable (e.g. Multiphysics).

•**MFX - multiple code:** The enhanced Multi-field solver used for simulations with physics fields distributed between more than one executable. At release 10.0 the MFX solver can couple ANSYS Multiphysics and CFX-FCS, or ANSYS Mechanical and CFX-FCS. The MFX solver can accommodate much larger models than the MFS version.

MFS-MFX Differences

•**MFS - single code**: *Uses iterative coupling where each physics is solved sequentially, and each matrix equation solved separately. Solver iterates between each physics field until loads transferred across physics interfaces converge.*

•**MFX - multiple code:** *Uses iterative coupling where each physics is solved either simultaneously or sequentially, and each matrix equation solved separately. Solver iterates between each physics field until loads transferred across physics interfaces converge.*

FSI Terminology / Definitions

We have two levels of CFD coupling capability available:

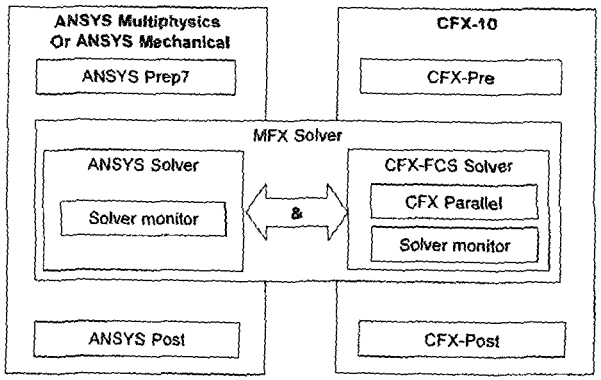
- **Basic FSI:**
  - Implemented for any "traditional" ANSYS and CFX product configuration today – No licensing control.
  - File based unidirectional coupling.
  - *Potentially available in ANSYS Workbench 10.0 for Professional and higher.*
- **Advanced FSI:**
  - Implemented for Multiphysics.
  - Includes basic capability plus:
    - Multi-field external field import (MFIMPORT - unidirectional)
    - Multi-field MFX - multiple code solver
    - Legacy FLOTRAN based FSI (MF – single code solver)
  - *MFX also implemented for ANSYS Mechanical*

Advanced FSI Use Cases

Multiple task, multiple user environment (10.0):

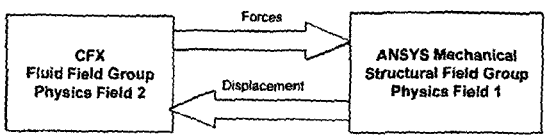
- Implemented for Mechanical & Multiphysics (Release 10.0)
- Advanced FSI use with CFX-FCS
- Separate licensed tasks
- Pre & post processing are simultaneous use

Advanced FSI at Release 10.0



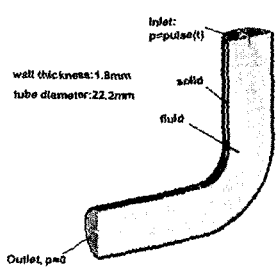
Demonstration Example

- Blood flow in an elastic artery
- MFX solver used
- Transfer of:
  - —Forces from CFX to ANSYS Mechanical
  - —Displacements from ANSYS Mechanical to CFX.

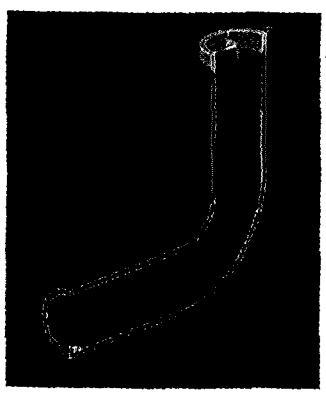


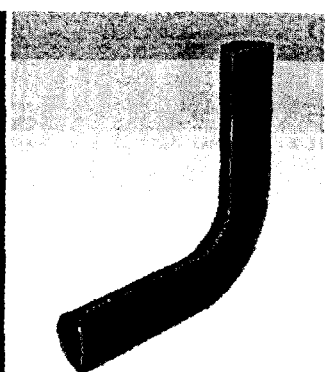
Problem Definition

- 90 deg 'elastic' elbow.
- Pulsatile flow:
  - Transient pressure pulse at inlet.
  - Zero pressure outlet.
- Resulting transient behavior of fluid and blood vessel motion investigated.
- Blood modelled as weakly compressible liquid.
- Blood vessel modelled as elastic wall.
- Material properties:
  - Solid: blood vessel wall
    E = 4.5E6 [N m^-2]
    ρ = 1150 [kg m^-3]
    ν = 0.3
  - Fluid: blood
    ρ = 1050 [kg m^-3] at 1 atm
    μ = 5.0E-4 [m^2 s^1]
    Bulk Comp. = 36.0 [m^2 s^-2]



Results





PART II.

- Basic FSI (Unidirectional)
- Advanced FSI (Bidirectional)

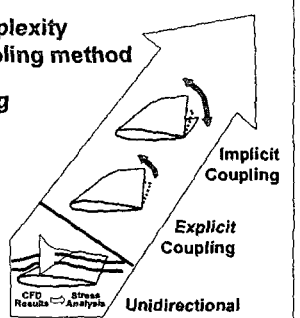
Presentation Outline

- Coupling Strategies
- Basic FSI (Unidirectional)
  - Direct import into WB Simulation
- Advanced FSI (Bidirectional)
  - Multi-field MFX-ANSYS/CFX solver
  - The coupling technology
  - Simulation setup
  - Simulation execution & restarts
  - When things go wrong...

Coupling Strategies

Physical model and coupling complexity determines most appropriate coupling method

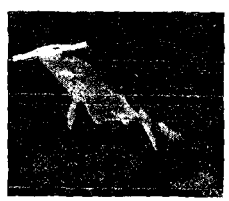
- ***Unidirectional physical coupling***
  - Unidirectional Coupling
- **Steady bidirectional physical coupling**
  - Explicit Coupling
- **Moderate to strong transient bidirectional physical coupling**
  - Implicit Coupling (Direct & Segregated)

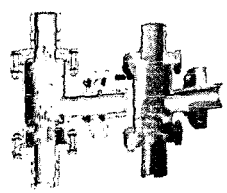


Basic Fluid Structure Interaction

Predominantly unidirectional interaction between fields in fluid and solid domains

- **CFX: Pure moving mesh applications**
- **CFX & ANSYS: Mechanical or heat transfer**



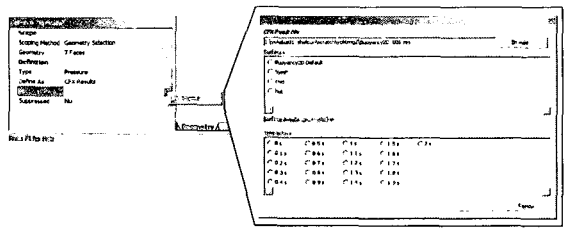


Basic FSI: CFX & ANSYS
Exporting Results from CFX

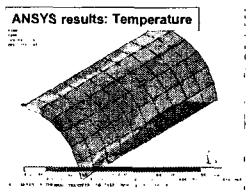
- Surface export via CFX-Post
  - Import ANSYS surface mesh
  - Interpolate CFX results to ANSYS mesh and export
  - Stress, temperature, heat flux & transfer coef.
- Surface & volume export via Solver Manager
  - Export results on CFX mesh as ANSYS cdb file
  - Read data into ANSYS MFS Multi-field solver
  - Force, temperature, heat flow rate

Basic FSI: CFX & ANSYS
Direct Import CFX ⇨ Simulation

- **When applying Pressure load in WB Simulation, specify "CFX Results" in the Details view**
  - Under "CFX Surface," the CFX results file can be specified
  - Select the surface and time values (for transient analyses) to import results from



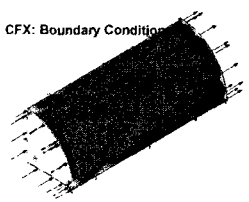
Basic FSI: CFX & ANSYS
Importing Results from ANSYS

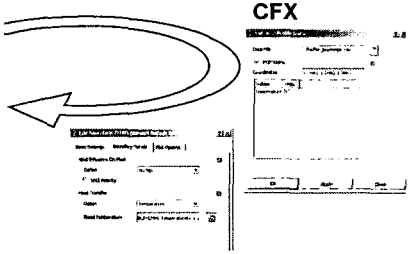


ANSYS

- Identify surface or volume interfaces
- *Export data: displacement, temperature,* heat flux, momentum and energy sources

CFX

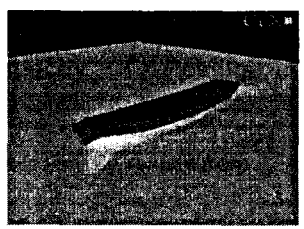


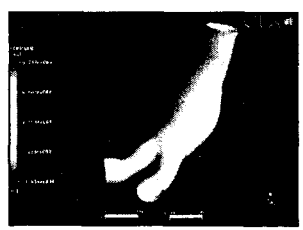


Advanced Fluid Structure Interaction

Predominantly bidirectional interaction between fields in fluid and solid domains

- **CFX: CEL or User Fortan solid dynamics**
- **CFX & ANSYS: Mechanical or heat transfer**



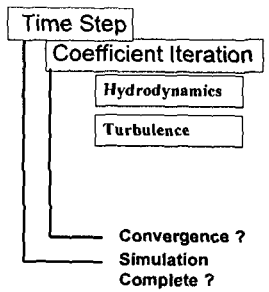


Advanced FSI: CFX & ANSYS
The MFX Multi-field Solver

Automated framework for coupling data from different field solvers (e.g. ANSYS and CFX)

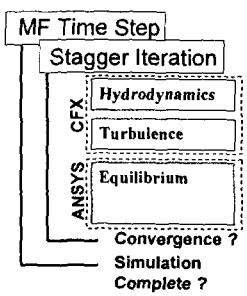
- MFX is the multiple-code extension of the original ANSYS MFS (single code) Multi-field solver
- Field solvers are treated as segregated systems, coupled via 'lagged' influences (i.e. equations' RHS)
  - Mechanical and thermal data on boundaries (other data and volume transfer in future...)
- ANSYS Multi-field solver couples segregated field equations the same way that CFX does...

Advanced FSI: CFX & ANSYS
A CFX View of the ANSYS Multi-field



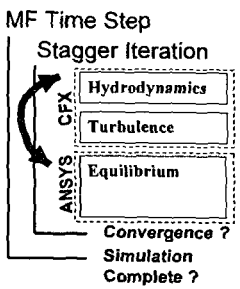
- Transport equations advanced through simulation duration together
- Coefficient iterations ensure convergence of
  - equation non-linearities
  - segregated equation coupling
- Fully implicit solution possible during each time step

Advanced FSI: CFX & ANSYS
The MFX Multi-field Solver



- Field solvers advanced through simulation duration together
- Stagger iterations ensure convergence of
  - Equations per field solver
  - Coupling data
- **Fully implicit solution for each MF time step**

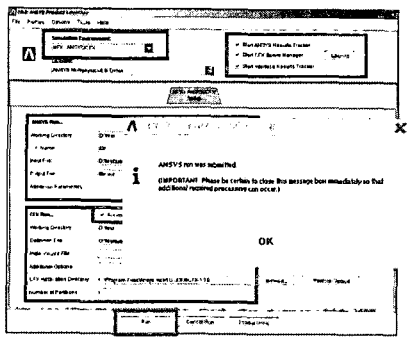
Advanced FSI: CFX & ANSYS
Field Solver Coupling Technology



- **Data transfer across standard TCPIP internet sockets**
  - Efficient, proven technology
  - Heterogeneous mix of archs. over LAN, WAN, internet
- **Our own Client-Server based communication API**
  - Each field solver gathers data from other solvers, and then uses the data

No Third Party Software

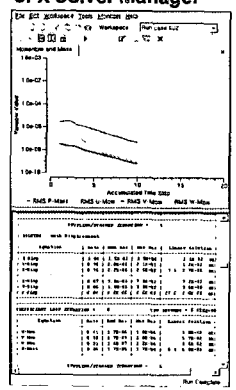
Simulation Execution
The ANSYS Launcher



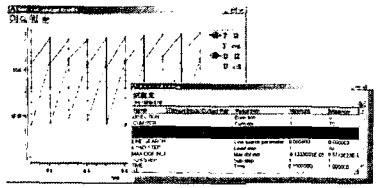
- Set Sim. Environment
- Setup ANSYS Run
- Setup CFX Run
  - Auto-start serial and local parallel runs
  - Manual start remote and distributed parallel runs
- Run the simulation
- Launch
  - CFX Solver Manager
  - ANSYS Trackers

Simulation Execution
Monitoring Simulation Convergence

CFX Solver Manager



ANSYS Tracker

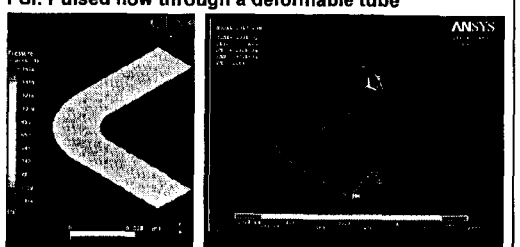


- Convergence of data transferred is reported in ANSYS `.out` file (Negative values = converged)
- **TIP:** Add a text monitor of ANSYS output in the CFX Solver Manager

Simulation Post Processing

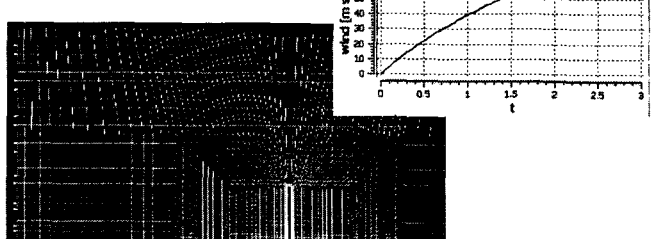
- Must post process ANSYS & CFX results separately
- CFX shows true displacements whereas ANSYS displacements can be amplified

FSI: Pulsed flow through a deformable tube



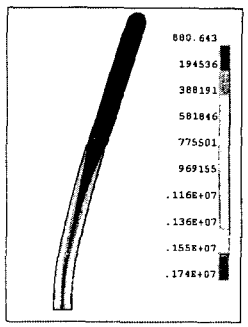
Example

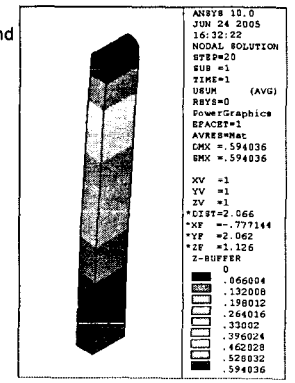
- Consider a wall that is 4 meters high, and a wind arises 20 meters upstream of the wall *with a velocity vs. time profile as shown...*
- The wall, modeled in ANSYS, is a quarter meter thick with a modulus of elasticity of 1.0E+8 (nt/m2).
- How will the wall deform with time?



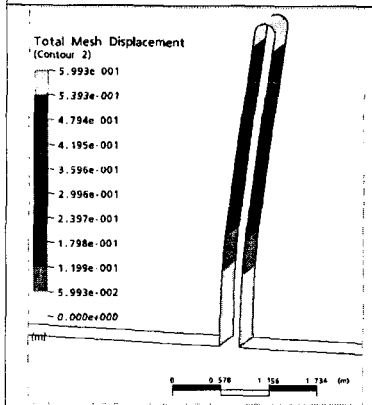
ANSYS Solution – Static Result

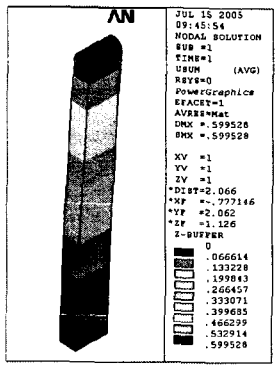
- Right: Displacement
- Below: Von Mises Stress, 1.0 second
- True Scale



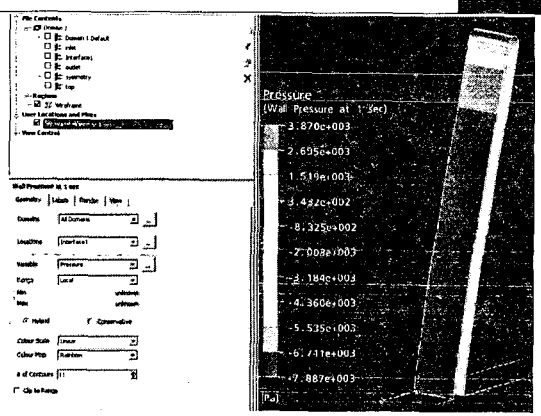


CFX and ANSYS Displacement

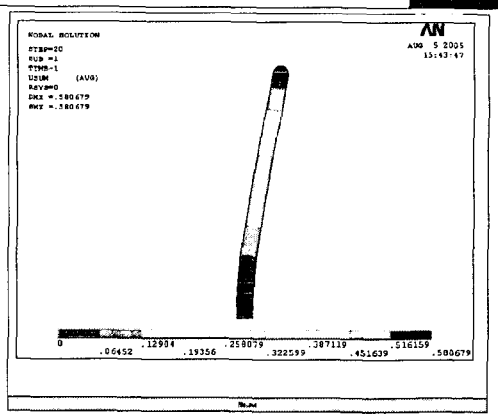




Fluid Pressures at 1.0 second



Wall Movement

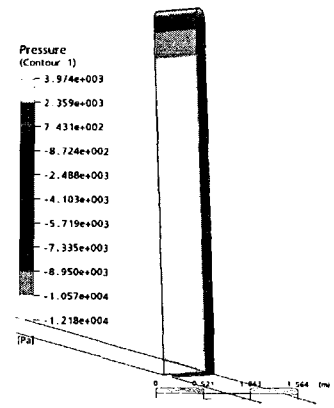


Flow Only Result at 1.0

ANSYS

Peak Pressure 3974 without FSI

Peak Pressure 3870 with FSI



33