ANSYS Fluid Dynamics Update

Dr Mike Marchant
ANSYS Sub-Program Director - Fluids
ANSYS Fluid Dynamics

ANSYS Fluid Dynamics Products
• ANSYS Fluent
• ANSYS CFX
• ANSYS CFD-Post
• ANSYS TurboGrid
• ANSYS Polyflow

ANSYS Fluids Dynamics at 14.0
• Significant benefits for users
  • Including geometry and meshing
  • Look for detailed materials at release time
Our Fluids Strategy

We remain committed to developing and releasing an integrated, unified fluids product
• That will draw upon the best of both Fluent and CFX, as well as add new functionality
• That will minimize migration hurdles for our fluids customers

The first release of this product is targeted for ANSYS R15.0

Fluent and CFX will continue to be developed in the meantime (and actively maintained for a few releases beyond R15.0)
• To ensure smooth migration for our fluids customers
Fluid Dynamics Themes

- Rapid & Robust Meshing
- Workflow & Usability
- Multiphysics and Systems Coupling
- Solver and HPC Performance
- Rotating Machinery
- Automotive Power Train Modeling
- Multiphase Flow Modeling
- Comprehensive CFD Capabilities
- Special Material Processing
- Summary
Rapid & Robust Meshing

Top on the list of challenges engineering companies are facing is shortened product development schedules while at the same time the product designs themselves are becoming increasingly complex.

Meshing of these designs introduce challenges in terms of speed, robustness and accuracy.

Courtesy Siemens AG.
Rapid & Robust Meshing

Enhanced productivity through increased automation, flexibility, efficiency and robustness

- Assembly Meshing (Tet and CutCell)
- Performance (Speed, Robustness)
- Selective Meshing
- Virtual Topologies
- Hex Meshing
- ICEM CFD/TGrid
Simulation departments are looking for improved usability and the ability to glean more information from fluid dynamics simulations for all users – from occasional designers to experienced analysts – from geometry creation through post-processing.
Focus on enhancing your productivity through new features, increased flexibility, efficiency and usability

• ANSYS DesignModeler
  – Core modeling improvements
  – Application-specific modeling

• ANSYS SpaceClaim Direct Modeler
  – Improved Workbench integration
  – Enhanced Model Preparation

• Interoperability
  – Support for new CAD releases
  – New CAD file readers
Workflow & Usability – ANSYS DX

• Workbench Design Exploration and Optimization for increased understanding, innovation & simulation ROI

• New at 14.0
  • Reduced time required
    • 2 new adaptive DOE’s
    • Distributed solve
    • Design point sorting
  • Increased Robustness
    • Reserved licensing$\beta$
    • Support for partial DOE’s
  • Increased understanding
    • New charts
    • Improved GOF
    • Project report

R14

Workflow & Usability – ANSYS DX

• Workbench Design Exploration and Optimization for increased understanding, innovation & simulation ROI

• New at 14.0
  • Reduced time required
    • 2 new adaptive DOE’s
    • Distributed solve
    • Design point sorting
  • Increased Robustness
    • Reserved licensing$\beta$
    • Support for partial DOE’s
  • Increased understanding
    • New charts
    • Improved GOF
    • Project report

R14

Workflow & Usability – ANSYS DX

• Workbench Design Exploration and Optimization for increased understanding, innovation & simulation ROI

• New at 14.0
  • Reduced time required
    • 2 new adaptive DOE’s
    • Distributed solve
    • Design point sorting
  • Increased Robustness
    • Reserved licensing$\beta$
    • Support for partial DOE’s
  • Increased understanding
    • New charts
    • Improved GOF
    • Project report

R14
ANSYS Remote Solver Manager at R14 (FL and CFX)

- Major improvements for fluids as well as general usability and robustness improvements to RSM
- Queue multiple jobs on a local machine
  - Overnight or other low-usage times
- Submit jobs to remote machines
  - Distributed clusters
  - Management of files, including UDFs
- Update design points in parallel via RSM
Improved workflow and usability in ANSYS Workbench

• Extended User Preferences (FL)
  – General options
  – Launcher options

• Extended parameters for Fluent
  – Real and profiles variables in zone and domain settings
  – Examples:
    • Phases: Nucleation Rate, Coalescence and Breakage Kernels
    • Phase Interaction: Surface Tension Coefficients, Lift Coefficient, Restitution Coefficient
    • Contact Angles (on wall BC)
Workflow & Usability – ANSYS CFD

Process more information per simulation (FL)

• Monitor more than a single lift, drag, or moment monitor
  – Multiple monitors for each type
• Assign any variable to a custom field function for use with unsteady statistics

Export data on per cell zone basis for faster file write and reduced storage needs (FL)

• CFD Post, EnSight, and FieldView

Multiple force monitors

Custom field functions for statistics
Optimization of transient chart creation performance (CFD-Post)

- Update only relevant objects during transient chart creation
  - Significant speed-up when current state contains many un-related objects

Additional ability to reduce transient data (CFX)

- Output selected solution data only at boundary
  - E.g. pressure at surfaces
- Large data reduction possible

<table>
<thead>
<tr>
<th>Transient chart creation speed-up*</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>FFT Chart</td>
<td>48 [%]</td>
</tr>
<tr>
<td>Time Chart</td>
<td>54 [%]</td>
</tr>
</tbody>
</table>

* On standard test case; will vary depending on particular case and post-processing state
Workflow & Usability – ANSYS CFD

Enhanced ability to use ANSYS CFD with other tools
• CGNS library version upgrade to 3.0
• Support face-based regions in CFD-Post (in addition to node-based)
  – Enable adoption use as common CFD post-processor for 3rd party tools

Control of default launcher working directory (CFX) β
• Ability to switch default to directory in which launcher was started

Improved licensing preferences (CFX)
• Adherence to global license settings
Easier specification of animation resolution for high definition playback (CFD-Post)

- Extended output options to include HD standard sizes
  - HD Video 720p
  - HD Video 1080p
- Also: Automatic internal adjustment to avoid Windows Media Player playback limitations
Multiphysics and Systems Coupling

Engineers need to accurately predict how complex products behave in real-world environments with real world physics.

ANSYS is continuously improving the ease of use and efficiency of simulating real world interactions between fluid dynamics, structural mechanics, heat transfer, and electrodynamics within a single, unified engineering simulation environment using systems coupling.
Two-way surface force/displacement coupling between Fluent and Mechanical via Systems Coupling

- Steady/static and transient two-way FSI
- Integrated post-processing with CFD-Post
- Workbench based setup and execution
  - Windows and Linux
- Alternative execution from command line
  - including cross-platform
- Parallel processing with ANSYS HPC
  - RSM currently not supported
- Restarts for fluid-structure interaction
- Parameterization, design exploration and optimization

Non-Newtonian blood flow through a three leaf mitral valve
Multiphysics – 1-way FSI

Significantly faster surface mapping for 1-way FSI (CFD-Post) $^\beta$

- New Octree mapping method $\rightarrow$ significantly faster algorithm
  - Need to set Option in CFD-Post
- 1-way FSI in ANSYS Workbench uses CFD-Post ‘under-the-hood’
  - Will use mapping option set by user in CFD-Post (which is stored in user preferences)
  - Status message with diagnostics report indicates new mapping method is being used
Multiphysics – Fluent-Maxwell

Complex multiphysics modeling
• New: Electromagnetic-thermal interactions inside Workbench using Fluent with Maxwell
  – One-way and two-way $\beta$ coupling
• Combine with 1-way FSI
Companies need to make informed development decisions about their increasingly complex products in increasingly shorter time frames.
Adjoint Solver for Fluent fully tested, documented, and supported at R14.0

- Provides information about a fluid system that is very difficult and expensive to gather otherwise
- Computes the derivative of an engineering quantity with respect to inputs for the system
- Engineering quantities available
  - Down-force, Drag, Pressure drop
- Robust for large meshes
  - Tested up to ~15M cell

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Predicted</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>---</td>
<td>555.26</td>
</tr>
<tr>
<td>Mod. 1</td>
<td>577.7</td>
<td>578.3</td>
</tr>
<tr>
<td>Mod. 2</td>
<td>600.7</td>
<td>599.7</td>
</tr>
<tr>
<td>Mod. 3</td>
<td>622.0</td>
<td>621.8</td>
</tr>
</tbody>
</table>
Solver and HPC Performance

Mesh Morpher and Optimizer

**Fluent Mesh Morpher and Optimizer**

- Create your own optimization functions using parameters
- Faster mesh deformation
- Constrain some boundaries and allow others to deform
- More easily assess the effectiveness of the optimization routine
  - Save and plot the value of the objective function as a function of design iteration number
- Execute TUI commands before and after the optimization loop
Solver and HPC Performance

Focus on robustness, accuracy, and efficiency

• Improved performance (FL)
  – Higher order term relaxation (HOTR)
  • More aggressive initial settings
  • Improved convergence
  – Out of the box performance
  • New defaults
    – 2\textsuperscript{nd} order for some equations
    – hybrid initialization
  • Solver optimizations
    – Migration manual
• Beta: NRBCs with PBNS (FL)
  – Compatible with combustion models

\begin{center}
\begin{tabular}{c c}
\textbf{RAE-2822 Airfoil} & \\
\textbf{M=0.73} & \textbf{AOA=2.8 deg} \\
\end{tabular}
\end{center}

\begin{center}
\begin{tabular}{c}
\textbf{Standard @ CFL=200} \\
\end{tabular}
\end{center}

\begin{center}
\begin{tabular}{c}
\textbf{HOTR @ CFL=200} \\
\end{tabular}
\end{center}

\begin{center}
\begin{tabular}{c}
\textbf{Standard @ CFL=100} \\
\end{tabular}
\end{center}

\begin{center}
\begin{tabular}{c}
\textbf{HOTR @ CFL=100} \\
\end{tabular}
\end{center}

\begin{center}
\textbf{RAE-2822 Airfoil}
\end{center}

\begin{center}
\textbf{M=0.73}
\end{center}

\begin{center}
\textbf{AOA=2.8 deg}
\end{center}

\begin{center}
\textbf{Normalized scales residuals comparing HOTR to standard relaxation at different CFL settings}
\end{center}
Faster convergence on stretched meshes

- Convergence Acceleration for Stretched Meshes (CASM) with the density-based implicit solver

- Recommended for:
  - Steady-state simulations
  - Anisotropic meshes with high stretching in the local flow direction
  - Stretched meshes with $Y+$ near 1

Convergence acceleration for stretched meshes requires nearly 10x fewer iterations in this case (300 vs. 3000 iterations)
Solver and HPC Performance

Improved scalability (FL)
- Scalability to higher core counts
- Simulations with monitors including plotting and printing
- Cluster-to-cluster view factor file writing optimization

Example data for scaling with R14 monitors

Sample cluster-to-cluster view factor data writing using 32-way parallel and Infiniband.
Solver and HPC Performance

Faster auto-partitioning (FL)
- Optimized for multi-core clusters
- All simulations benefit
- New default
- More constant time required with fixed overhead

Improved usability (FL)
- Better error tracking
- Latest Platform and Intel MPI versions on all platforms

Work in Progress: GPU investigation
- R14.0: Viewfactor and ray tracing calculations on GPUs (FL)β

Example viewfactor calculation times for different combinations of GPUs and CPUs

<table>
<thead>
<tr>
<th>Time (seconds)</th>
<th>192</th>
<th>384</th>
<th>768</th>
<th>1536</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time</td>
<td>5.307</td>
<td>4.542</td>
<td>6.177</td>
<td>8.109</td>
</tr>
</tbody>
</table>
Solver and HPC Performance

Optimize parallel partitioning in multi-core clusters (CFX)\(\beta\)

- Partitioner determines number of connections between partitions and optimizes part.-host assignments

Re-use previous results to initialize calculations on large problem (CFX) \(\beta\)

- Large case interpolation for cases with >~100M nodes

Clean up of coupled partitioning option for multi-domain cases (CFX)

- Eliminates ‘isolated’ partition spots
Rotating machinery plays a key role in many industries, including aerospace, power generation, automotive, marine, HVAC and healthcare. Manufacturers are currently challenged to improve the performance of their machines, more than ever before.
Rotating Machinery

Highly efficient time accurate simulations with Transient Blade Row capability (CFX)
• Several models available
  – Time Transformation (TT)
    • Inlet Disturbance
    • Single Stage TRS
  – Fourier Transformation (FT)
    • Inlet Disturbance
    • Single Stage TRS $\beta$
    • Blade Flutter $\beta$

Surface pressure distribution (top) and monitor point pressure (left) from an axial fan stage:
Equivalent solution with Time Transformation at fraction of computational effort

Reference solution without a TBR method, requiring 180 deg model
Time Transformation solution, requiring only 3 stator and 2 fan blades
Rotating Machinery

Time Transformation

- Solve in transformed ‘inclined’ time, fully implicit, with turbulence
- Results in inclined time stored using Fourier-based data compression
- Results re-constructed in physical time for analysis

Fourier Transformation

- Fourier–series used for reconstruction of solution history on pitch-wise boundary and inter-row interfaces for efficient data storage & convergence
- Two passages for better signal collection and faster convergence than single passage
Workflow extensions for Transient Blade Row (CFX)

• Incorporation of TBR methods in Turbo setup mode
  – Switch to general mode for case-specific details
• Profile replication and clocking for inlet disturbances
  – Automatically ensure complete circumferential overlap of profile with mesh
  – Select profile and specify rotation rate
Integrated Transient Blade Row analysis (CFD-Post)

- Full integration of single passage TBR analysis
  - Each plots can be at selected time or phase position in period being modeled
  - Expansion of results (to show multiple passage simultaneously) possible with macro provided with installation
Rotating Machinery

Additional integrated turbo analysis capabilities (CFD-Post)
• Reduce need to export data and manipulate externally
• Directly assess stream-wise changes in span-wise distributions of circumferential averages
  – Look at differences or ratios of existing variables, e.g. pressure ratio
• Further derived data possible with ability to define separate lines

Easier set-up of monitor points for rotating machinery (CFX) $\beta$
• Specification in cylindrical coordinates
Rotating Machinery

Highly automated blade row meshing – without sacrificing quality (TurboGrid)

- ‘ATM’ method expanded to handle (single) splitter blades
- Additional enhanced templates available
Automotive Power Train

With government pressure and market demand for the better fuel efficiency and reduced environmental impact of vehicles, there is a strong need to improve simulation technology for applications such as hybrid/electric powertrains, turbo chargers, direct injection, etc..
New for R14: IC Engine Analysis System provides unified and simplified set-up for internal combustion engines

- Controls simulations based on user inputs
  - Geometry decomposition
  - Meshing
  - Dynamic mesh set-up
  - Reporting
- Parametric updates
- Pervasive use of crank angle
- Option to include journal files
  - Automate combustion and spray set-up
Automotive Power Train – IC Engine System
Automotive after-treatment (FL)

- Selective Catalytic Reduction (SCR)
  - Urea droplets and particles material
  - Improved DPM droplet vaporization and boiling
  - Uniformity index for post-processing

- Catalytic Converter Light-Off
  - Non-equilibrium thermal model for porous media predicts fluid and solid temperatures separately
  - “Dual cell approach” automatically creates connected mesh for solid zone
  - Include surface chemistry
Automotive Power Train – Solver meshing

Fluent MDM improvements driven by internal combustion engine needs

- Retain and remesh boundary layers during tetrahedral remeshing (FL)
  - Boundary layer settings from original mesh
  - Example applications: internal combustion engines and FSI

- Improved robustness and usability for dynamic mesh (FL)
  - Mesh smoothing
  - Cut cell remeshing
  - Parallel
Automotive Power Train – Battery Model

Fluent Battery Model: a pragmatic approach to full-scale battery computations

- Temperature effect on battery dynamics
- Appropriate for various types of batteries
- Robust numerical algorithm for fast convergence
- User-customizable properties and correlations

Discharge curves for a prismatic Li-ion cell

Temperature

Current Density

Contours on a prismatic Li-ion cell

2 Ah Prismatic Cell Discharge Curve
Many industrial processes involve the simultaneous flow of multiple phases.

Most of these processes are impossible to observe directly. Therefore, engineers rely on models and experiments to gain insight into improving the efficiency, throughput, safety and reliability of their processes.
Multiphase – DPM

Accuracy and robustness improvements

• Accurate heat and mass transfer for low and high evaporation rates in the same simulation (FL)
  – Convection/_diffusion controlled vaporization

• More accurate modeling for high Weber number sprays (FL)
  – Stochastic Secondary Droplet Breakup Model (SSD)

• Improved tracking for DPM with moving and deforming meshes (FL)
  – Particles are tracked in the cells deformed/moved each time step
Multiphase – DPM

Easier analysis and visualization of particle data (FL)
• Display particles as spheres with varying size
• Display vectors/cylinders on particles to indicate a direction of velocities, forces, etc.
• Filter particle display to view only a small part of particle tracks

Similar improvements for FLUENT particle tracks in CFD-Post
Model dense particulate flows with DEM (FL)
• DEM enabled as a collision model in the DPM model panel
• Use in combination with single phase and DDPM simulations
• Works in parallel
• Particle size distributions
• Prediction of the packing limit
• Head-on collisions
• Collisions with walls
• Example applications: Bubbling and circulating fluidized beds, particle deposition in filtering devices, particle discharge devices (silos)

NETL Fluidized Bed Simulations using DEM with DDPM

3% fines
start-up -15 sec and 15-30 sec

12% fines
0-25 sec

Note that channeling is observed in the 15-30 sec animation
Multiphase – DPM

Easily replicate particle or droplet injection at different locations or in different directions (CFX)

• Specify local coordinate frame for each Particle Injection Regions (PIR)
• Much more convenient for set-up of large numbers of PIRs
  – Applications with multi-port fuel injection, spray dryers, scrubbers, etc.

Directly specify swirling injection at PIRs (CFX)

• Use CEL to flexibly define cylindrical components as f(position, time, ...)
• Includes extension to LISA model for pressure-swirl atomizers
Multiphase – DPM

Improve predictions of droplet behaviour in the superheated regime (CFX)$^\beta$

- Model mass transfer based on degree of superheating (Temperature excess above boiling temperature)
- Differentiate between temperature at droplet interface and temperature within the droplet
  - Interface temperature based on local energy balance of heat transfer into and away from particle
  - Internal heat transfer rate based on empirical correlation from Adachi et al
Multiphase – DPM

Generate additional particle data output for further analysis (CFX) β

- Frequency distributions of particle quantities at boundaries to *.csv for histograms in CFD-Post or elsewhere
- Wall particle monitoring per patch
  - Instantaneous and time-integrated

Extend models with algebraic AVs for particles (CFX) β

- Analogous to fluid AVs

Enhanced FLUENT particle track post-processing (CFD-Post)

- Vector variables and sizing by diameter

Histogram of particle size distribution at an outlet

Time-integrated mass of particles hitting a wall
Multiphase – Condensation

Ability to include global effect of wall condensation without multi-phase details (CFX)

• Single phase, multiple components
  – Mixture of one condensable and one or more non-condensable species

• Condensable component extracted by sink terms at walls and CHT boundaries, as function of concentration through boundary layer
  – Liquid film is not modeled

• Key application: nuclear accident scenarios looking at containment pressure variation over time need to include macroscopic effect of condensation
Multiphase – Free Surfaces

Better robustness and faster convergence for free surface steady-state cases using coupled VOF (FL)

• Improved in R14.0

Options for better trade-off between stability and accuracy (FL)

• Hybrid treatment of Rhie-Chow face flux interpolation with special treatment near free surfaces
Multiphase – Free Surfaces

Continuum surface stress method for surface tension flexibility (FL)

- Surface tension as a function of any variable
- Alternative to existing CSF model in some difficult cases
  - For example: cases with sharp interface discontinuities

Sample data: surface tension and viscosity as a function of temperature

Experiment image compared to Fluent results.
Image from: Thermally Induced Marangoni Instability of Liquid Microjets with Application to Continuous Inkjet Printing by Furlani et al.
Boiling model extensions, and testing (FL)

• Critical heat flux (CHF) for modeling boiling dry out conditions
• Transition smoothly between the bubbly and droplet regimes
• Boiling with bubble size distributions using interfacial area concentration (IAC) models
  • Accurate interfacial areas for heat and mass transfer calculations in non-equilibrium boiling conditions
• Example applications: Nuclear industry, engine jacket cooling
Multiphase – Eulerian

Combine phase change and varying dispersed phase size distributions for improved accuracy (CFX) $\beta$

- Homogeneous and inhomogeneous MuSiG can be applied together with general phase change and RPI wall boiling models

Simulation and comparison with experimental results from DEBORA Test Facility, courtesy of HZDR (formerly FZD)
Multiphase – Population Balance

Model diameter and other variable changes (bin fraction, moments) due to density changes in the dispersed phase (FL)

- Compressible dispersed phase
- Example applications: Geophysical flows, oil and gas flows, compressible flows with population balance

Model growth and nucleation with the inhomogeneous discrete model (FL)

- Example applications: Crystallization, bubble columns with mass transfer

Effect of expansion on bubble diameter in bubble column with discrete method: Monodispersed bubbles injected at bottom and results compared with analytical diameter (in white)
DQMOM Population Balance captures the segregation of poly-dispersed phases due to differential coupling with the continuous phase (FL)

- Faster solution time than the inhomogeneous discrete model
- Multi-fluid model convects different dispersed phase sizes using different velocities
- Example applications: Fluidized beds, gas solid flows, spray modeling, bubble columns
Multiphase – Eulerian Wall Film

Eulerian Wall Film Model for rain water management, deicing and other applications

- Available models:
  - Momentum coupling
  - DPM coupling
  - Particle collection, splashing and shearing
  - Heat transfer

The wall film on a car mirror with droplets released due to wind shear

Contours of temperature in an Eulerian Wall Film with heat transfer case

\[ \Delta T_{film} = \frac{Q}{\dot{m}C_p} = \frac{3000}{0.01 \times 4000} = 75 \text{ K} \]
Comprehensive CFD Capabilities

Manufacturing companies today face many challenges, ranging from increased product complexity to tightened quality requirements to yield and productivity pressures.

As product complexity increases and the margin for error decreases, CFD must rise to meet growing demands for comprehensive and advanced product capabilities.
Comprehensive CFD – Motion

Improved accuracy with simplicity of immersed solids (CFX)

• Addition of boundary model for more realistic velocity forcing with immersed solids
  – Track nodes nearest to immersed solid
  – Assume constant shear (laminar) or use scalable wall function (turbulent) to modify forcing at immersed solid ‘wall’

• Can improve immersed solid predictions significantly
  – Continuing development for further improvements and broader applications
Comprehensive CFD – Motion

Further user control on deforming mesh to allow retention of optimal mesh quality (CFX) $\beta$

- Diffusion of boundary mesh motion can be defined to be anisotropic – i.e. preferential diffusion in different directions

Sample case: Deformation of an originally square domain

$$\gamma_y = \gamma_x$$

Final mesh with isotropic diffusion: skewed elements

$$\gamma_y \ll \gamma_x$$

Final mesh with anisotropic diffusion: improved mesh quality, greater range of motion possible
Comprehensive CFD – Interfaces

Easy simulation of opening and closing (CFX)
- Conditional GGIs that can open and close as the solution progresses
- Define condition as CEL function of solution
  - e.g. After set time, based on solution values (single or integral values)
- Example applications
  - Membranes bursting, windows shattering, valves opening/closing
- Define as reversible (e.g. valve) or irreversible (e.g. burst membrane)
Comprehensive CFD – Dynamic Mesh

Greater accuracy and speed with default interface settings (CFX)
• Direct intersection method is now default
  – Significantly faster
    • biggest impact on transient cases with re-intersections
  – More accurate than bitmap method

Support for dynamic mesh on meshes with polyhedral elements (FL)
• Smoothing on polyhedral elements
• Polyhedral elements skipped during remeshing
Comprehensive CFD – Turbulence

Focus on wall treatment accuracy (FL)
- More accurate rough wall treatment for epsilon-based models
  - Avoids reduction in roughness when the near-wall mesh is refined
  - New default for all simulations using rough wall treatment

- Reduced sensitivity to the near-wall mesh density for the Spalart-Allmaras model
  - New enhanced wall treatment is the default for S-A turbulence model

Sensitivity of the skin friction coefficient to mesh density in an incompressible flat boundary layer modeled with Spalart-Allmaras
Comprehensive CFD – Turbulence

Improved accuracy for turbulent flows with strong rotation and streamline curvature with one and two-equation models (FL)

- Option to apply a correction term sensitive to rotation and curvature
- Accuracy comparable with the RSM, with less computational effort

Improved accuracy for turbulence near porous jump interfaces (FL) $\beta$

- Use wall functions to include the effects of solid porous material on the near-wall turbulent flow on the fluid side of porous jump interfaces
More accurate solution of high Re wall bounded flows using LES (FL)
- Algebraic Wall Modeled LES (WMLES) formulation based on Smagorinsky model
- Benefits gas turbine combustors and other internal flow applications

Apply scale-resolving turbulence models only locally, as required, to balance cost vs. accuracy (CFX) β
- Zonal LES model using turbulence forcing
- Forcing zone defined by logical CEL function
Validations and model extensions (FL)
• Convective effect for FW-H acoustics solver
  – Option to include the effect of far-field velocity on the generated sound for the Ffowcs-Williams & Hawkins solver
  – Improves accuracy when modeling aeroacoustics and external flows

• Model Doppler effects due to the relative motion of acoustic sources and receivers (FL)
  – For example: Sound from a source moving with a constant speed (airplane, car)
Comprehensive CFD – Radiation

More Efficient Discrete Ordinates Radiation (FL)
• DO radiation calculations ignore solid zones not participating in radiation or participating in heat transfer by conduction only
• Avoids unnecessary CPU time and memory allocations
• Performance improvement varies depending on specifics of case
  – Up to 20% improvement for test cases

<table>
<thead>
<tr>
<th></th>
<th>2x2</th>
<th>3x3</th>
<th>6x6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Serial/Old</td>
<td>218.63 s</td>
<td>269.95s</td>
<td>536.32s</td>
</tr>
<tr>
<td>Serial/New</td>
<td>202.40s</td>
<td>241.56s</td>
<td>429.40s</td>
</tr>
<tr>
<td>Parallel 2-proc/Old</td>
<td>86.4s</td>
<td>107.4s</td>
<td>205.3s</td>
</tr>
<tr>
<td>Parallel 2-proc/New</td>
<td>80.7s</td>
<td>98.4s</td>
<td>167.2s</td>
</tr>
</tbody>
</table>

Sample data for a 3D case with 5 solid zones
Comprehensive CFD – Shell Conduction

Shell conduction: Improved accuracy and ability to include combustion (FL)
• Non-premixed and partially-premixed combustion models
• Example applications: Gas turbines, those with thin walls and combustion

Improved shell conduction usability (FL)
• Updated documentation for wall temperature variables and shell zone and thin wall post-processing
• Post-process the external wall temperature if shell conduction is applied on a one-sided-wall

Oxidizer (air)
V=0.1 m/s, T = 300K
Mean Mixture Fraction=0

Fuel, V=1 m/s
T= 300K
Mean Mixture Fraction=1

External Wall with convective BC
h=2, Tin= 300K

Temperature on a mid-geometry surface
Comprehensive CFD – Reacting Flows

1D Reacting Channel Model (FL)
- Detailed chemistry in the channel (plug flow)
  - Simple geometry, complex chemistry
- 3D heat transfer in the furnace/shell
  - Complex geometry, simple (or no) chemistry
- Example applications: cracking furnace, fuel reformers, ...

1D reacting channel model in Fluent

1D reacting channel geometry in Fluent
Comprehensive CFD – Reacting Flows

Focus on Validation and Verification (FL)
• Real gas
  – RCM1, RCM3
• Surface chemistry
  – Kleijn CVD
• Turbulent non-premixed flames
  – Sydney bluff body flame
• Turbulent premixed flames
  – Chen F3
• Internal Combustion Engines

Example simulation results for the Kleijn CVD verification for surface chemistry
Comprehensive CFD – Combustion

More accurate, flexible, and efficient combustion simulations with CFX-RIF (CFX) β
• Variable thermodynamic conditions
• User-defined mechanisms
• Reduced materials list

Expanded modeling with inert residual material model (CFX) β
• Application example: Safety analyses involving hydrogen combustion together with fire suppression using water
Faster, more accurate, and more general unsteady combustion (CFX) β

- G-equation model upgrade
  - GUI exposure
  - Robustness enhancement
  - Additional correlations for flame speed and flame thickness
  - Improved compatibility with ignition models
  - More accurate and faster flame front re-initialization

Easy customization of non-ideal mixture properties (CFX) β
The polymer, glass, metals and cement processing industries face the special challenge of highly complex material behaviour when performing simulations to minimize physical prototyping in the manufacture of extrusion dies or improve the quality of thermoformed or blown products.

Importantly, for say the packaging industry, is the ability to predict more accurately the structural performance of the package in real-use scenarios.

ANSYS Polyflow provides a host of special material models and enhanced capabilities to meet these needs.
ANSYS Fluid Dynamics 14 contains enhancements in all products...

- ANSYS Fluent, ANSYS CFX, ANSYS CFD-Post
- ANSYS Turbogrid
- ANSYS Polyflow

...for rapid and robust meshing, workflow and usability, HPC and solver speed, rotating machinery, automotive and multiphase applications as well as overall to provide comprehensive CFD.
## Release Timeline

|------------|-------------|------------|------------|
| **Large release**<br> ✓ Fluent, CFX, Polyflow<br> ✓ Rapid meshing<br> ✓ Adjoint<br> ✓ System coupling<br> ✓ Industry specific<br> ✓ ...<br> **Development starting soon**<br> **Feature release**<br> ✓ Fluent, CFX, Polyflow<br> ✓ Hybrid vehicles<br> ✓ Turbomachinery<br> ✓ Full systems<br> ✓ ...<br> **Release the results of today's R&D topics!**<br> ✓ Fluent, CFX, Polyflow<br> ✓ Hybrid vehicles<br> ✓ Turbomachinery<br> ✓ Full systems<br> ✓ ...<br> **Release the results of today's R&D topics!**<br> ✓ Advanced physics<br> ✓ Best in class numerics<br> ✓ Parallel meshing<br> ✓ Reduced order modeling<br> ✓ Design automation<br> ✓ Co-simulation<br> ✓ Parameterization<br> ✓ ...
Questions?
Enhanced Workflow and Usability in Workbench

• Polydata will support named boundaries and subdomains, using Named Selections in ANSYS Workbench

• ANSYS POLYFLOW supports 1-D and 2-D pmeshes generated in ANSYS Meshing and ANSYS ICEM CFD
  – 1-D pmeshes for 2-D meshes
  – 1-D and 2-D pmeshes for 3D
Enhanced Usability in Workbench from Templates

• A database is now available of ANSYS Workbench Project Templates for common applications of ANSYS Polyflow – *Templates* folder in the POLYFLOW 14.0 installation

• Each project contains a complete calculation (Geometry to Post-processing) with parameters and a final report
  – Blow-molding, extrusion, thermoforming examples in ANSYS Workbench archived (.wbpz) format

• The template projects can be used for learning purposes or as starting points for individual projects.
Enhanced Extrusion Solutions from use of CutCell Meshes

- ANSYS POLYFLOW 14.0 can import and run Cutcell meshes for complex extrusion dies
  - New mesh generation method in Tgrid and ANSYS Workbench
  - Fast and efficient algorithm for rapid hex dominant mesh generation on highly complex geometry, such as extrusion dies

- A “Sliceable” mesh can be created for the free jet region from a CutCell mesh, to allow computation of final extrudate shape
- Creates conformal prismatic mesh in the free jet subdomain with CutCell mesh in the die
- Take full advantage of rapid hex dominant CutCell meshing for complex die geometries
• ANSYS POLYFLOW will now allow numerical decoupling of species and VOF volume/fill fractions from the flow calculation
  – Reduce memory requirements and improve solution speed by segregating the volume fraction field
• This is very useful for applications like multi-fluid co-extrusion (and glass forming)

Fill fraction of the primary fluid in multi-fluid co-extrusion: Flow through a complex die modeled with VOF method
Enhanced Solutions for Blow Molding and Thermoforming

- A force can be specified for mold motion with shell meshes for parisons in blow molding and thermoforming
- Specify maximum displacement of the mold to set a bounding criterion for the motion
- A new contact release method has been implemented for blow molding and thermoforming
- This allows determination of Parison “peeling” from mold for blow molding and plug release in thermoforming
- Contact release is compatible with both specified force and specified velocity for the mold
Virtual Prototyping for Packaging Applications - Mechanical

- Thickness data from ANSYS POLYFLOW can be sent to ANSYS Mechanical and ANSYS Explicit Dynamics via an automatic connection
- Complete Virtual Prototyping and Testing capability in ANSYS Workbench for packaging manufacturing:
  - Simulate blow molding or thermoforming process to get final thickness distribution with Polyflow
  - Based on Polyflow predicted variable thickness, perform stress and deformation analysis
- Top load stress analysis with ANSYS Mechanical shown here
Virtual Prototyping for Packaging Applications – Explicit Dynamics

• Where the deformation or loading rate are larger, then transfer the Polyflow thickness data to Explicit Dynamics system in Workbench
  – Explicit Dynamics is available as part of ANSYS Explicit STR and ANSYS AUTODYN

• Here we will load a blown part (created from Polyflow) with a rigid plate (in green in mesh plot)
  – Loading normal to the plate