ANSYS CFD v17
Update Seminar
Outline

- ANSYS CFD Demo
  - System coupling: CFX and Mechanical
  - Live CFX Solution Monitoring

- ANSYS CFX v17

- ANSYS Fluent v17

- ANSYS Icepak v17
CFX System Coupling demo

Fixed support

Flow outlet

FSI interface

Fixed support

Flow inlet

www.caeai.com

ENGINEERING ADVANTAGE
Outline

- ANSYS CFD Demo
  - System coupling: CFX and Mechanical
  - Live CFX Solution Monitoring
- ANSYS CFX v17
- ANSYS Fluent v17
- ANSYS Icepak v17
FSI with CFX and Systems Coupling

- 2-Way FSI between CFX and Mechanical
  - Supersede MFX with workflow and technology consistent with Fluent-Mechanical coupling
  - Initial support for steady and transient exchange of forces and resultant displacements
- Continuously driving down simulation times!
- Special focus on Transient Rotor-Stator
- Multiple areas improved
  - Assembly/discretization
  - Linear solver
  - GGI intersection
  - …

Overall impact of efficiency improvements over multiple releases, shown on example transient water turbine simulation with 40M nodes (benchmark case in collaboration with Voith Hydro and HLRS in Stuttgart)
Improved Solver I/O Efficiency

• Drastically reduced solver I/O for large/complex cases with very large number of regions/face sets (10s of thousands) when running parallel
  - Previously could become large enough to impact overall solution time
  - Activated automatically
    • No additional setting required

Reduction in wall clock seconds for I/O on an example test case with many regions
Advances in Turbulence Modelling

- New Shielded Detached Eddy Simulation (SDES) and Stress-Blended Eddy Simulation (SBES)
  - Faster switch to LES modes
  - Combine RANS and LES with blending
    → Blended eddy viscosity

- Intermittency Transition Model
  - Full release in CFX

- All turbulence model enhancements also released in Fluent
Significantly improved HPC scalability in speed of Monte Carlo solver

Dramatically better solver speed scalability for simulations with the Monte Carlo radiation model

Complex headlamp test case with 10E6 histories
Comparison when solving only radiation and energy
BladeModeler: Blade Row Geometry

- New Blade Neutral Data Format (NDF) file
  - Support in BladeGen and BladeEditor
    - Provides connection between BladeEditor and BladeGen
    - BladeEditor CADImport feature directly creates NDF file for loading into BladeGen
  - Future:
    - Support broader BladeGen to BladeEditor workflow and parameters
    - Provide improved connection to third party blade design tools
Turbo Blade Row Modeling

- Extended models and improved usability
  - Time Transformation and Fourier Transformation
    - Multi-stage, Asymmetric flow, Multi-disturbance
  - Transient Blade Row with CHT
  - Mixing plane improvements
  - Convergence monitoring for cyclically periodic flow ($\beta$)
  - Harmonic Analysis: frequency-domain method ($\beta$)
Enhanced CFX Multiphase Robustness

- Incorporation of numerous enhancements for challenging multi-phase applications
  - Volume weight averaging for turbulent dispersion coefficient (\(\beta\))
    - Better robustness of bubble boundary layers on fine meshes
  - New root finder for wall superheating (\(\beta\))
    - Faster and more generic root finder for wall boiling applications
  - Hosokawa model for wall lubrication force (\(\beta\))
    - Includes phase relative Reynolds number in liquid-gas bubble flows
  - No clipping of interfacial area density for interphase mass transfer (\(\beta\))
    - Greater accuracy where volume fractions \(\rightarrow 0\)
User Surface Results Output for CFD-Post

- Results output to arbitrary surface locations
  - Generate significantly less data
  - Leaner simulation and analysis

- Significant impact for large transient cases
  - Data (disk space) may be reduced by orders of magnitude
  - Less data sent between CFD-Post engine and GUI during post-processing

- Analogous capability for Fluent and CFX
CFD-Post Time Animation

- Ease-of-use for transient post-processing
  - ‘Music player’-like control
    - Play/stop
    - Next/previous time step
    - First/last time step

- Options to
  - Specify desired transient range by time step, time, or crank angle
  - Control which frame to use when between available frames
  - Save time animation to movie file
Outline

- ANSYS CFD Demo
  - System coupling : CFX and Mechanical
  - Live CFX Solution Monitoring

- ANSYS CFX v17

- ANSYS Fluent v17

- ANSYS Icepak v17
Fluent User Interface

**Ribbon:**
Access primary workflow

**Tree:**
Access model particulars and shortcuts
Overset mesh problems can be set up and solved for:

- Steady and transient (fixed mesh), 3D and 2D planar
- Pressure Based coupled solver
- Incompressible density method
- Single-phase or VOF multiphase
- Heat transfer
- $k$-epsilon and SST $k$-$\omega$ turbulence models
- BETA: moving mesh, compressible flow, VOF with surface tension, pressure-far-field BC, Workbench support, Pressure-Based segregated algorithms

Limitations:
- DBNS not supported
- Node-based gradients not supported
**Fluent Scalability**

- Removed bottlenecks at case reading and parallel build at high core count runs
- Communication optimizations
- Benefits for large and small cases

[Graphs and data showing scalability improvements with increasing core counts.]
Additional benefits from combustion code refactoring for cases with more than 1 cell zone with partially-premixed or PDF models

Ice_2m: 7 cell zones, 1.6 million cells
MDM, Spray, Partially premixed
Fluent Convergence Improvements

• Conservative Coarsening is now used by default with the coupled solver
  – Typically improves convergence
  – Especially helpful for cases with native polyhedral meshes and/or highly stretched cells.
  – Can be disabled on the Multigrid tab of the Advanced Solution Controls dialog box

• Algebraic multigrid solver now automatically reorders the linear system
  – Necessary to ensure proper ordering for multiple cell zones
  – Can be disabled with (rpsetvar 'reorder/amg 0)
Fluent Boundary Conditions

• Mass Flow Rate, Mass Flux, and Average Mass Flux are now defined according to the selected Reference Frame option in the boundary condition dialog box.

• Previously, these were always relative to the adjacent cell zone.
• Backflow quantities at Pressure Outlets can now be defined with respect to either the absolute reference frame or the adjacent cell zone.

• Non-Reflecting Boundary Conditions can now be used with compressible-liquid materials (beta)
You can now specify a transient profile for wall translation or rotation in the Wall Zone dialog box.
Adjoint Solver & Design Tool

- Compressible Flow support in adjoint solver
- Design tool Enhancements
  - General bounding surfaces for deformations
  - Prescribed motions (rotation, translation, scaling)
  - Improved usability - constraint checking
  - STL Export

Total pressure drop through a duct is minimized by modifying the duct geometry.

The duct is required to remain within a complex bounding surface, defined by an imported mesh, while the inlet and outlet are fixed.
Fluent Monte Carlo Radiation Model (beta)

- Re-application of CFX model
- Much better accuracy for collimated radiation compared with other methods
- Especially useful for headlamp simulations, etc.

Simple Lens Case using DO Model

Simple Lens Case using Monte Carlo Model

Limitations in R17.0
- Beta feature
- Serial-only
- 3D only
Fluent DPM/DEM Enhancements

- Rotation of DPM particles is now a full feature
  - Gliding friction in particle-wall collisions and lift forces (Magnus lift) can be included
  - Applicable for particulate flows in cyclones, etc.
  - Not compatible with MRF. Rotation not considered in O’Rourke collision model

- Additional DEM collision models
  - Rolling friction: applicable to bulk solid flows (e.g. fluidized beds and rotating drums)
    - Hertzian
    - Hertzian-dashpot

- Macroscopic Particle Model UDF is now included with Fluent as an Add-On model

- New Rough Wall model gives better predictions in confined geometries
Wall Film Enhancements

Dry-splash regime

Film-formation regime

- Eulerian wall-film model now supports variable density of the film material.
Outline

- ANSYS CFD Demo
  - System coupling: CFX and Mechanical
  - Live CFX Solution Monitoring
- ANSYS CFX v17
- ANSYS Fluent v17
- ANSYS Icepak v17
Icepak v17.0

- IDX ECAD File Format now supported
  - Similar functionality to IDF file format
  - Contains component location, thermal properties, and component grouping

- Trace and via conductivity mapping improvements
  - Improved algorithm to compute the orthotropic thermal conductivity map
Icepak v17.0

- HPC scalability improvements
  - Enhanced non-conformal interface handling results in up to 50x speed up
  - Fluid/solid load balancing enhancements

- Full support for Remote Solve Manager
  - Offload Icepak jobs to remote solve machines
  - WB integration means Icepak design points can be solved through RSM
Icepak v17.0

- 2D and 3D plotting of power maps
  - Heatflux, Power and Temperature variables supported
  - Results can be viewed pre- and post-solution

- Export geometry to ANSYS SpaceClaim
  - Icepak geometry can be exported in *.obj format