ANSYS Fluid Dynamics

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

ANSYS Fluid Dynamics at 14.0
• Significant benefits for users
  • Including geometry and meshing
  • Look for detailed materials at release time
Product Creation Teams (~600 Members)

**Central Development Unit**
- Brian Drew
  - Framework, Tools
  - Geometry/CAD
  - Meshing
  - Engineering Knowledge Manager
  - Design Explorer
  - Dev Services
  - Funded development
  - ~30% dev. staff

  Technologist: Chris Hawkins
  Product Mgmt: Scott Gilmore

**Physics Business Unit**
- Joe Solecki

  **Fluids Systems**
  - Nelson Carter
    - FLUENT, CFX, common platform
    - Specialty CFD
    - R&D initiatives
    - Funded development
    - ~25% dev. staff

  Technologists: Dave Conover, Michael Engelman, Dipankar Choudhury
  Product Management: André Bakker

  **Mechanical Systems**
  - Joe Manich
    - Mechanical
    - Explicit
    - Multibody dyn.
    - R&D initiatives
    - Funded development
    - ~25% dev. staff

**Electronics Business Unit**
- Shane Emswiler
  - High Frequency
  - Low Frequency
  - Electro-mechanical
  - Signal Integrity
  - RF/Electronics
  - Funded development
  - ~ 20% dev. staff

  Dev. Mgmt: Nancy Lambert
  Product Mgmt: Larry Williams

**Product Strategy & Planning, Partnerships, Corporate Product Management**
- Josh Fredberg, S. Subbiah, and Todd McDevitt
Our Fluids Strategy

We are developing both Fluent and CFX:
• Both offer best-in-class solver technology, relied upon by leading industrial companies
• New feature releases are already being planned (R14.x and R15)
  – Increased technology sharing between the two solvers
  – Continued improvements in common Workbench based workflow

In addition, we remain committed to developing and releasing an integrated, unified fluids product:
• That draws upon the best of both Fluent and CFX, as well as add new functionality
• That provides smooth adoption for our fluids customers

The first release of this product is targeted for ANSYS R15
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
- Solver meshing
Assembly Meshing enables dramatically reduced time to mesh for typical CAD models by eliminating the tedious geometry clean-up.

Top-down approach to mesh all parts at once
- Uses Virtual Bodies (material points or groups) to automatically extract internal regions from assemblies
- Supports:
  - Meshing solids from sheet bodies
  - Conformal mesh between parts without requiring multibody parts
  - Overlapping bodies
  - Tet (linear) and CutCell (hex-dominant) mesh types
  - Inflation
Productivity enhancements through increased robustness, flexibility, and efficiency

- Stability and usability
  - ~160 defects and feature requests resolved
- User interface
  - Speed (display, selection)
  - Model tree
- Tetra/Prism
  - Prism editing
  - Upgraded TGrid implementation
- Hexa
  - Options, usability, smoothing
  - MultiZone
Rapid & Robust Meshing – TGrid

Productivity enhancements through increased robustness, flexibility, and efficiency

• Assembly Meshing
• CutCell-to-Tet conversion with remeshing
• Prism speedup (~2x)
Solver meshing

Fluent MDM (moving-deforming mesh) improvements: 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
  – Polyhedral support: can now use MDM for meshes that partially consist of polyhedra

Remeshing a tetrahedral mesh with boundary layers during a simulation

Tetrahedral mesh with boundary layers after remeshing
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.
Workflow & Usability – Geometry Advances

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
  – GAMBIT reader improved
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 DOEs
    • Distributed solve
    • Design point sorting
  • Increased robustness
    • Reserved licensing
    • Support for partial DOE’s
  • Increased understanding
    • New charts
    • Improved GOF
    • Project report
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)
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 reduced data for faster file write and reduced storage needs

• Fluent – on a per cell basis to CFD Post, EnSight, and FieldView
• CFX - Output selected solution data only at boundaries during transient
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 electromagnetics within a single, unified engineering simulation environment using systems coupling.
Multiphysics and Systems Coupling – FSI

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 → 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
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)
  - Out of the box performance
    - New defaults
      - 2nd order for some equations
      - Hybrid initialization
    - Solver optimizations
      - Higher order term relaxation (HOTR)
    - More aggressive initial settings
    - Improved convergence
  - Migration manual
- Beta: NRBCs with PBNS (FL)
  - Compatible with combustion models

- Standard @ CFL=200
- HOTR @ CFL=200
- Standard @ CFL=100
- HOTR @ CFL=100

Normalized scales residuals comparing HOTR to standard relaxation at different CFL settings

RAE-2822 Airfoil
M=0.73
AOA=2.8 deg
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
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
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 >\(~100\)M 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
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
• Improved turbo-specific post-processing in CFD-Post
Rotating Machinery

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

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

12% fines
0-25 sec

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

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

Water condensation near a wall at a fluid-solid interface
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

Coupled VOF after 500 iterations

Coupled P-V, Segregated VOF after 1400 iterations
Multiphase – Eulerian

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

Test case based on the data in Hoyers et. al. showing dry out at the wall

IAC model allows bubble diameter variation at different stations along the pipe to be modeled. Data from: Bartoleimei et. Al
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

![Graph](attachment:image.png)

- Velocity big bubbles > velocity small bubbles
- All bubbles move with same velocity

![Graph](attachment:image.png)

- Bubble diameters for DQMOM (white) compared to QMOM (red)
- Contours of volume fraction of the phase with the largest diameters in a bubble column.
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

\[
\Delta T_{film} = \frac{Q}{mC_p} = \frac{3000}{0.01 \times 4000} = 75 \text{ K}
\]

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

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

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) ^B

- Zonal LES model using turbulence forcing
- Forcing zone defined by logical CEL function

Flow over a wall mounted hump

A mixing layer with resolved turbulence using SAS initiated by the forcing model
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

Geometry and details for shell conduction with non-premixed combustion test case

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, Tinf= 300K

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

Focus on Validation and Verification (FL)

• Real gas models:
  – RCM1, RCM3

• Surface chemistry model:
  – Kleijn CVD

• Turbulent non-premixed flame model:
  – Sydney bluff body flame

• Turbulent premixed flame model:
  – Chen F3

• Internal Combustion Engines

Example simulation results for the Kleijn CVD verification for surface chemistry
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.
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
Summary

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

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Large release</strong></td>
<td>Development starting soon</td>
<td>Release the results of today’s R&amp;D topics!</td>
<td></td>
</tr>
<tr>
<td>✓ Fluent, CFX, Polyflow</td>
<td>✓ Fluent, CFX, Polyflow</td>
<td>✓ Fluent, CFX, Polyflow</td>
<td></td>
</tr>
<tr>
<td>✓ Rapid meshing</td>
<td>✓ Hybrid vehicles</td>
<td>✓ Advanced physics</td>
<td>✓ Fluent, CFX, Polyflow</td>
</tr>
<tr>
<td>✓ Adjoints</td>
<td>✓ Turbomachinery</td>
<td>✓ Best in class numerics</td>
<td>✓ Fluent, CFX, Polyflow</td>
</tr>
<tr>
<td>✓ System coupling</td>
<td>✓ Full systems</td>
<td>✓ Parallel meshing</td>
<td>✓ Fluent, CFX, Polyflow</td>
</tr>
<tr>
<td>✓ Industry specific</td>
<td>✓ ....</td>
<td>✓ Reduced order modeling</td>
<td>✓ Fluent, CFX, Polyflow</td>
</tr>
<tr>
<td>✓ ...</td>
<td>✓ ...</td>
<td>✓ Design automation</td>
<td>✓ Fluent, CFX, Polyflow</td>
</tr>
</tbody>
</table>

### Improved Features
- Fluent, CFX, Polyflow
- Rapid meshing
- Adjoints
- System coupling
- Industry specific
- Hybrid vehicles
- Turbomachinery
- Full systems
- Advanced physics
- Best in class numerics
- Parallel meshing
- Reduced order modeling
- Design automation
- Co-simulation
- Parameterization
- ...