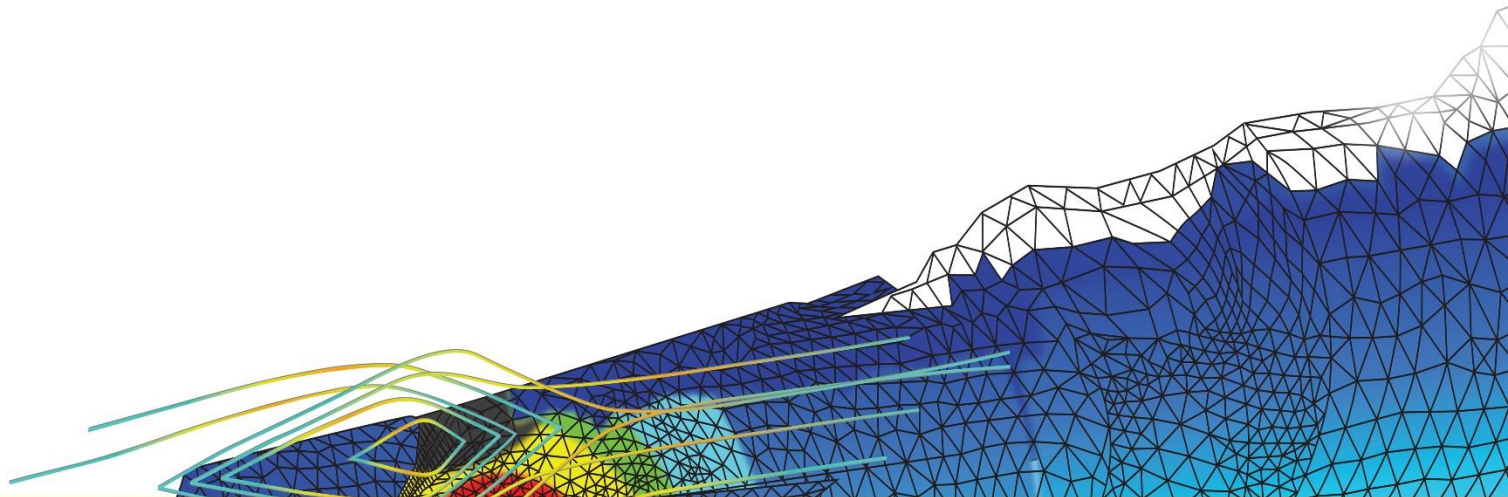




# ANSYS Fluent and CFX R17.x

ANSYS UK

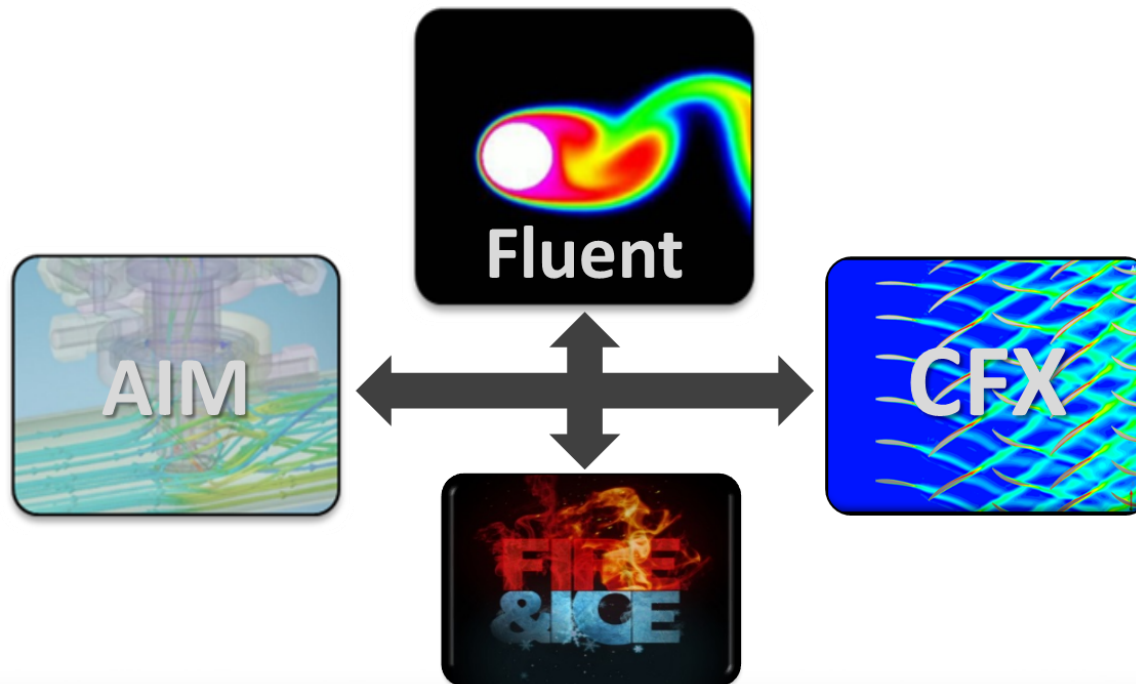


# Agenda

- **Fluids Roadmap**
- **Model Preparation**
- **Solver Usability and Performance**
- **Model Improvements**
- **ANSYS 17.2**
- **Future Objectives**
- **ANSYS CFD**
- **Q & A**

# Fluids Roadmap

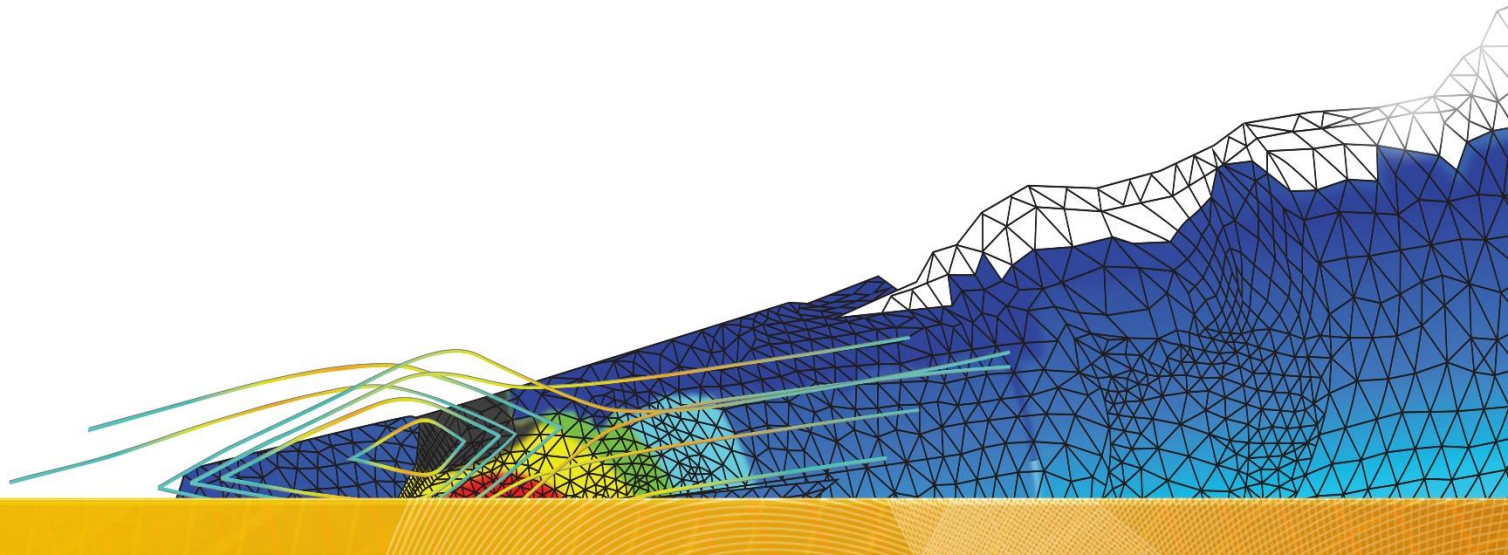
- **ANSYS is committed to developing our portfolio of CFD tools**
  - **Fluent, CFX, Polyflow, Forté**
  - **These are now bundled together as ANSYS CFD**
  - **ANSYS AIM has recently been launched, featuring a new CFD solver**
    - **This will be presented in the following talk**
    - **ANSYS is in the process of deciding how to make it available to existing CFD customers**





# Model Preparation

Geometry and Meshing

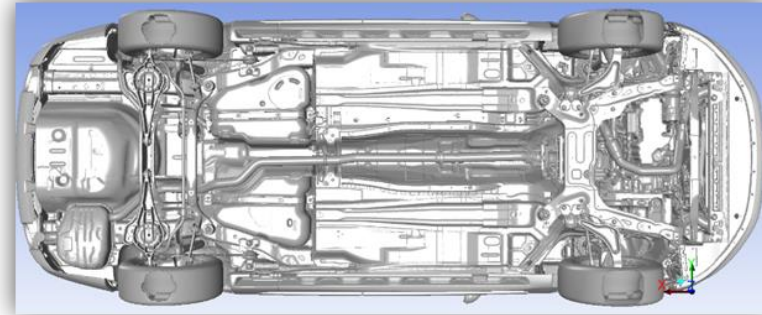


# Accurate CFD requires a big effort

## Problem

**ANSYS CFD is recognized for its outstanding accuracy but...**

- Large CAD models presented problems
- Repetitive manual effort to prepare models
- Simulation size (and time) grew faster than resources available to solve them
- Accelerating trend towards CFD with design engineers – *Not experts*



**CAD-based applications might seem to be fast and easy but force users to settle for “good enough” results**



# Now You Can Have It All

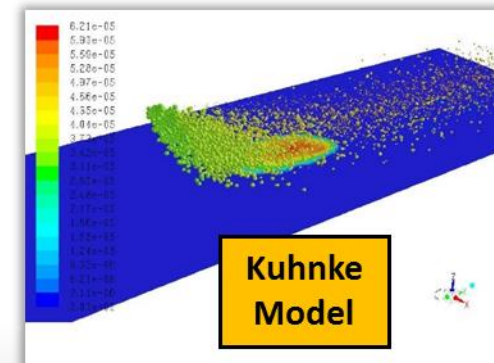
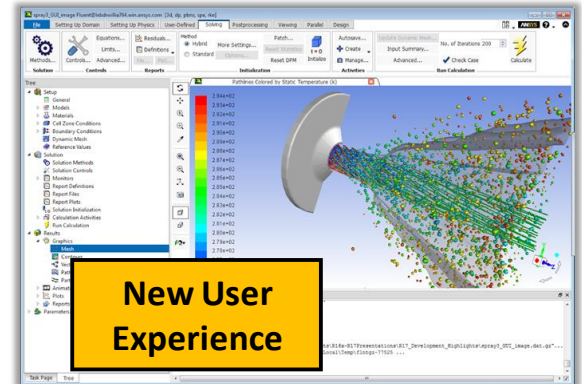
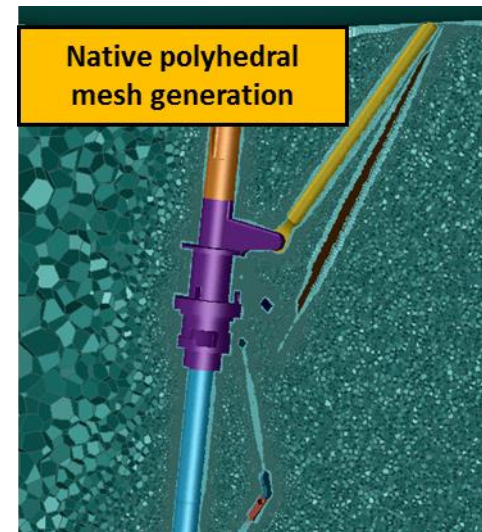
## Solution

ANSYS 17.0 makes it **easier than ever for every engineer** to create **well-validated results**

**Innovations across the entire workflow radically accelerate the time to results** without compromising accuracy

ANSYS 17.0 **reduced pre-processing time** for complex geometries **by up to 40-80%**

**Many new capabilities** extend the range of simulation possibilities



# Volvo S80 External Aerodynamics – Scripted Automatic Meshing

## Challenges

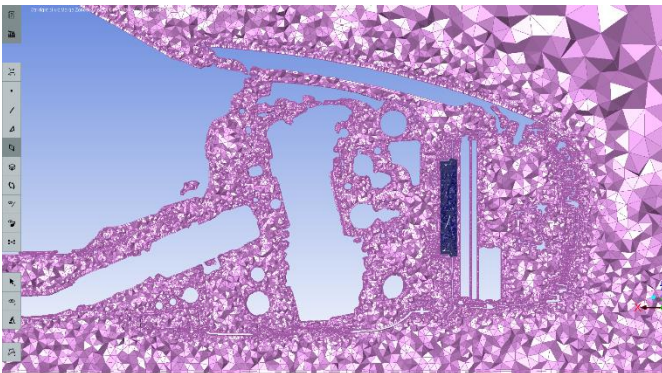
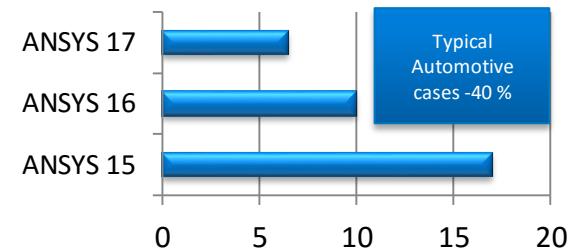
- Too many parts and too many details
- Many zero thickness baffles
- Disconnected facets
- Many duplicates, overlaps and intersections
- Leakage to engine, transmission, and cabin
- Two fans – MRF
- 3 Heat Exchangers

## Solutions

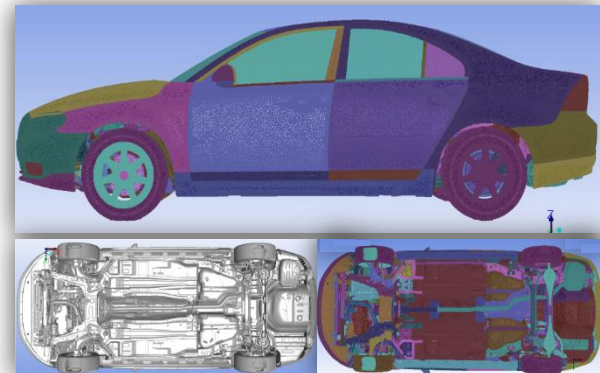
- SpaceClaim Direct Modeler simplifies model clean up
- Scripting speeds complicated repetitive processes
- New, faster meshing tools

## Results

- 2-3 days CAD/Mesh Effort
- 95M Cell Count (2 layers)

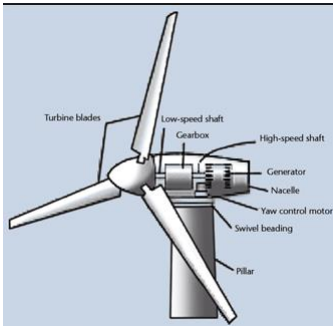


Courtesy of Volvo



Courtesy of Volvo

# Processing Times for Complex Geometries in ANSYS 17.0



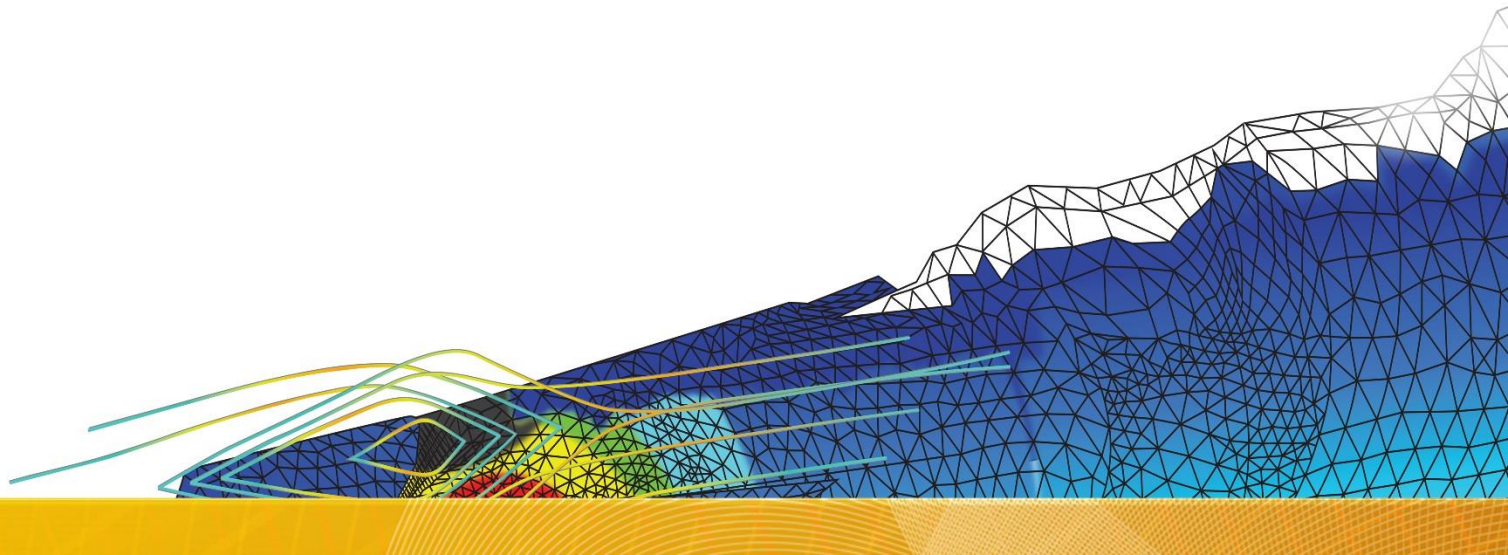
Application	Objective	Cell Count	Effort
Loader Machine	Front End Air Flow	28 million tet (2 layers)	1 day
Small Truck	Front End Air Flow	46 million tet (2 layers)	1 day
Oven	Flow Rate	55 million tet (3 layers)	1.5 days
1/8 <sup>th</sup> Scale Truck and Trailer	External Aerodynamics + Front End Air Flow	40 million hexcore	1 day
Full-Scale Truck and Trailer	External Aerodynamics	95 million hexcore (3 layers)	1 day
Wind-Turbine	External Aerodynamics	46 million (10 layers)	0.5 day
Combustor (1 sector)	External Aerodynamics	54 million (6 layers)	0.5 day
Nacelle	External Aerodynamics	45 million (4 layers)	1 day
Oil Refinery	External Aerodynamics	40 million (10 prism layers)	0.5 day
Passenger Car	External Aerodynamics	80 million (2 layers) → 170 million (21 max layers - all adapted)	2-3 days
Motorcycle	Under Hood Thermal Management	10 million (2 layers)	< 0.5 day

*Times may vary.*





# Improved Usability and Performance in ANSYS Fluent

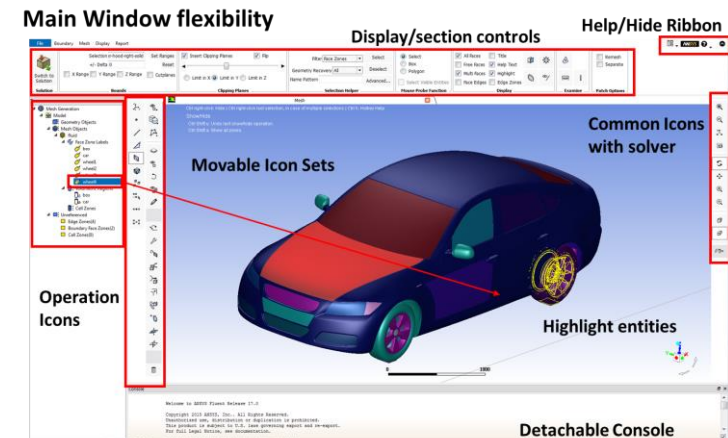
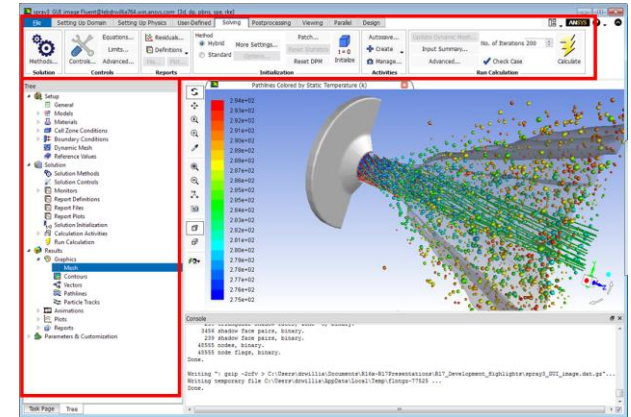


# Fluent New User Experience

## Solution

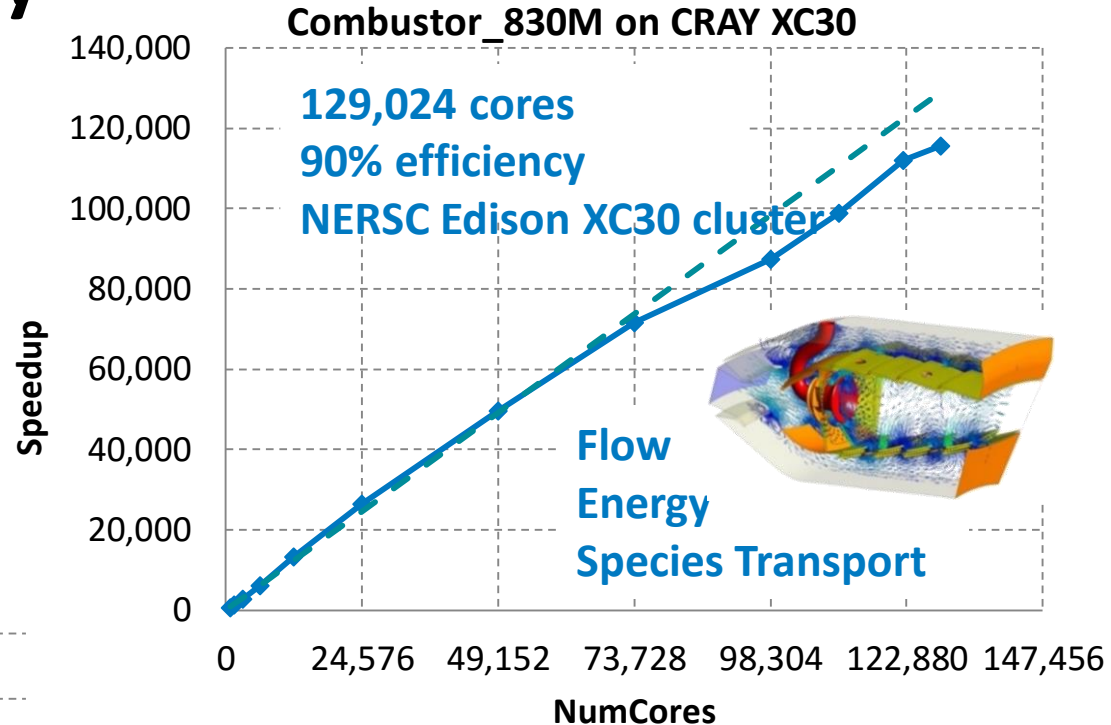
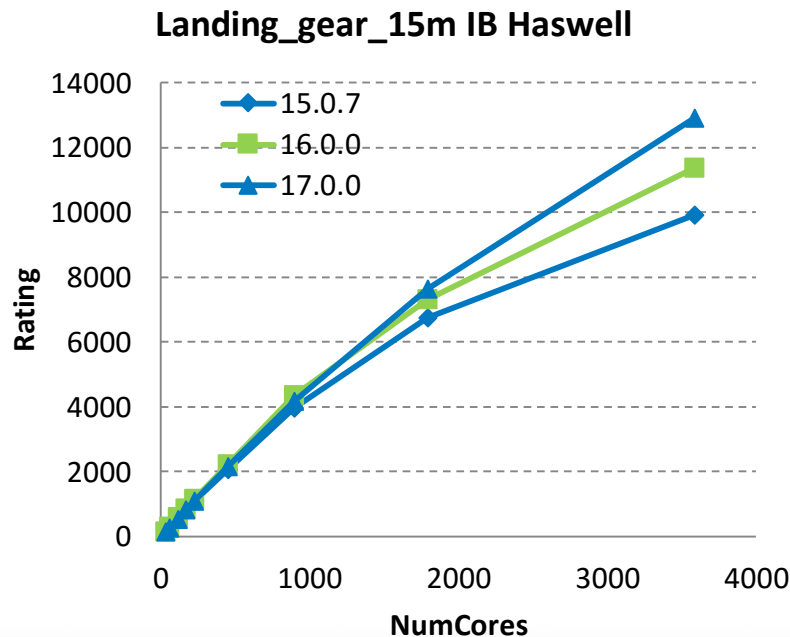
**ANSYS 17.0 Fluent and Fluent Meshing user interface has workflow that is easily learned by new or infrequent users, while remaining efficient, powerful and familiar to experienced users.**

- **Ribbon-style tool bars and other improvements make navigation more intuitive, faster, reducing the number of mouse clicks.**
- **Workflows can be completed with up to 12% fewer clicks**
- **Video overview available**



# Fluent HPC Scalability

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



# Faster METIS Partitioning

## Solution

Updated library and optimized algorithms deliver significant partitioning speed-up for many larger cases, particularly those with adapted meshes

- 64-bit indexing in METIS and for partition storage to enable larger models (Future proofed: Tested up to 2 billion cells)

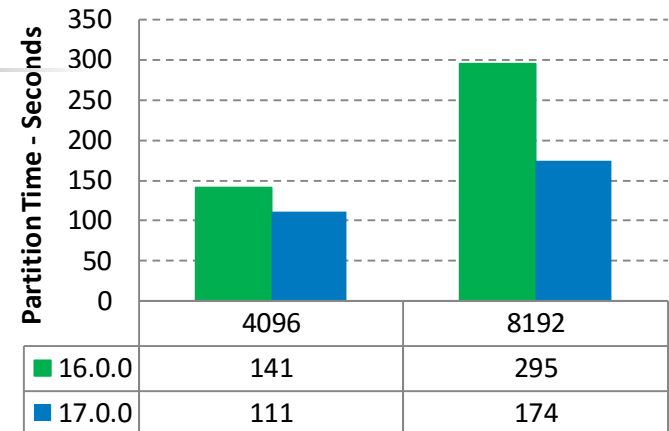
### Combustor:

- 40% faster to partition for 8192 cores
- Less than 3 minutes

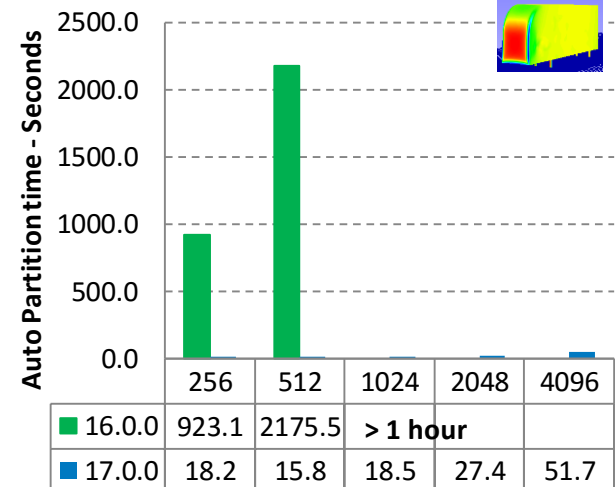
### Truck:

- 99% faster to partition for 512 cores
- Just 18 seconds (versus 36 minutes!!)

Combustor 830M Cells CRAY XE6



Truck 134M Cells





# Model Weighted Partitioning

## Problem

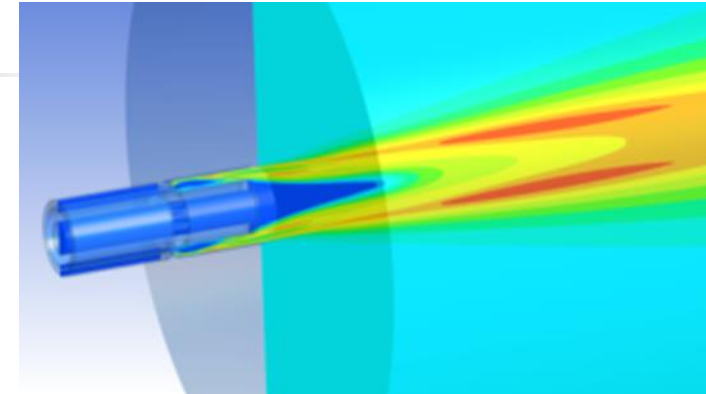
DPM and combustion models pose challenges to parallel performance as users attempt to load-balance flow and physics calculations

## Solution

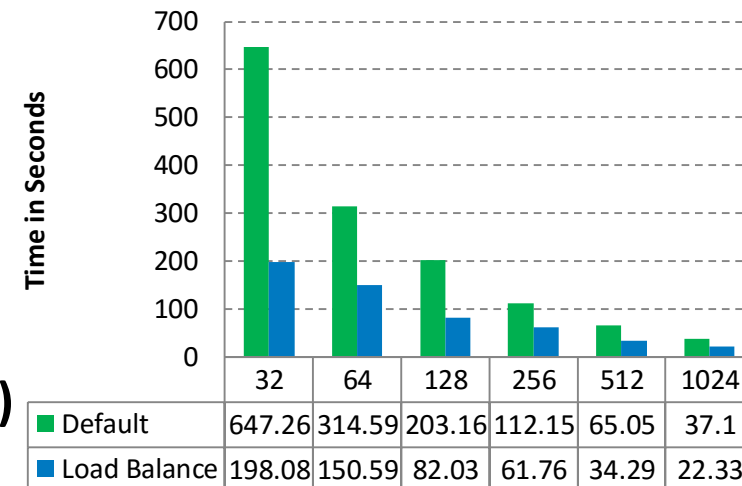
Model-Weighted Partitioning automatically weights multiple physics models across the full set of processors within a specified load imbalance tolerance . Users can select the factors and relative weightings.

### Oxy-Fuel Burner:

- **60% faster for 128 cores (Just 82 seconds)**
- Turbulence, combustion, radiation, detailed kinetic mechanism (25 species, 113 reactions)



Oxy-fuel Burner, 1.9M hex cells



# Neighborhood Creation Optimization

## Problem

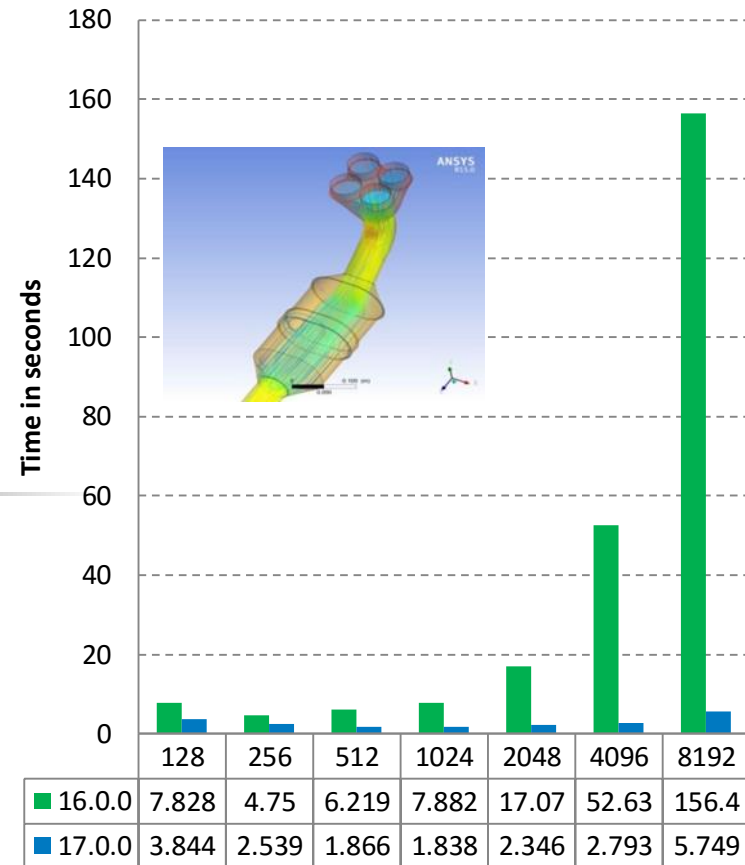
Partitions need to communicate with each other. Lack of optimization can slow performance, especially for moving/dynamic mesh cases where the neighborhood needs to be updated frequently

## Solution

Optimize communication algorithms and improve interface identification for better performance and completeness

- Speed-up from 1X to 30X depending on case and number of cores
- Better identification of interfaces improves robustness

Exhaust 33M Neighborhood Creation



# Big Speed-Up for Moving Dynamic Mesh

## Example Case

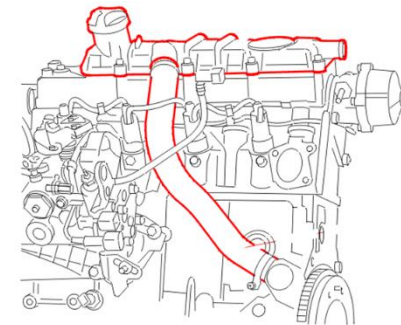
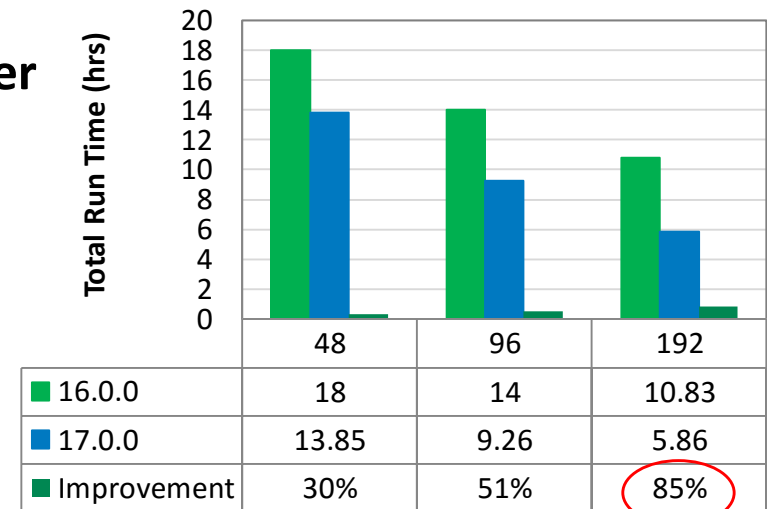
Cumulative speed-up from multiple solver improvements

- Neighborhood optimization
- Sliding interface optimization
- Parallel solver optimization

Engine Crankcase Lubrication Model:

- 85% faster run time (<6 hours)
- Faster than recent competitive benchmark
- Crankshaft Rotation in a sliding mesh zone, Piston motion through dynamic mesh layering, Oil slosh modeled with VOF, 5M cell Poly Mesh

Engine Crankcase Lubrication Model  
Total Run Time per One Cycle



Representative Illustration

# Faster Moving Dynamic Mesh with Combustion

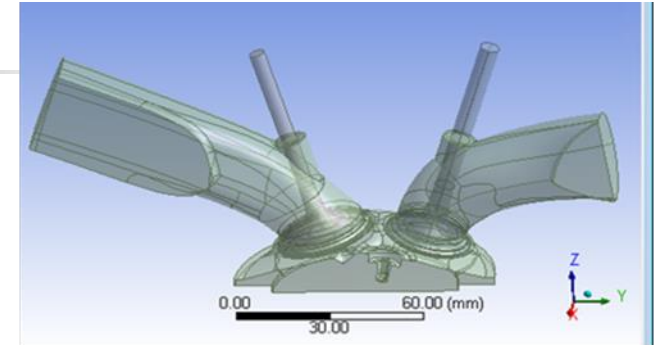
## Example Case

Cumulative speed-up from multiple solver improvements

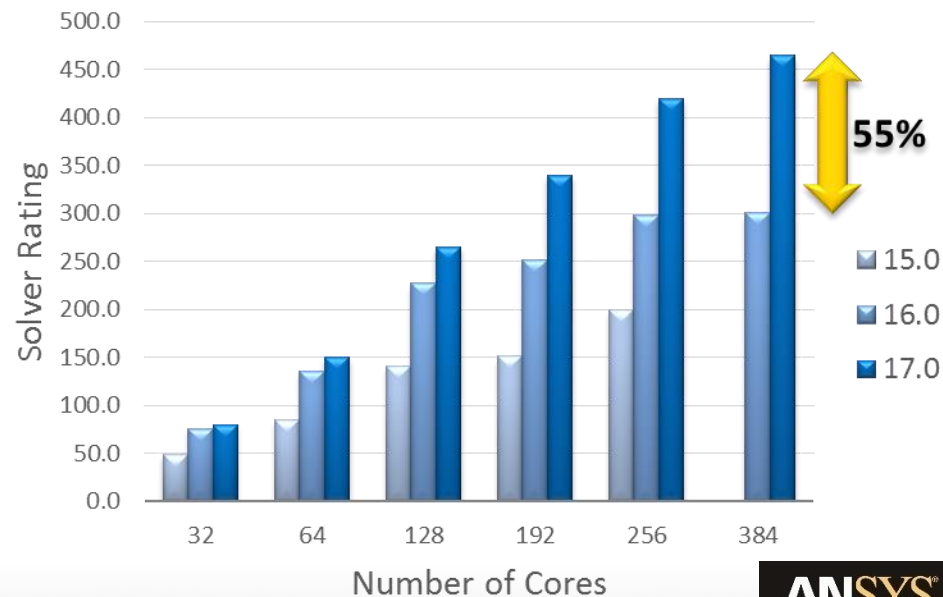
- Neighborhood optimization
- Sliding interface optimization
- Parallel solver optimization
- Combustion code refactoring

In-Cylinder Combustion Model:

- 55% faster at 384 cores
- 7 cell zones, MDM, Spray, Partially premixed, 1.6 million cells



2M-Cell In-Cylinder Combustion Case





# Faster, More Robust Convergence

## Problem

Fluent's priority has been to deliver the best results, not the fastest convergence

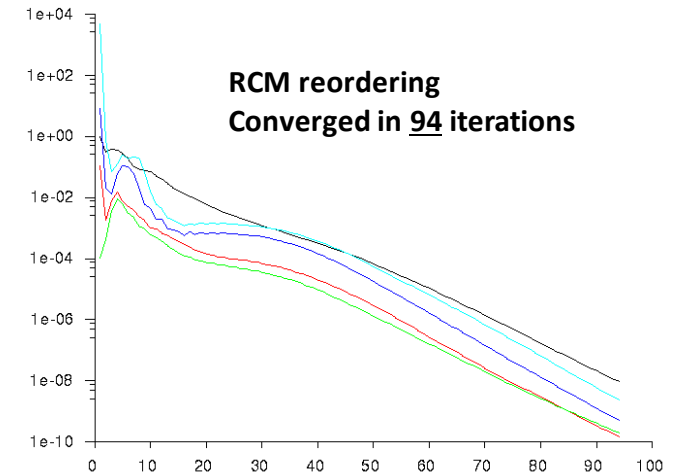
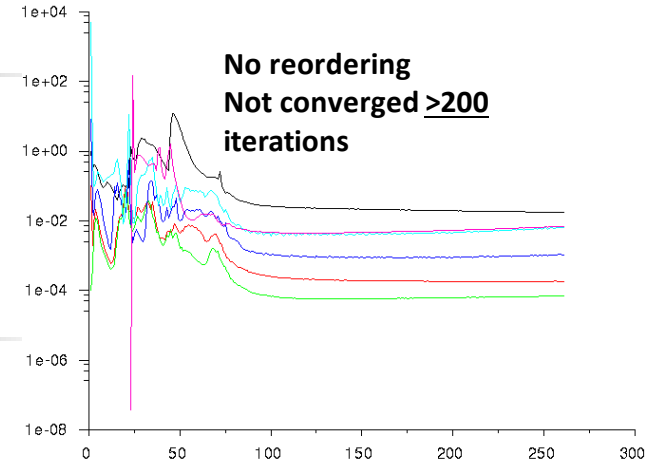
## Solution

Conservative Coarsening improves convergence for pressure-based coupled solver

- Especially helpful for native polyhedral meshes and/or highly stretched cells

Algebraic multigrid (AMG) solver now automatically reorders the linear system

- Ensures proper ordering in multiple cell zones (was limited to within a single cell zone)



# Warped-Face Gradient Correction

## Problem

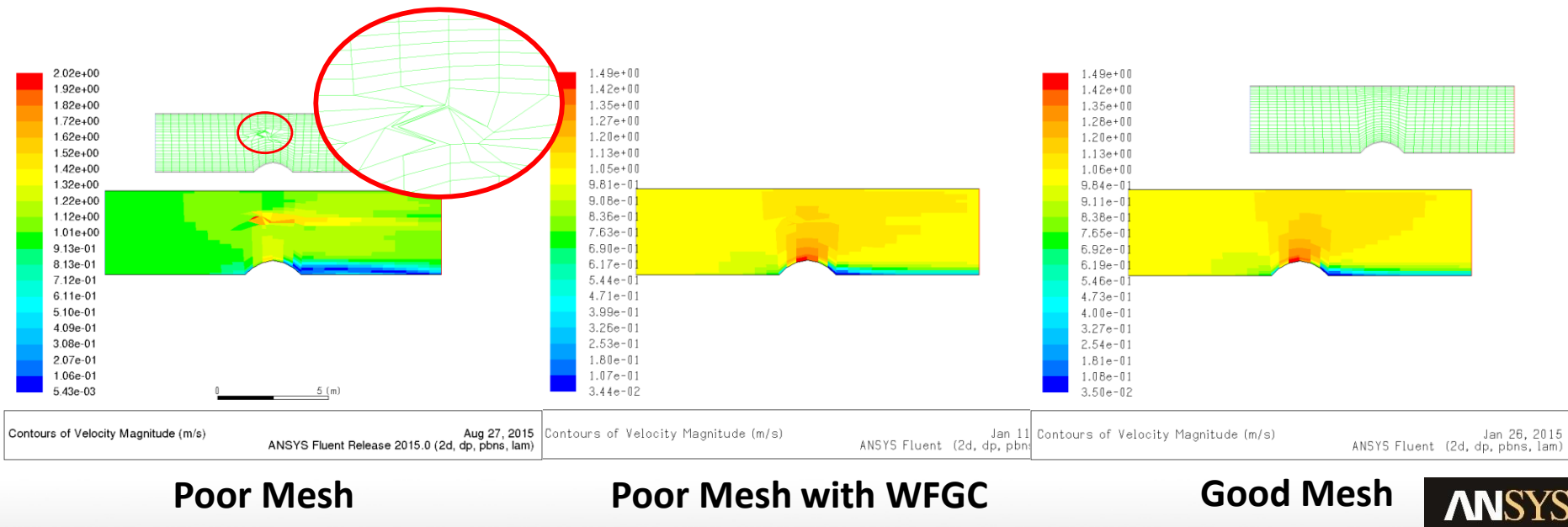
Hex and poly cells with highly non-planar faces or centroid outside of cell require careful, time consuming repair to ensure good results

## Solution

Gradient correction option available for the pressure-based solver

- Speed-optimized
- Memory-optimized

Delivers results closer to good mesh results



# User Defined, Complex Bounding Envelope for Adjoint Solver

## Problem

Must ensure that morphed shapes do not end up violating a physical envelope within which they must fit when performing design optimization

## Solution

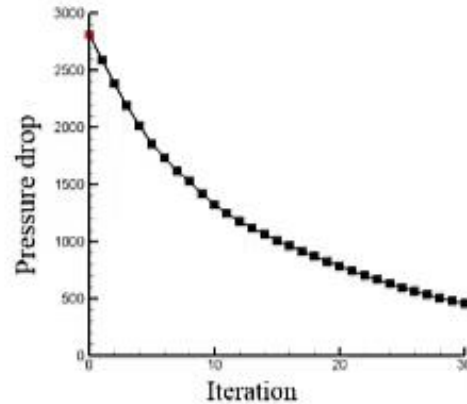
- Now any arbitrary STL surfaces can be applied as boundaries for the design morphing
- Ideal for under-dashboard ductwork or other tightly-packaged applications

## Other Enhancements:

- Compressible flows are now supported
- Selected wall regions can have user-defined rotation, scaling, or translation
- Reports to the user when constraints that they have applied are in conflict
- STL Export

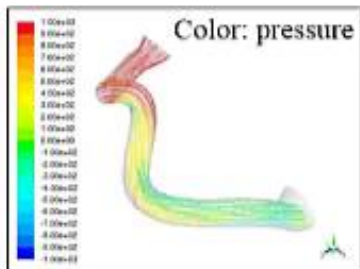
# Adjoint Solver Optimizes a Duct Within a User Defined Envelope

## Example Case



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.

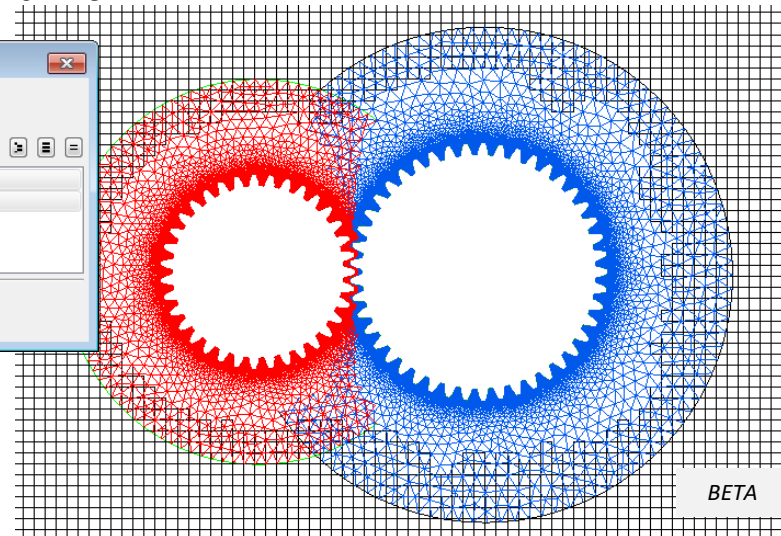
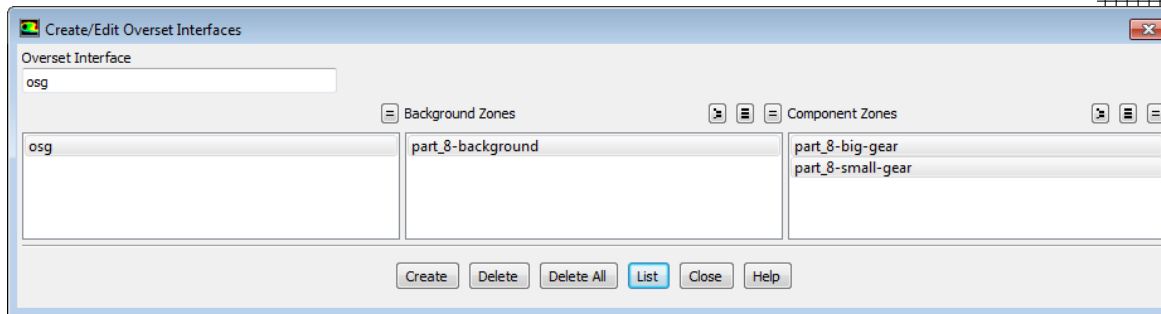




# Overset Mesh

## 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

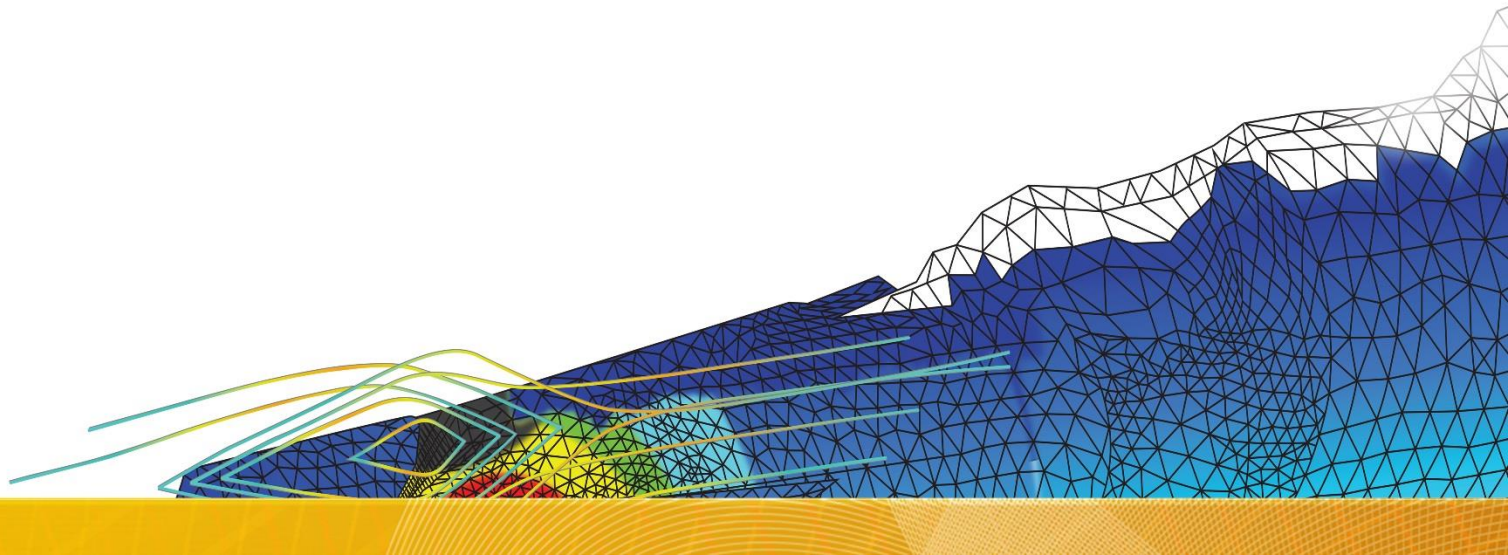


## Limitations:

- DBNS not yet supported
- Node-based gradients not yet supported

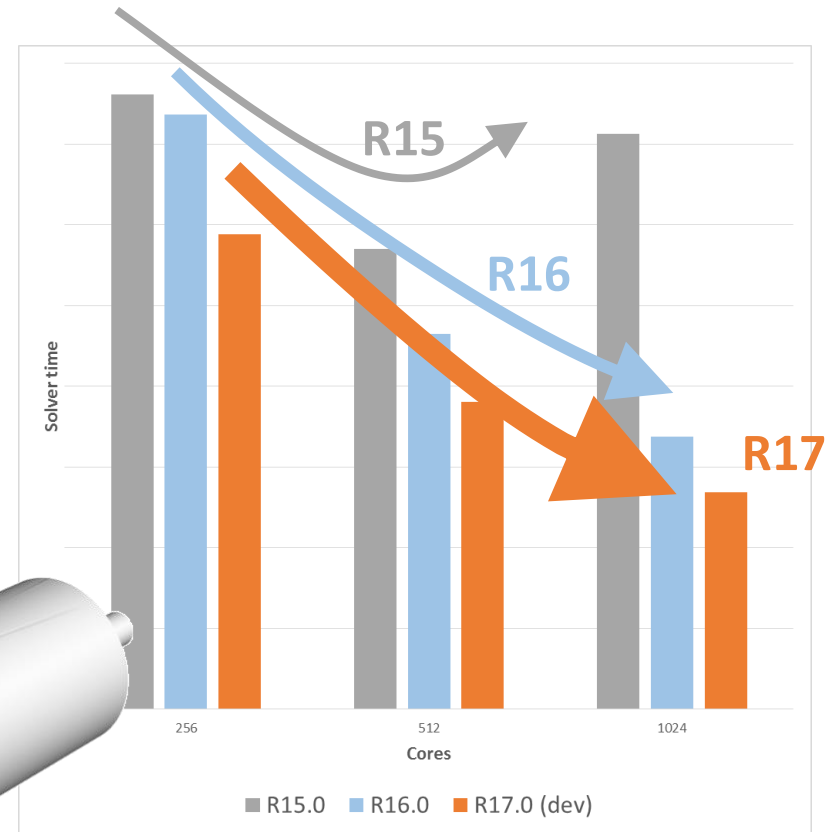
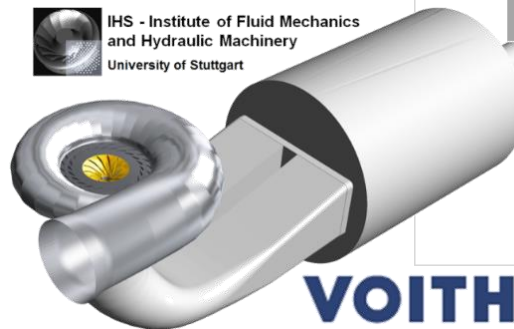


# Improved Usability and Performance in ANSYS CFX



# CFX HPC Performance

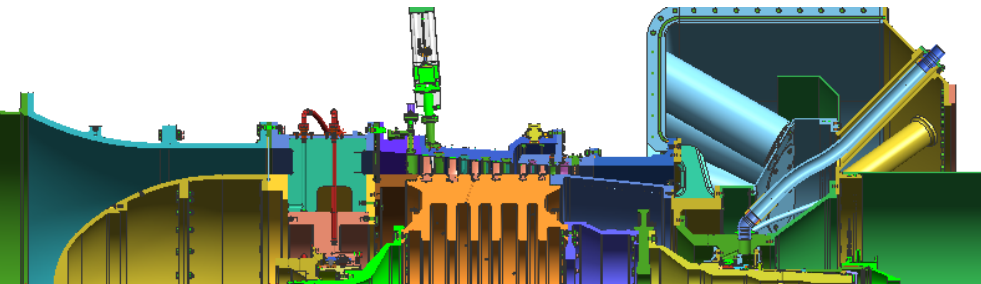
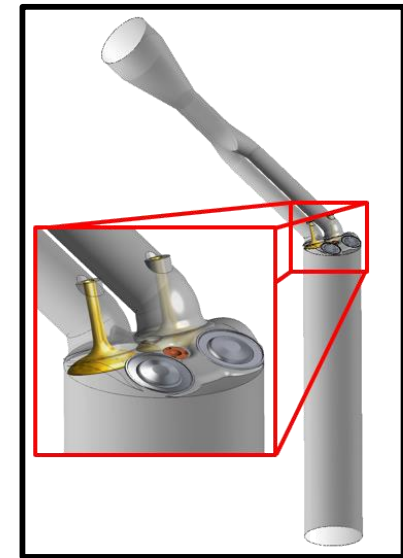
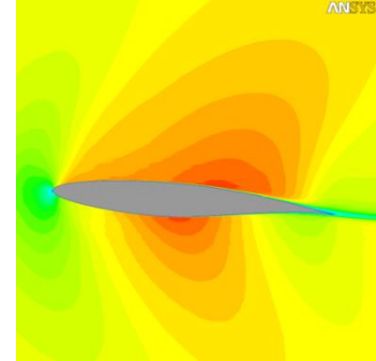
- Continuously driving down simulation times!
- Special focus on Transient Rotor-Stator
- Multiple areas improved
  - Assembly/discretization
  - Linear solver
  - GGI intersection
  - ...



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)

# CFX HPC Performance

- Numerous other benchmarks
  - Transient full hydro turbine application
    - 10-30% reduction in solver time
    - 20% improved scalability at 512 cores
  - External aerodynamics application
    - ~40% faster on 4096 cores
  - Engine internal flow application
    - >30% less solver time, scaling to 4096 cores
  - 6 stage transient axial compressor application
    - 20-30% reduction in solver time



*Courtesy Siemens AG, Mülheim, Germany,  
ASME IGTI Paper GT2013-94639*

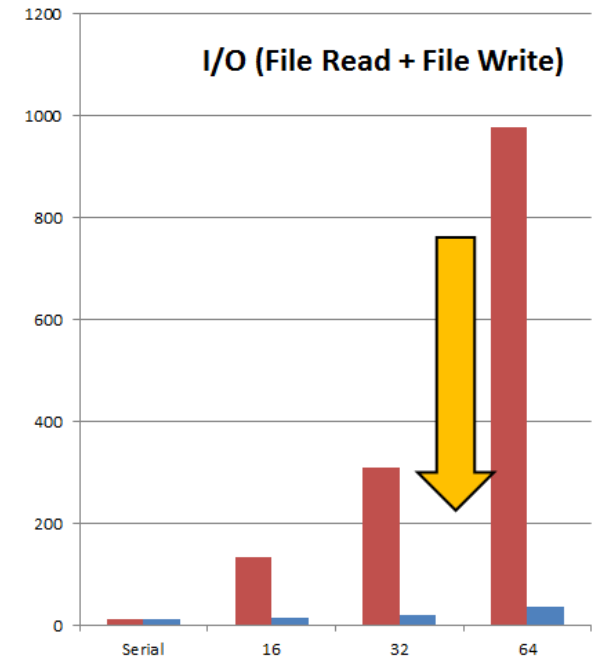
**ANSYS**



# 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**

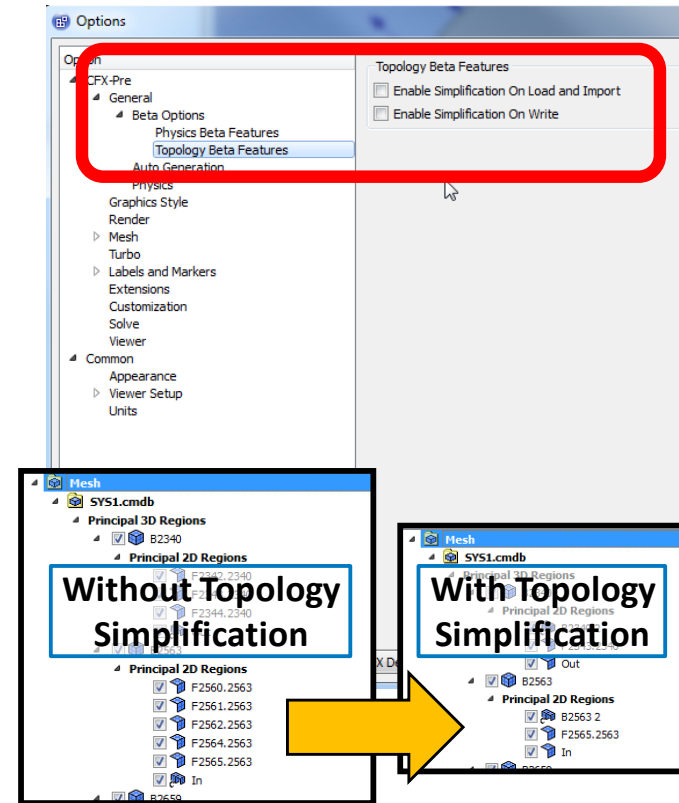
- Previously could become large enough to impact overall solution time
- Activated automatically
  - **No additional setting required**



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

# Topology Simplification ( $\beta$ )

- Complex CAD models contain many regions that the user may not need
- Represent unnecessary overhead that cause inefficiencies in the UI and solver (e.g. many unused primitive regions)
- You can avoid the possible overhead of extraneous regions
- Limit import and/or write to minimum
- Also note change in options menu to activate beta features

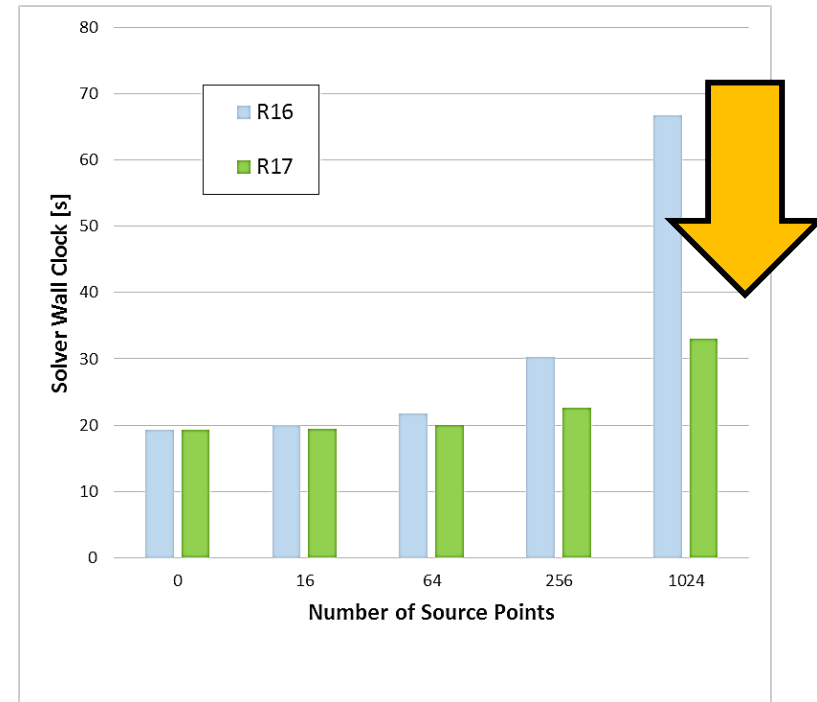


Example import (of a simple case) with and without topology simplification shows how the number of imported regions are pared down

# Source Point Performance

## Improved efficiency with large numbers of source points

- Example application: modeling large numbers of cooling flow injections as source points

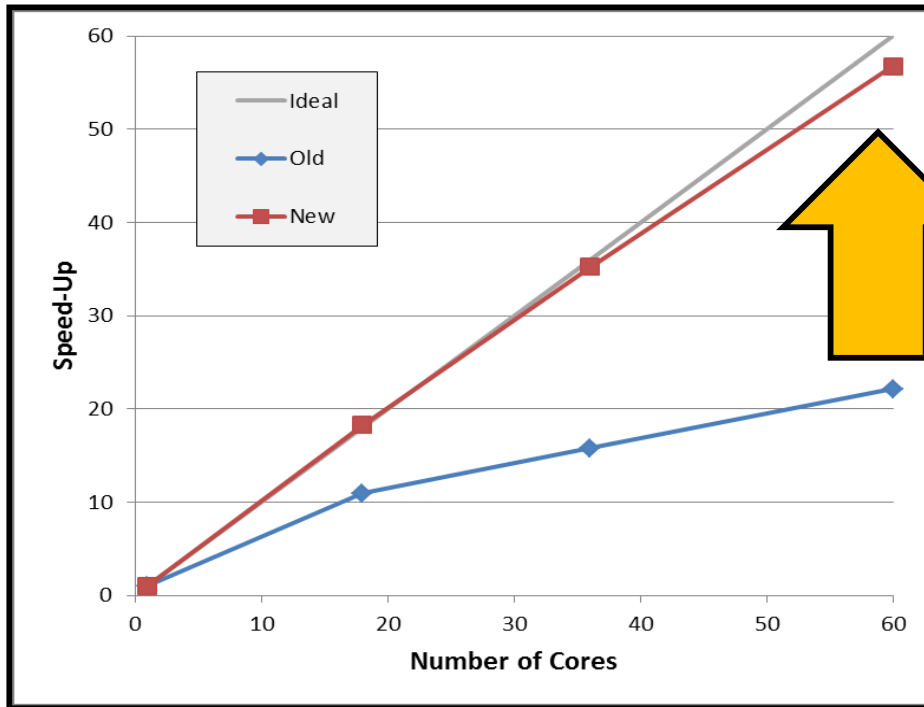


Test case showing reduction in total CPU time when using large numbers of source points (reduction of additional computational cost of source points by as much as 70%)

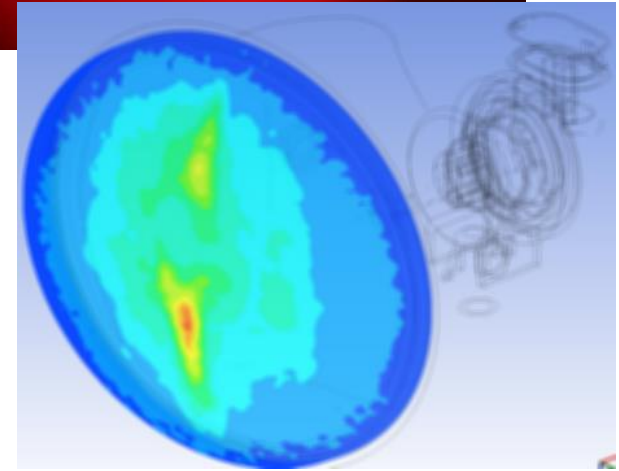
"GaTurbineBlade" by Tomeasy - Own work by uploader; produced with Adobe illustrator. Licensed under CC BY-SA 3.0 via Commons - <https://commons.wikimedia.org/wiki/File:GaTurbineBlade.svg#/media/File:GaTurbineBlade.svg>

# CFX Monte Carlo Radiation Model Scalability

- Significantly improved HPC scalability in speed of Monte Carlo solver



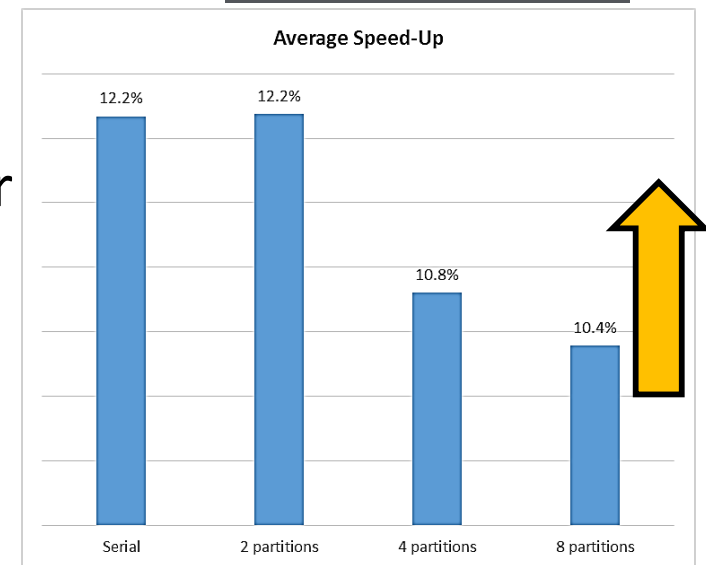
**Dramatically better solver speed scalability for simulations with the Monte Carlo radiation model**



**Complex headlamp test case with 10 million histories  
Comparison when solving only radiation and energy**

# CFX Linux Compiler → Intel Fortran

- Consolidated upon Intel Fortran compiler for both Windows and Linux
  - Consistency between platforms
  - Consistency with other ANSYS products
  - Further possibilities for code optimization to maximize solver performance
- N.B. Requires Intel Fortran for all User Fortran, on all platforms
  - GNU Fortran and PGI Fortran compilers no longer supported



**10% or more average solver performance gain for a set of test cases run on Linux operating system**



# Workbench CFX Solution Caching

- New CFX Solution cell properties:
  - Cache Solution Data: cache CFX solution data for faster re-solves with multiple retained design points
    - Example: run speedline for given design, modify design and re-solve from previous cached DP solutions
  - Keep Latest Solution Data Only: automatically remove older solution data files to minimize disk usage

	A	B
1	Property	Value
2	General	
3	Component ID	Solution
4	Directory Name	TFF
5	Keep Latest Solution Data Only	<input checked="" type="checkbox"/>
6	Cache Solution Data	<input checked="" type="checkbox"/>
7	Initialization Option	Update from cached solution data if possible ▼

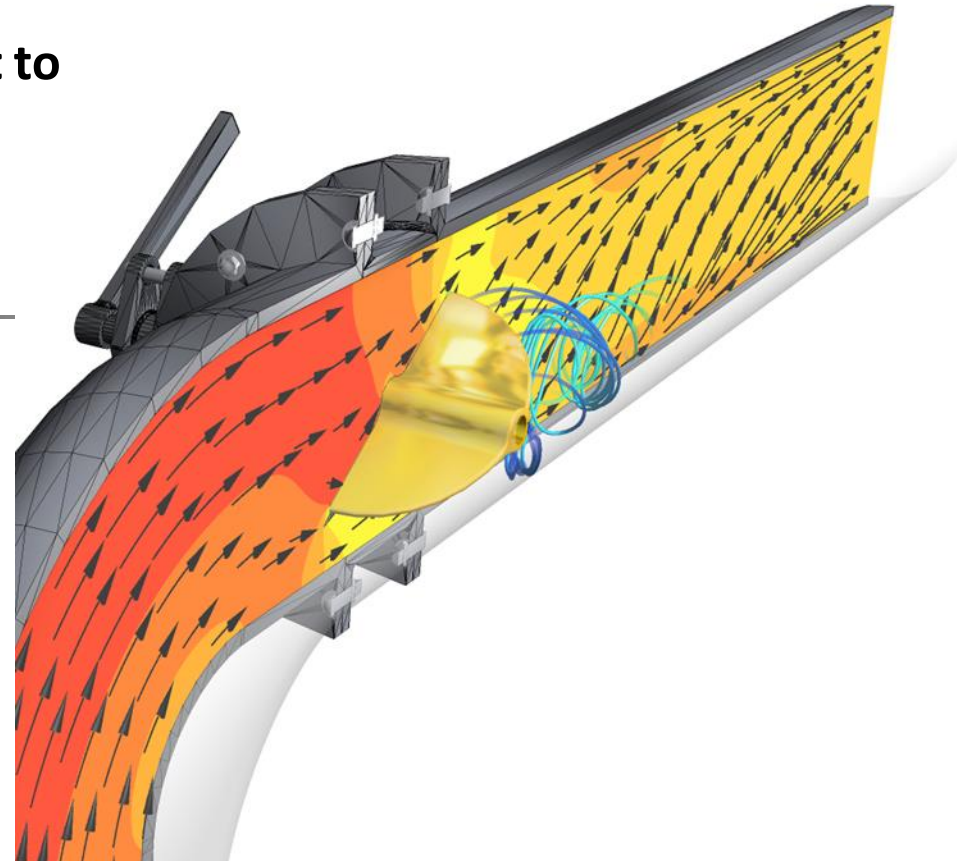
# CFD-Post: Import of STL Geometry to Add Context for CFX

## Problem

Simulation images may be difficult to interpret without key identifying landmarks

## Solution

- Import and export .stl format geometry definitions
- Define post-processing locations
- Import as contextual geometry
- Export any surface locator for other uses - for example surfaces in Fluent



Original CAD geometry surfaces around flow domain brought in as STL to give improved context to simulation results

# Live CFX Solution Monitoring in CFD-Post ( $\beta$ )

Building on introduction of User Surface results output

## CFX Live Solution Monitoring Beta feature of ANSYS Release 17.0

# CFD-Post: Animate Transient Variations of Flow Solutions for CFX

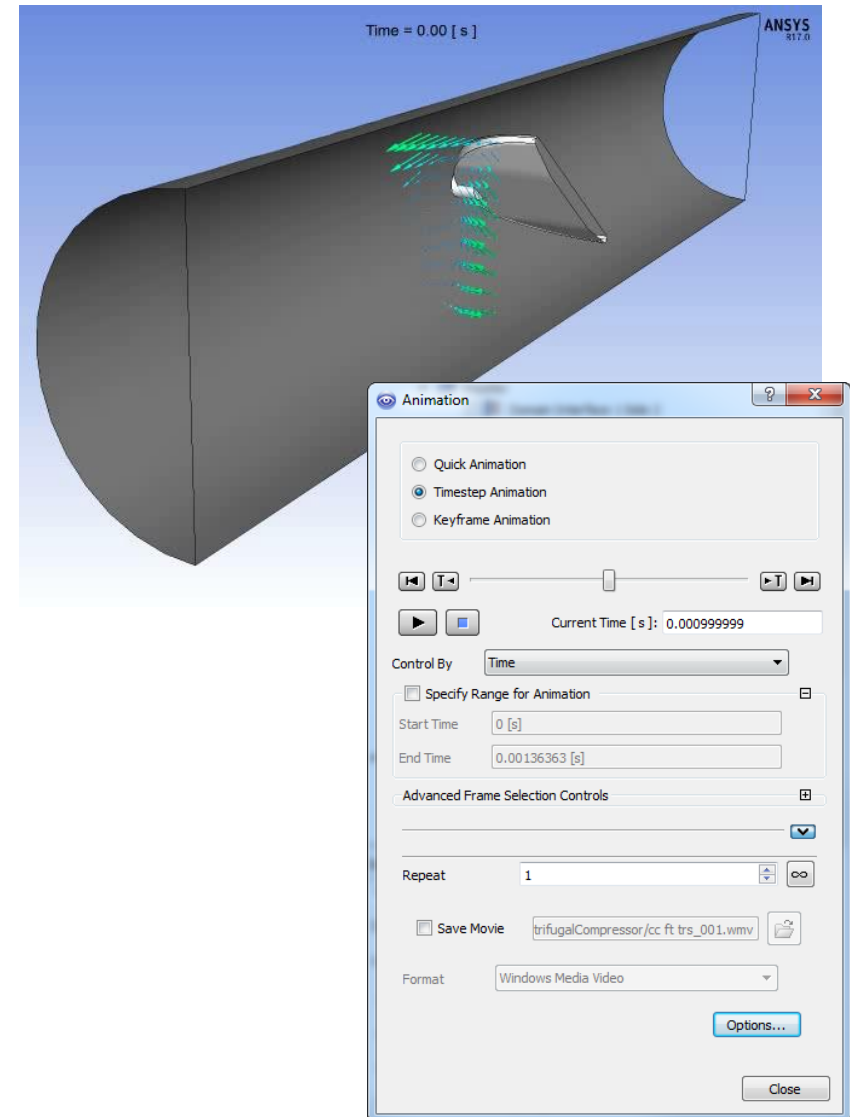
Easy to use “Music player”-like controls

- Play/stop
- Next/previous time step
- First/last time step

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



# More Responsive Data Transfer in CFD-Post

## Problem

Response times slowed by data compression steps between internal components: CFD-Post, Engine, GUI

## Solution

Avoid compression/decompression during data transfer

- More data transferred but compression/decompression cost eliminated
- Up to 40% speed-up





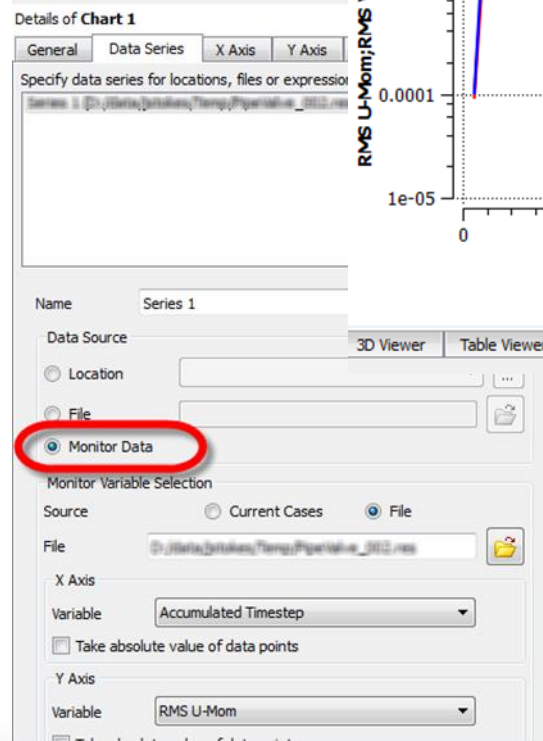
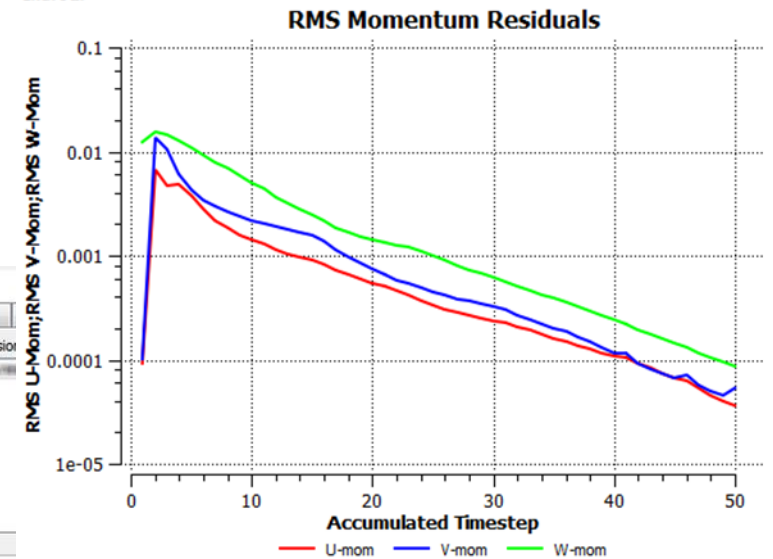
# CFD-Post Charts Available for CFX Monitor Data

Provides access to CFX monitor data for use  
in charts in CFD-Post

Include convergence plots in reports  
generated with CFD-Post

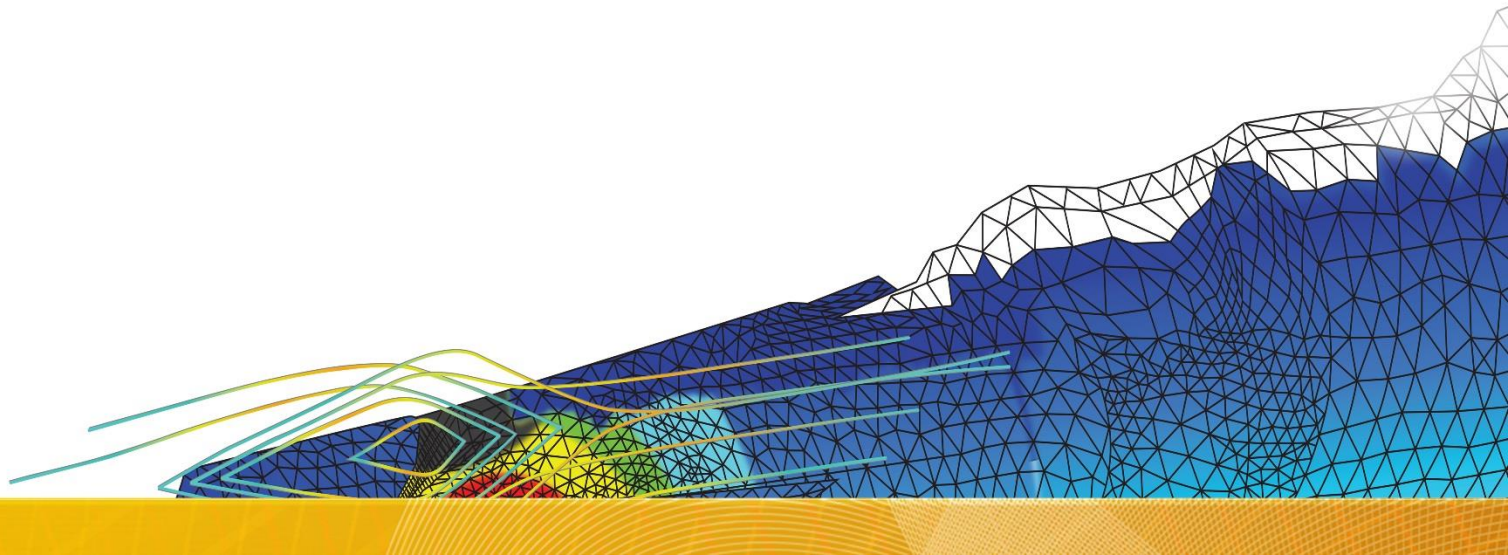
## 4. User Data

Chart 1.



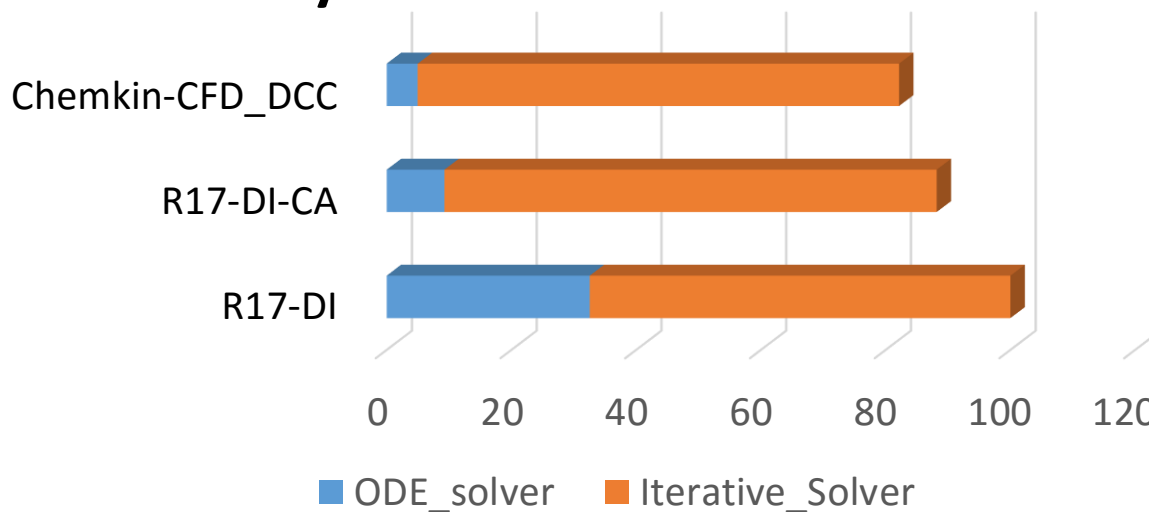


# Model Improvements in ANSYS Fluent



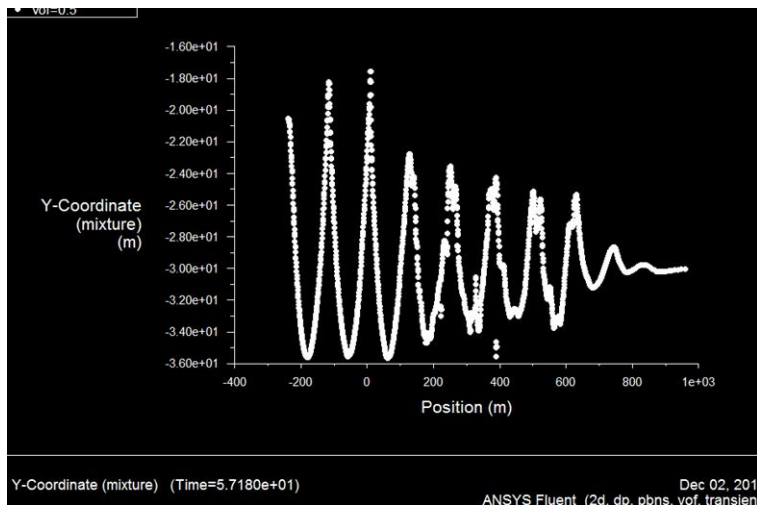
# Reacting Flow Enhancements

- **CHEMKIN-CFD Solver can now be used with no additional license**
- **Full compatibility with CHEMKIN mechanisms**
- **Dynamic Cell Clustering with CHEMKIN-CFD offers significant speed-up for detailed chemistry**

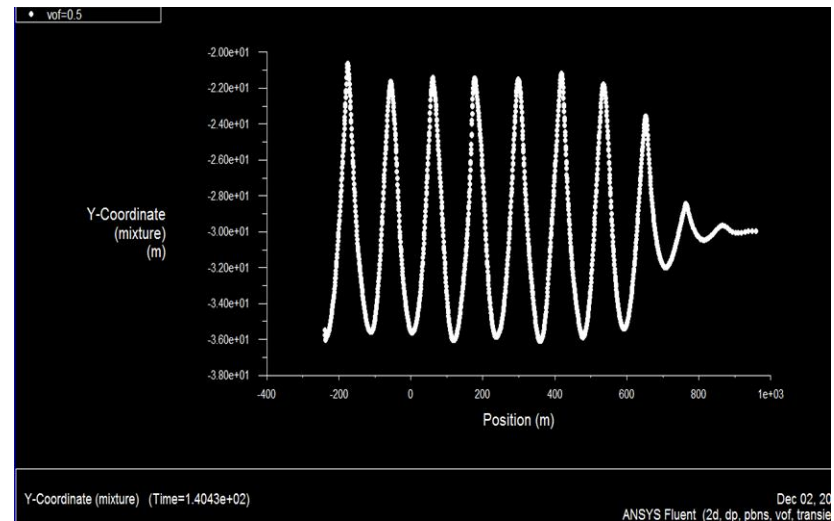


# Offshore/Marine VOF

- Can separate velocity inputs for primary and secondary phases and the moving object for open channel boundary conditions
- Multi-directional numerical beach suppresses numerical reflections arising from multiple pressure-outlet boundary conditions
- Fenton formulation of Stokes Wave Theory for open-channel flows
  - Improved accuracy for steep waves for which earlier implementations would predict premature breaking



Previous wave results



Fenton formulation is closer to experiment

# Fluent Discrete Phase & Discrete Element Method Enhancements

## Rotation of DPM particles for Lagrangian Multiphase

- Gliding friction in particle-wall collisions and lift forces (Magnus lift)
- For particulate flows in cyclones, etc.
- Not compatible with MRF. Rotation not considered in O'Rourke collision model

## Additional DEM collision models

- Rolling friction for bulk solid flows (e.g. fluidized beds and rotating drums)
- Hertzian
- Hertzian-dashpot

## Macroscopic Particle Model UDF is now included at no cost

- Models particles that are large enough to cover multiple mesh cells

**New Rough Wall model gives better predictions in confined geometries such as pipes and cyclones**



# Wall Film Modelling Enhancements

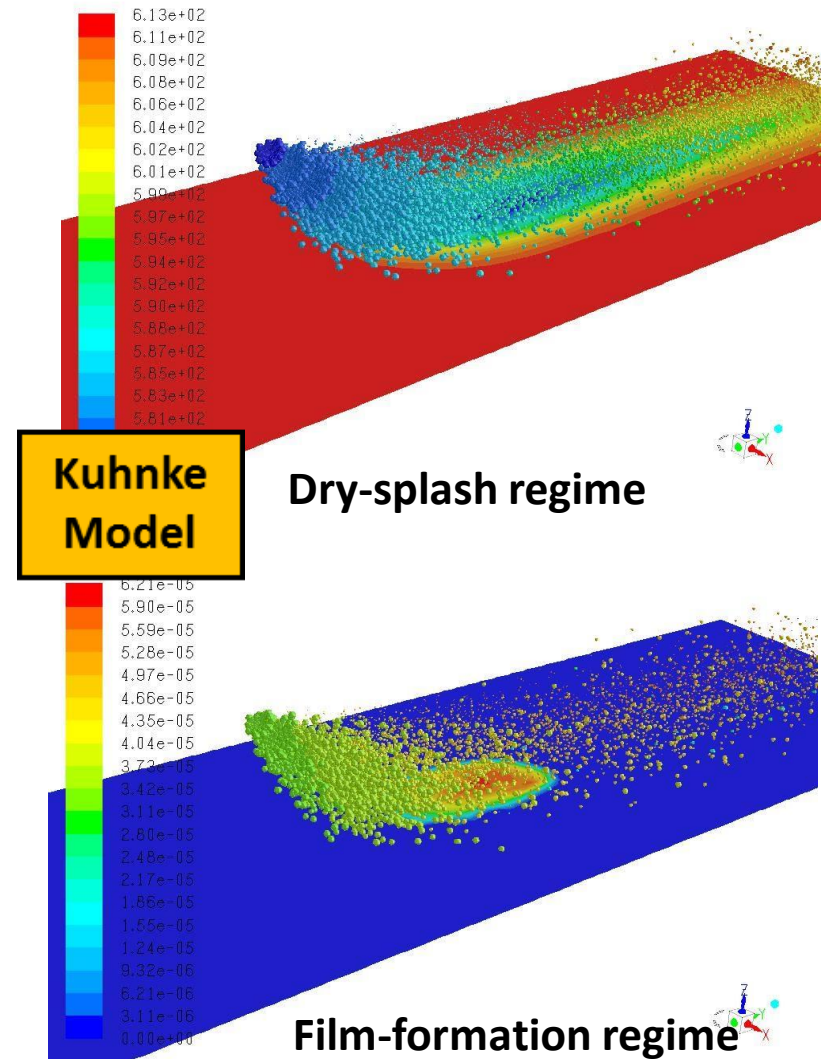
Lagrangian wall-film boundary condition can be modelled using Kuhnke model

4 regimes: spread (or deposition), rebound, splash, and dry splash (thermal breakup)

The deposition and splash regimes lead to formation of a wall film

The Kuhnke spray impingement model is initially validated for selective catalytic reduction (SCR) after-treatment but can be applied to a wide range of applications

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



# Battery Model Enhancements

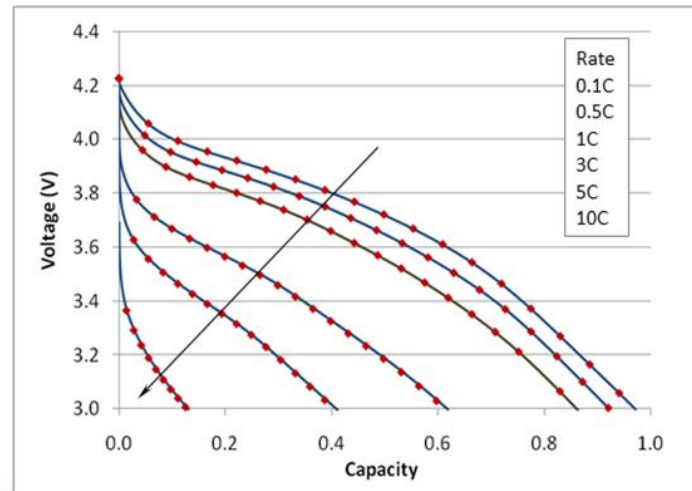
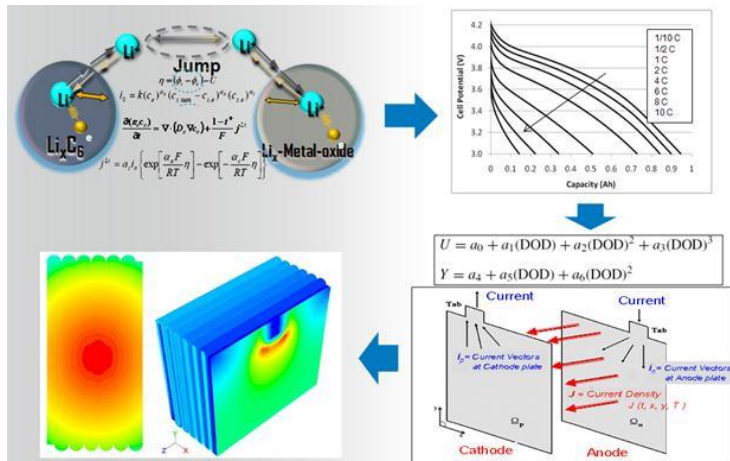
Reduced Order Method for Multi-Scale, Multi-Domain (MSMD) potential equations

Computationally efficient method for use when:

- Electric conductivity does not depend on temperature
- Transfer current density is uniform over a battery's active zone

One-equation and Four-equation thermal abuse models

- Can be used to simulate thermal runaway processes

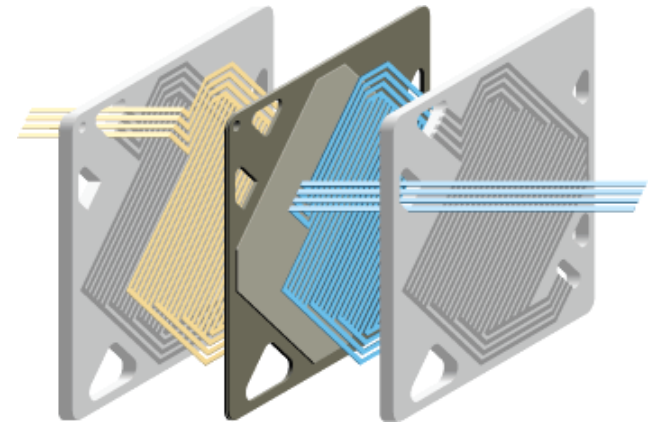


Full model  
(solid) v  
ROM  
(dotted)

# Fuel Cells/Electrochemistry

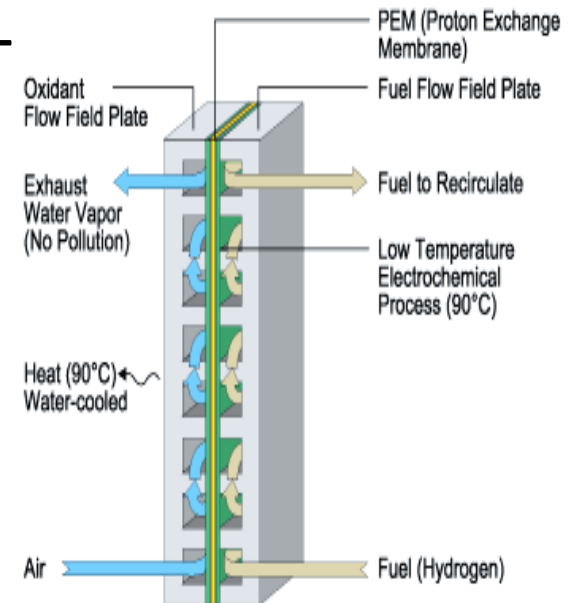
## New PEMFC Model

- Distinct anode and cathode micro-porous layer properties
- Solves the capillary pressure equation in porous media
- 3-phase water transport modeling (gas, liquid, dissolved)
- Dissolved water transport through catalyst-membrane-catalyst assembly
- Liquid saturation in gas channels enables modelling pressure drop increase due to presence of liquid water



## Electropotential Model

- Use in isolation or with electrochemistry (now fully supported)
- Applications: flux batteries, electroplating, corrosion
- Note: Electrochemical reactions occur only on surfaces



# Continuous Fiber Model Enhancements

## Support for Parallel Processing

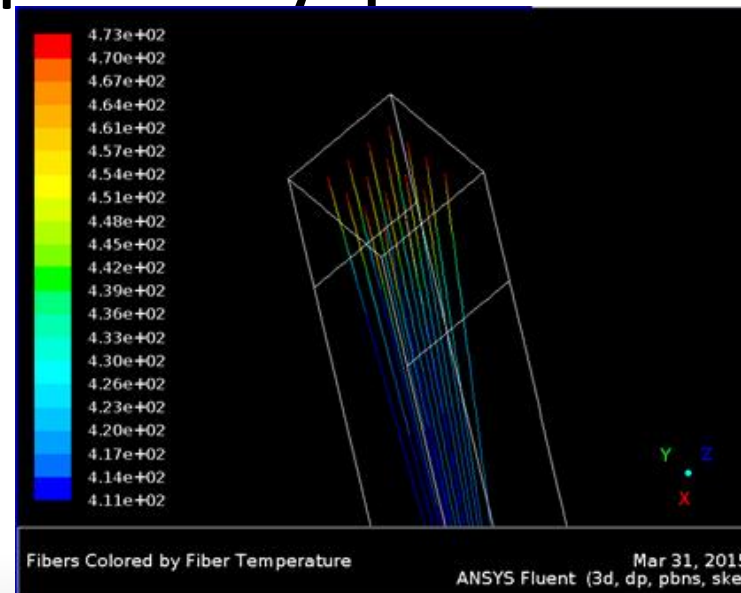
File injections allow file based specification of fiber points

Extended UDF access allows better customization of fiber properties

Rheology model is now able to deal with generalized Newtonian flows for melt-spun and dry-spun fibers

## Postprocessing

- Fiber color by variable
- Fiber display with mesh



# Scale-Resolving Turbulence Modelling

## Fluent and CFX

### Problem

User want improved accuracy in cases that have been challenging for the existing turbulence models

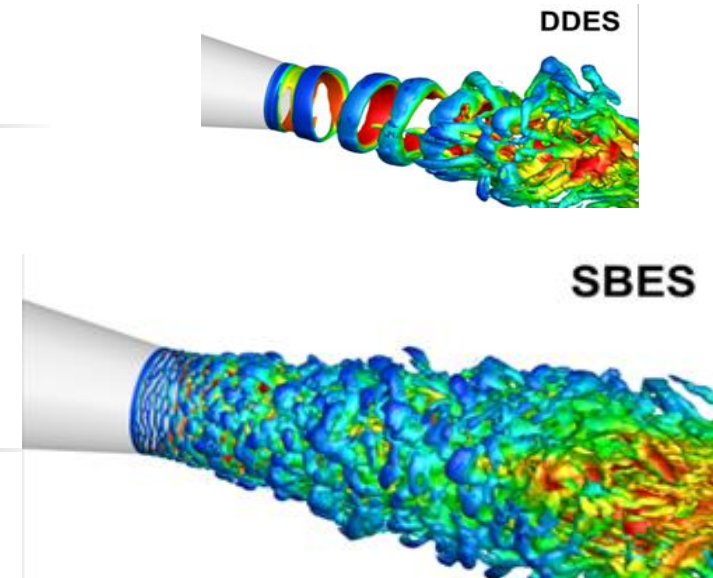
### Solution

#### Shielded Detached Eddy Simulation (SDES)

- Stronger shielding than Delayed DES
- Clear separation of RANS and LES mode
- New option for existing DES SST

#### Stress-Blended Eddy Simulation (SBES)

- Incorporates and builds on SDES
- Faster switch to LES modes
- Combines RANS and LES, with blending (f)



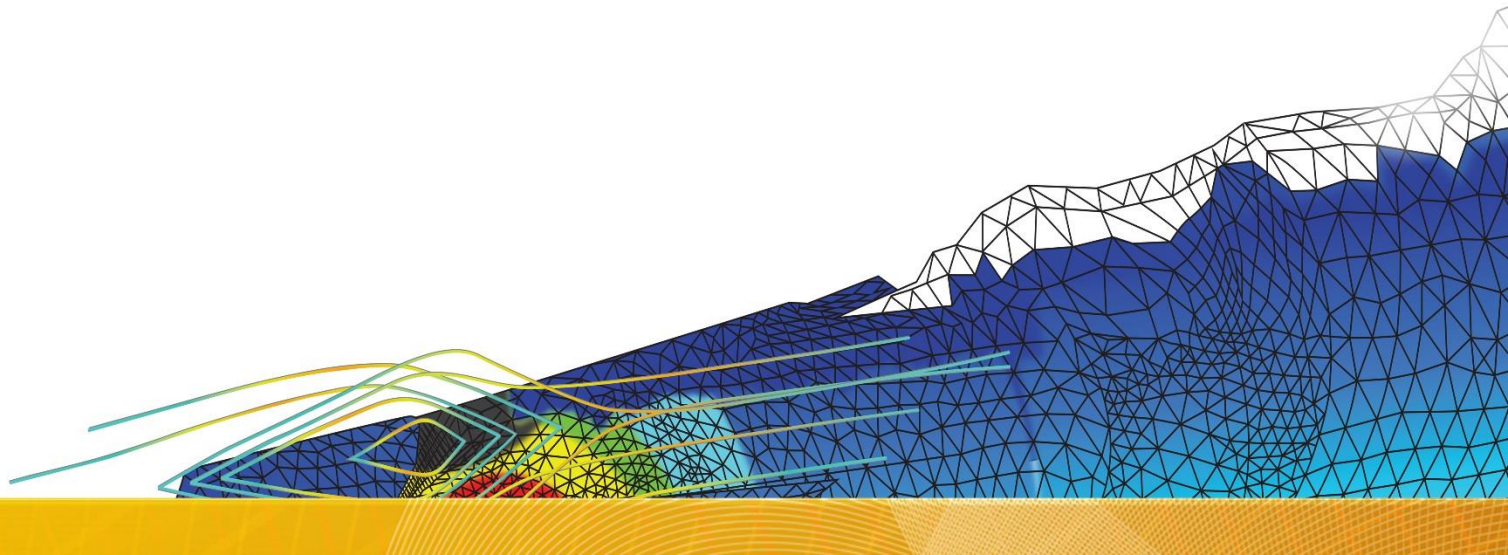
#### Stress-BSL Model

- Linear model for pressure-strain like stress- $\omega$ , but solves scale equation from BSL
- Removes free-stream sensitivity of stress- $\omega$





# Model Improvements in ANSYS CFX



# Intermittency Transition Model for Turbulence

## Problem

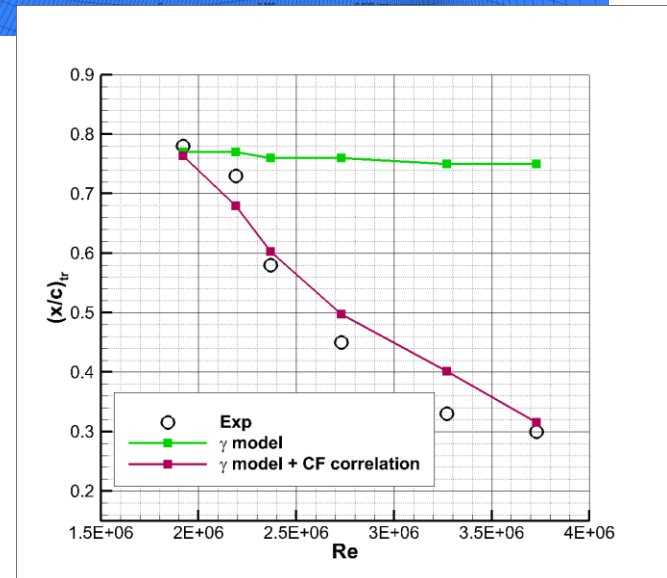
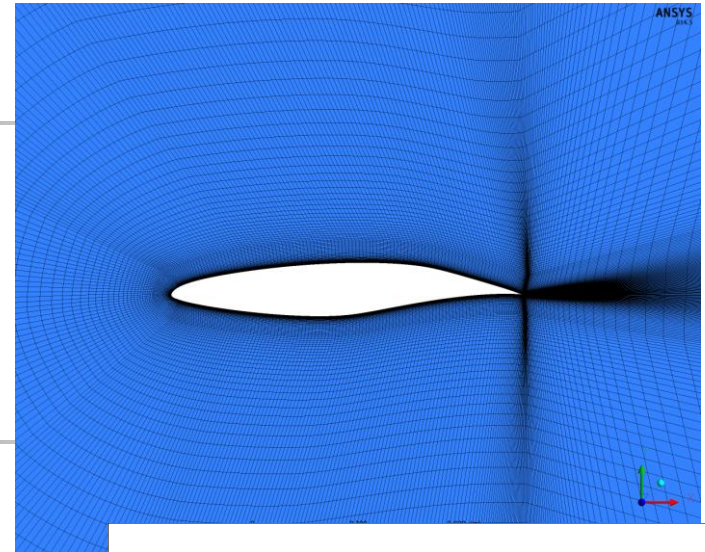
Turbulence transition models require multiple equations and are computationally intensive

## Solution

New one-equation intermittency-based transition model is advance on  $\gamma$ - $Re\theta$  model

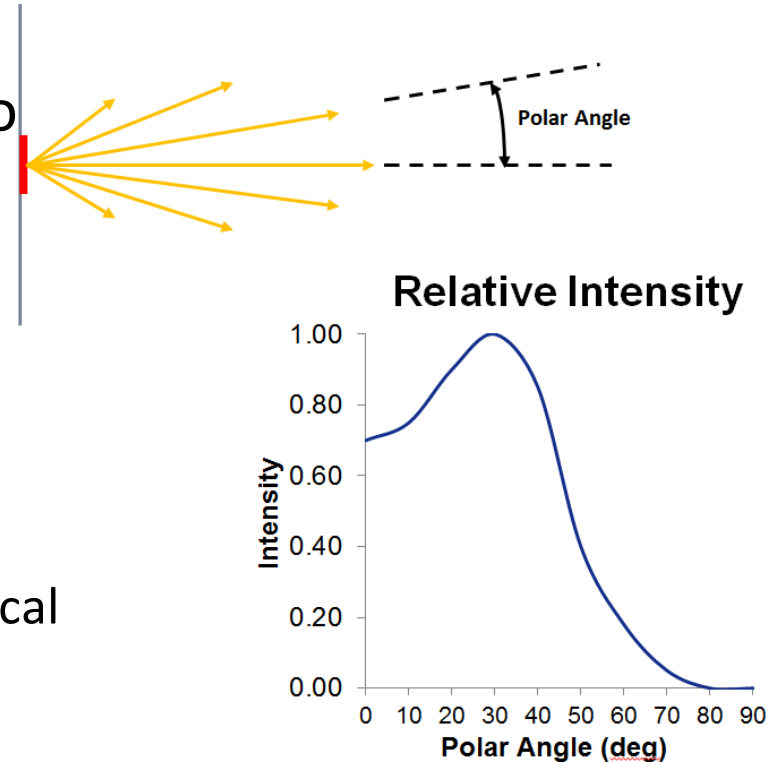
## Advantages

- Reduce number of equations
- Galilean invariant
- Simplified correlations
- Crossflow instability



# CFX Radiation: Additional Boundary Sources

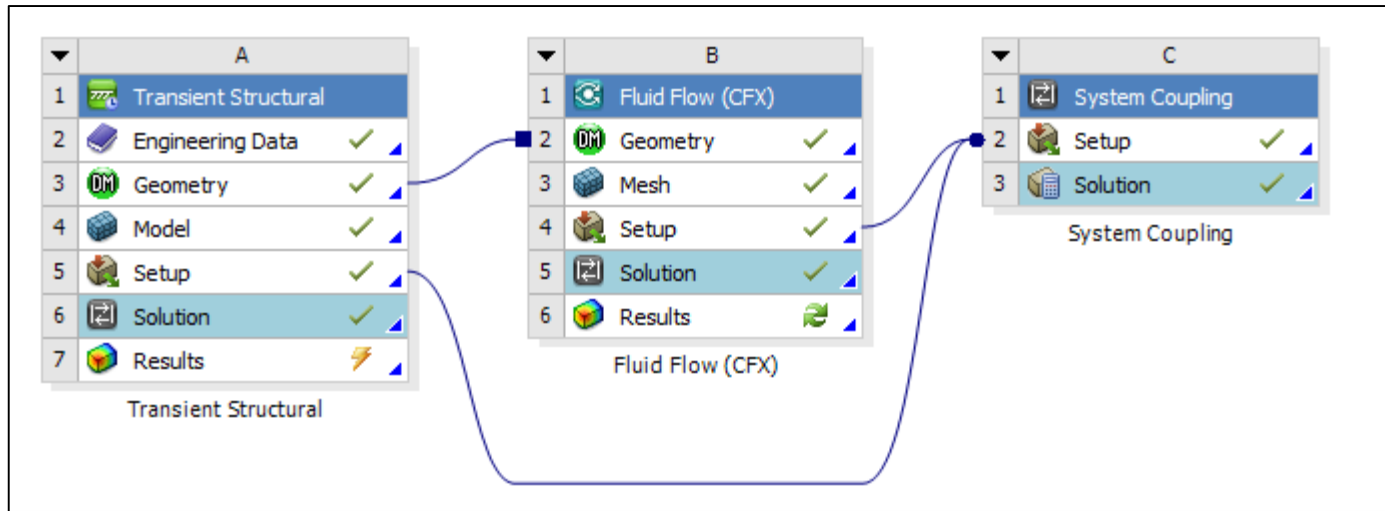
- Extended Directional Radiation Boundary Sources for Monte Carlo Radiation Model with new Polar Distribution Function ( $\beta$ )
- Model inherent scattering, or characteristic divergence, of many radiation sources
- Creates 3-dimensional distributions of intensity as spherically symmetric in local coordinate system
- For use with Monte-Carlo Radiation
- Currently available as hidden CCL



# 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  $\rightarrow 0$

# CFX – System Coupling Connection

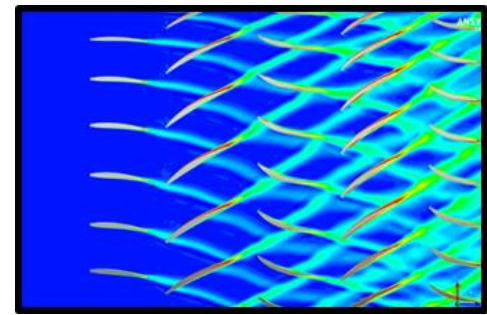


- **In R17.0: Force – Displacement coupling for CFX and Mechanical via System Coupling**
- **In R17.1: Thermal coupling added**
- **Replaces MFX workflow**
  - Supports Mechanical features that were missing with MFX (contacts, remote loads/constraints, springs, etc)
  - Improved restarts
  - Consistent FSI workflow for our CFD tools

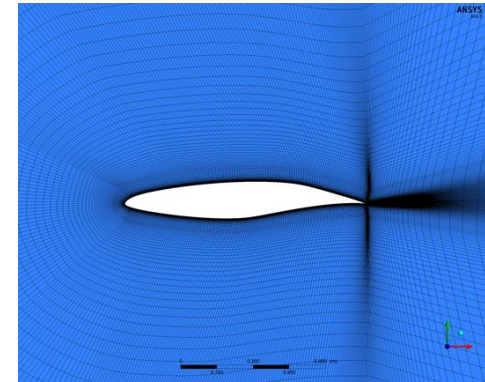


# Turbomachinery Modelling

- Faster multi-stage transient blade row problems by calculating as few as one blade per row
- Breakthrough one-equation turbulence transition model
- Fourier Transformation for multi-disturbance and multiple-frequency asymmetric flows
- Forced Response Analysis for multi-stage and enhanced numerics for more accurate blade flutter
- Harmonic transformation method under development



Single passage per row TT/STT model



Turbulence Transition Modeling

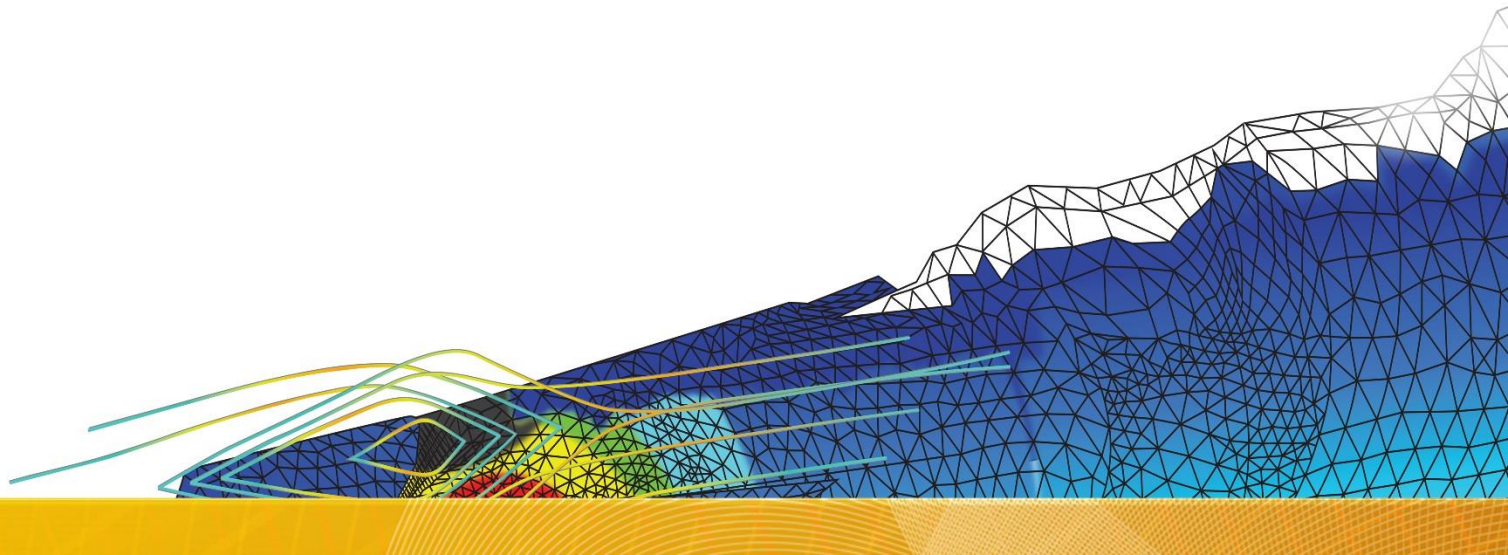


Fourier Transformation  
of large multi-frequency  
disturbance





# ANSYS 17.2 and Beyond



# Fluent 17.2

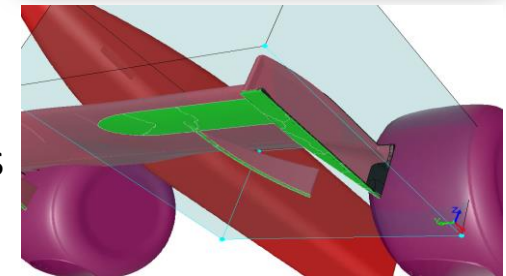
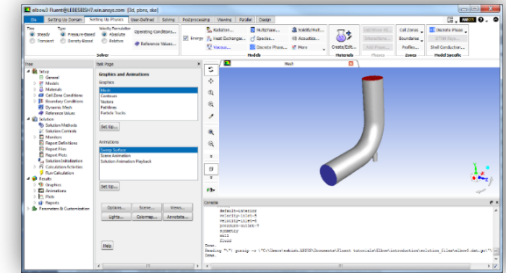
- **Overset improvements :**
  - Metis becomes the default auto-partitioning method
  - Improved gradient computation and hybrid initialization available
- **Export of surface data to Radtherm in parallel**
- **GUI/TUI for time-invariant 6DOF properties, 1-DOF with linear spring and motion stops without UDF**
- **ACT as a full feature (now Beta)**
- **UI: Scroll wheel zoom**
- **User defined hook for setting time scales in “Relax to Equilibrium Model”**

# Fluent R17.2 – Under the hood improvements

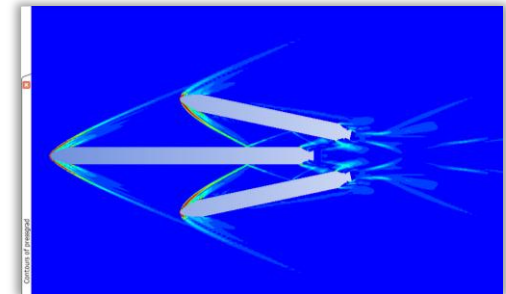
- **AIM to Fluent mesh transfer**
- **Presto limiting as hidden feature controlled by rpvar**
- **Exposing AFD API in 17.2**
- **Remove Visual Studio 2005 dependencies (VKI/ODB Library Upgrade)**
- **PEMFC: enhancements/defect requests**

# Future Development Objectives - Fluent

- **Workflow, Automation, Robustness, Performance**
  - Easier problem setup with reliable convergence
  - 2-way workflow for meshing & solving
- **Graphics & Post-processing**
  - User interface improvements across the board
  - Improved Transient post-processing workflow & capabilities
  - Interactive solution monitoring
- **HPC & Large Models**
  - Improve performance CPU & GPU
  - Graphics/GUI performance with large models
- **Physics and Modelling Enhancements**
  - Extend R17.0 Overset capabilities
  - Adjoint optimization with constraints



Geometry constraint: final design must stay above plane  $z = 0.066$



# CFX R17.2 – New released features

- **CFX thermal system coupling with wall heat flow**
- **Blade row efficiency monitoring**
- **Flank milled blades**
- **Added support in TurboGrid to release complete Flank Milled Blade capability**
- **Turbo performance maps in Workbench**
- **BladeEditor: Improved error reporting for the FlowPath and Blade features**

# CFX R17.2 – New beta features

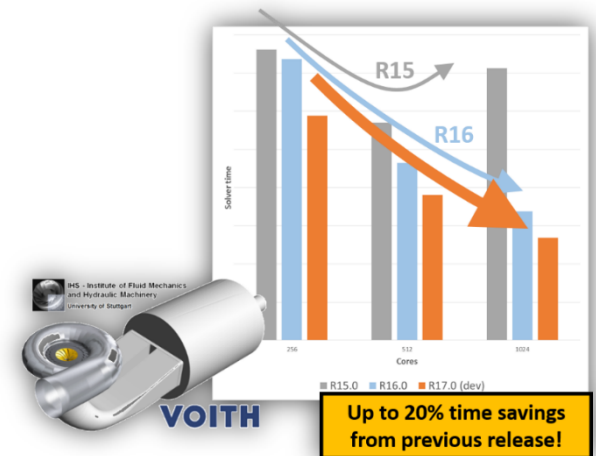
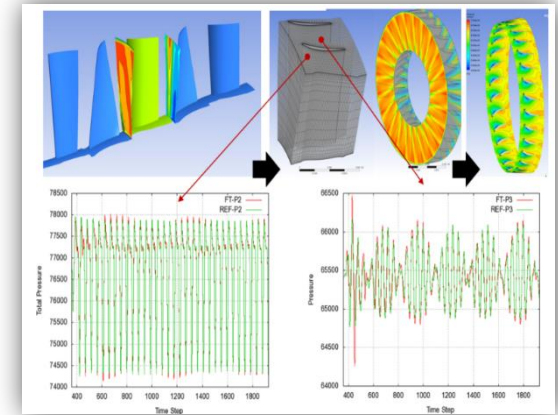
- **Blade Flutter using the Harmonic Transformation method**
- **Fourier Transformation - Blade flutter for radial machines**
- **CFD-Post blade flutter post-processing**
- **CFX solution monitoring in CFD-Post**
  - **Evolving Transient Solutions**
  - **Steady State Solutions**
- **Improved solver performance with large number of face sets**
- **CFX multiphase flow improvements**
  - **Ability to switch off boiling at or near boundaries to improve robustness**
  - **Improved controls at dilute fluid limit when modelling three or more phases**



# Future Development Objectives - CFX

Development emphasis for CFX will be turbo and specific needs of customers:

- **Continued turbomachinery development**
  - Transient blade row methods – Harmonic Analysis
  - Blade row meshing (TurboGrid)
- **Improving solver HPC performance and I/O efficiency**
  - Ongoing enhancements for industrial cases
- **Specific enhancements**
  - Multiphase
  - Heat transfer and radiation
  - Usability
  - Post-processing
  - ...



# ANSYS DesignModeler Update

- Following the acquisition of SpaceClaim Inc, ANSYS are focusing their geometry capabilities and developments on SpaceClaim
  - Please check out the SpaceClaim sessions at 16.00 (Track 4)
  - Please go and try it in the Q&A/workshop room (also Track 4)
  - Please speak to your account manager about the migration options
- Nonetheless ANSYS DesignModeler will continue to be available for customers who want to use it
  - We have many customers with processes built around DesignModeler
  - SpaceClaim is still being enhanced to fill the gaps done better by DesignModeler
- Customer support for ANSYS DesignModeler will continue to be offered as there is on-going need
- At R17.x, some minor improvements were made to DesignModeler...

# Summary

- **Commitment to future development of both CFX and Fluent as a flagship ANSYS products**
- **Major advances in parallel performance on any number of cores**
- **Usability and post-processing improvements including new ribbon-based GUI and reacting flows in Fluent and live solution monitoring in CFX**
- **Advanced physics developments including reacting flow, adjoint solver, hybrid RANS-LES and transition turbulence model, multiphase porous media, wall boiling, etc.**
- **R17.2 due for release in Sept.**
- **ANSYS promoting move to ANSYS CFD – see next presentation**

# ANSYS CFD

- **ANSYS CFD**
  - Over five years ago, we launched a licence called ANSYS CFD
    - Giving access to Fluent and full-physics CFX
  - At release 17.0, two more products were added into the licence
    - Polyflow – thermoforming/blow moulding, extrusion CFD simulation with couplings to our Structures FEA
    - Forte – In-cylinder combustion
  - Customers with ANSYS CFD R17 compatible licence key installed can access these immediately
  - For those with ANSYS CFX, ANSYS Fluent keys, a small upgrade will access you these additional products (speak to your account manager, the costs are small)
- **ANSYS Mechanical CFD, ANSYS Mechanical CFD Maxwell** customers can access these new CFD tools as well as all the tools included with the Mechanical Enterprise re-bundling