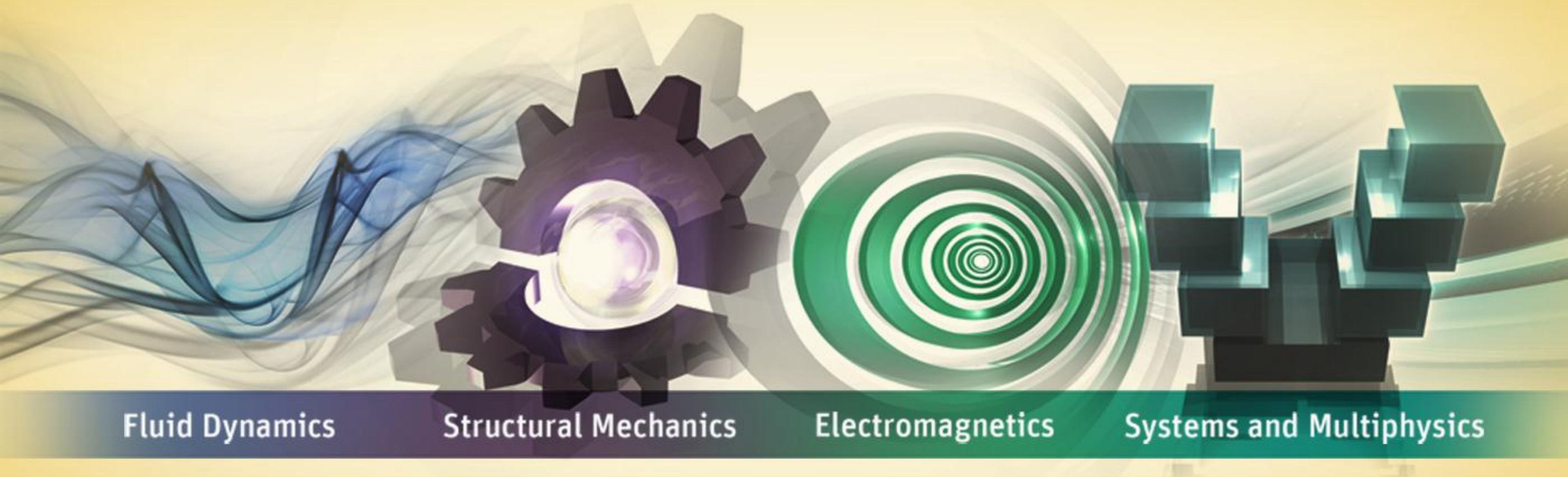
ANSYS

Realize Your Product Promise™

ANSYS Release 15
Fluids Update



Important Note

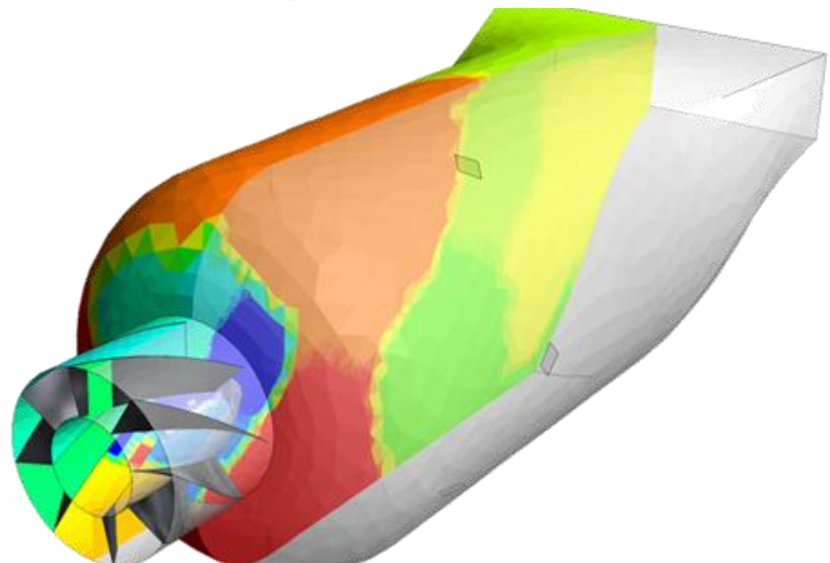
- **This presentation is an overview of the major improvements in ANSYS Fluid Dynamics at R15.0**
- **This presentation does not include**
  - Geometry, Meshing, Systems Coupling, ANSYS Icepak, Polyflow
- **Please go to the ANSYS Customer Portal for more information**
  - Documentation of new capabilities
  - Release notes
  - Migration manual

ANSYS Fluids – Next Release Highlights

- High Performance Computing
- Advanced Solver Technology
- Comprehensive Physics Modeling
- Turbomachinery
- Enhanced Usability

High Performance Computing

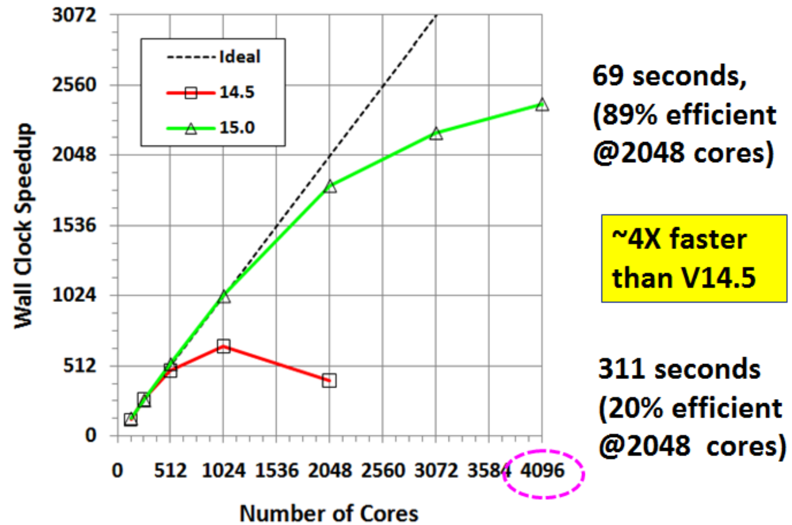
ANSYS works continuously to optimize parallel scalability and performance of our CFD solvers, allowing customers to leverage the latest hardware advancements and arrive at an accurate solution fast.



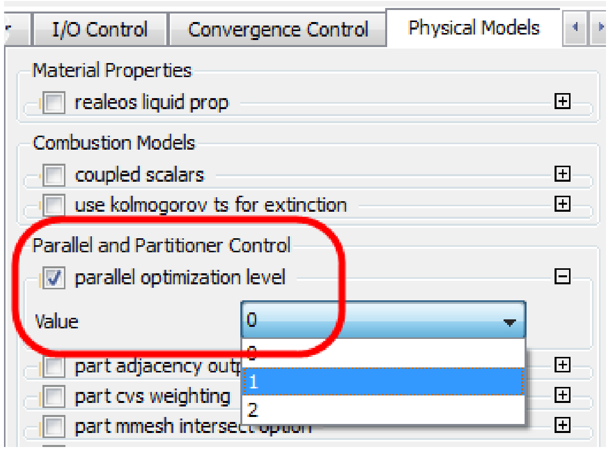
Parallel Scalability

(CFX)

- **R&D project to improve HPC**
  - Investigation of various solver parallel scalability limitations
  - Industrial benchmarks
    - Single and multi-domain (incl. two-stage radial compressor and six-stage axial compressor)
    - Steady and transient
  - Implemented improvements accessible via expert parameter
    - Default setting does not incorporate changes



Solver wall clock speed-up on 150M node intake case

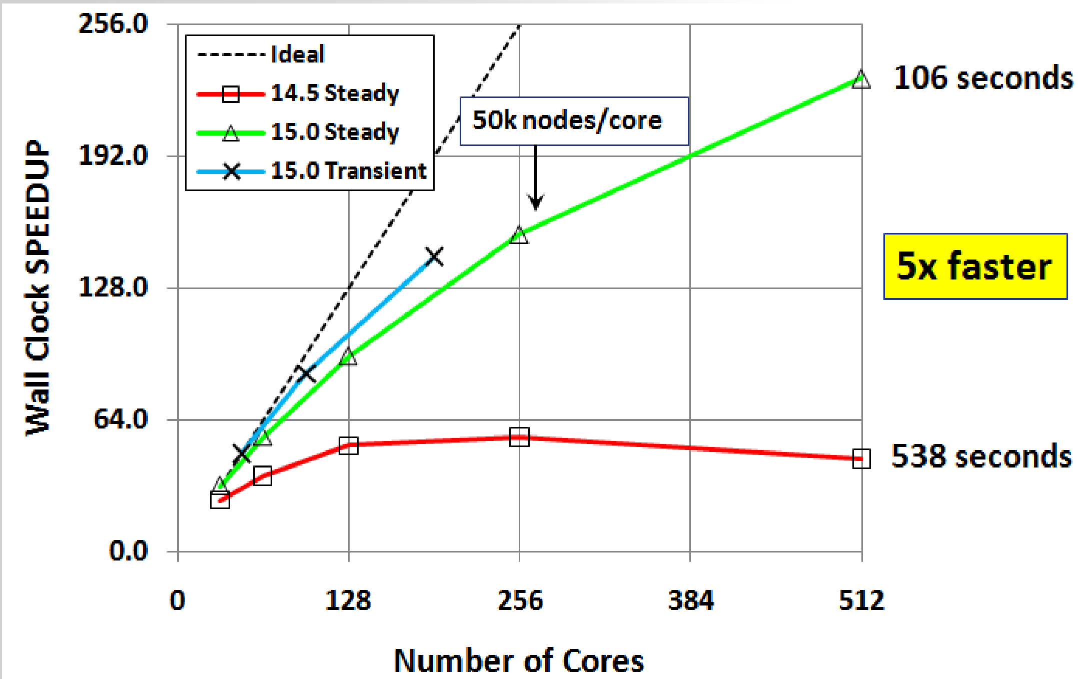


ANSYS

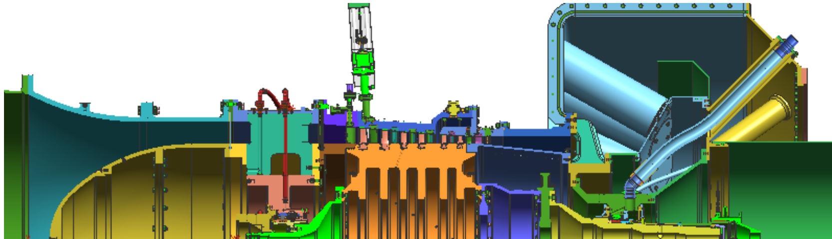
(CFX)

Parallel Scalability

- **Industrial benchmark application**
  - 6-Stage Axial Compressor
  - 13m nodes, 14 Domains, 12 Mixing Planes



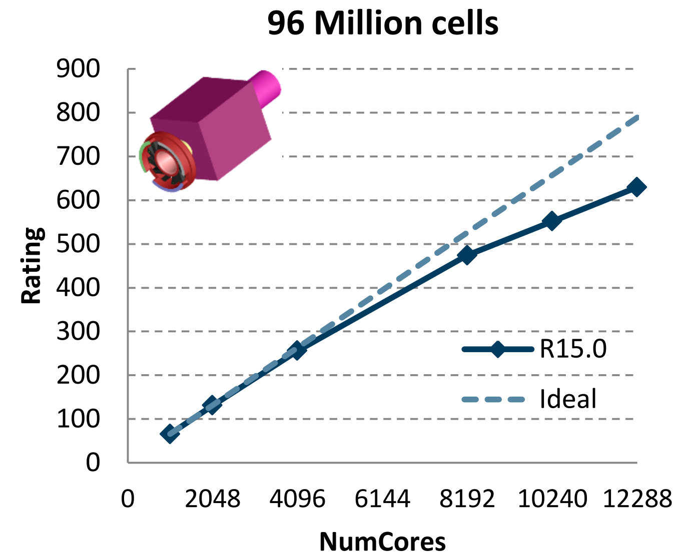
Courtesy Siemens AG, Mülheim, Germany, ASME IGTI Paper GT2013-94639

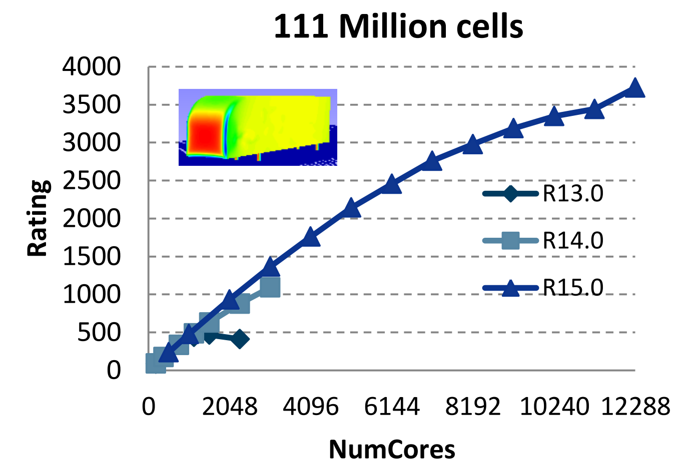


Parallel Scalability

(Fluent)

- **High solver scalability at large core counts**
  - ~84% efficiency for 96M cell case at 10240 cores
    - Coupled solver, LES, and species transport
  - Similar trend for 111M cell standard benchmark
    - Segregated solver

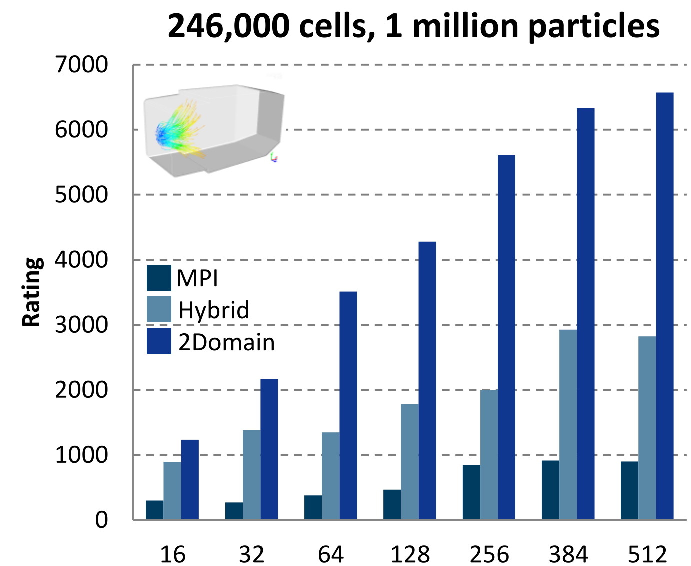


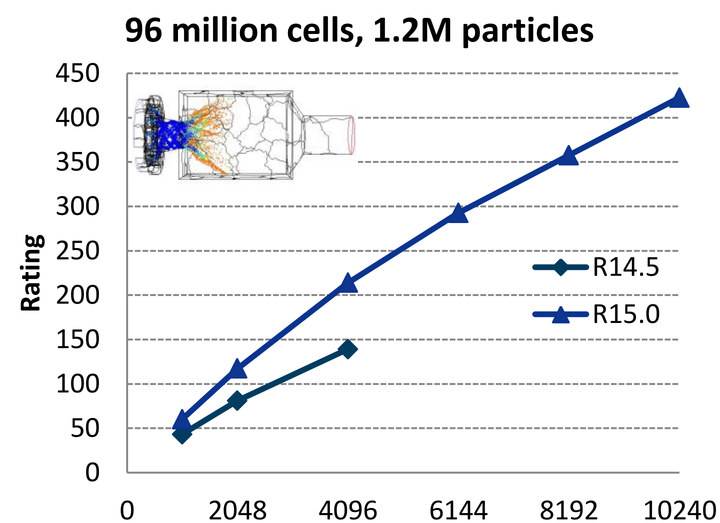


Parallel Scalability for Discrete Phase

(Fluent)

- **New 2-Domain method**
  - Balance the continuous and discrete phase independently
  - Over 2x improvement seen for 512-way parallel
- **Improved scalability for hybrid method**
  - Default method for parallel particle tracking



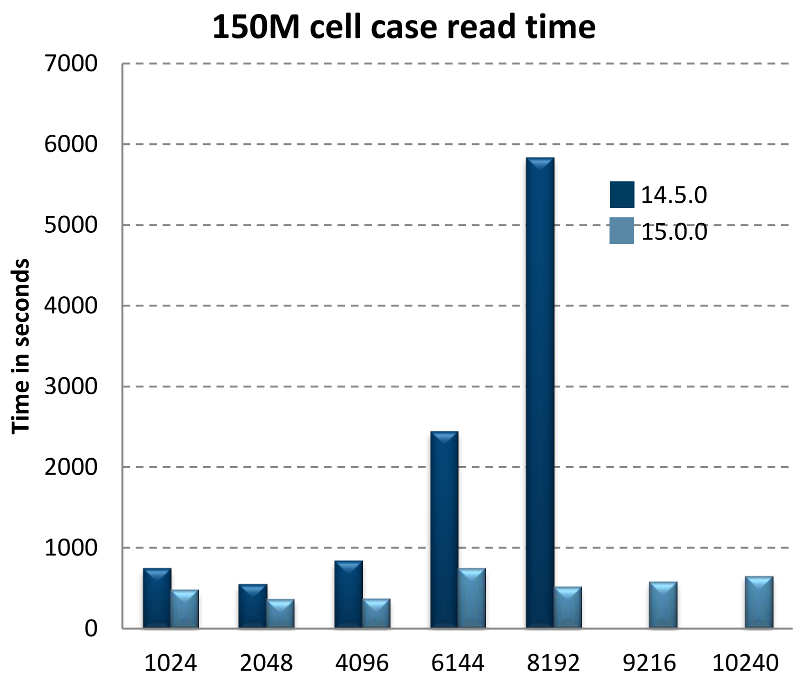


ANSYS

Parallel File I/O and Startup

(Fluent)

- **More efficient parallel I/O and startup**
  - Case read time reduced significantly at high core counts
  - Start-up time for 8192-way parallel reduced from 30 minutes to 30 seconds
- **Effective configuration of parallel processes**
  - Use different number of processes for meshing and solve modes



Other Parallel Enhancements

(Fluent)

- **Improved parallel error handling**
  - Ability to restore running simulations to a usable state after a crash
- **Faster solutions using GPUs**
  - Accelerated AMG solver performance for 3D coupled pressure-based solver cases
- **Support for Intel Many-Integrated-Core (MIC) (β)**
  - Intel Xeon Phi

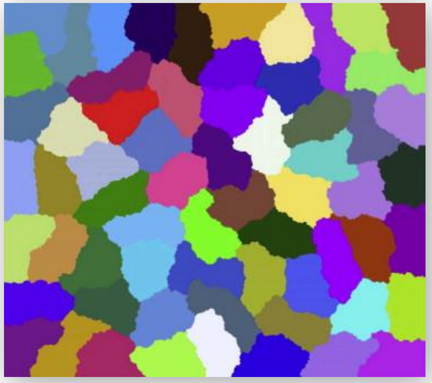
Internal Flow Steady Computation							
		Fluent		NVAMG			
		AMG	Total	AMG		Total	
Serial	SP	352	558	74	4.76x	289	1.93x
	DP	485	777	97	5.00x	417	1.86x
T12	SP	71	100	86	0.83x	114	0.88x
	DP	132	179	119	1.11x	166	1.08x

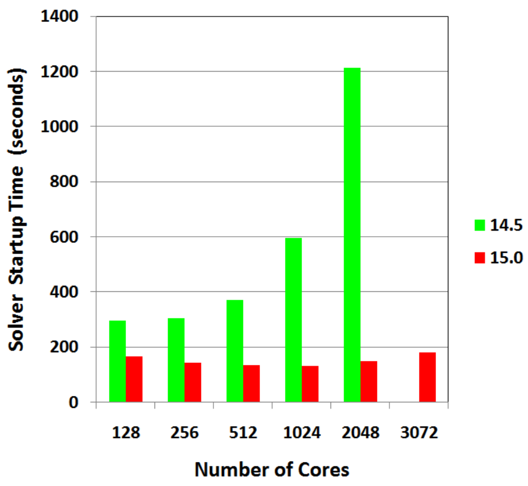
Internal Flow Unsteady Computation							
		Fluent		NVAMG			
		AMG	Total	AMG		Total	
Serial	SP	1983	3168	394	5.03x	1582	2.00x
	DP	2832	4593	517	5.48x	2263	2.03x
T12	SP	496	653	454	1.09x	611	1.07x
	DP	933	1221	630	1.48x	920	1.33x

Other Parallel Enhancements

(CFX)

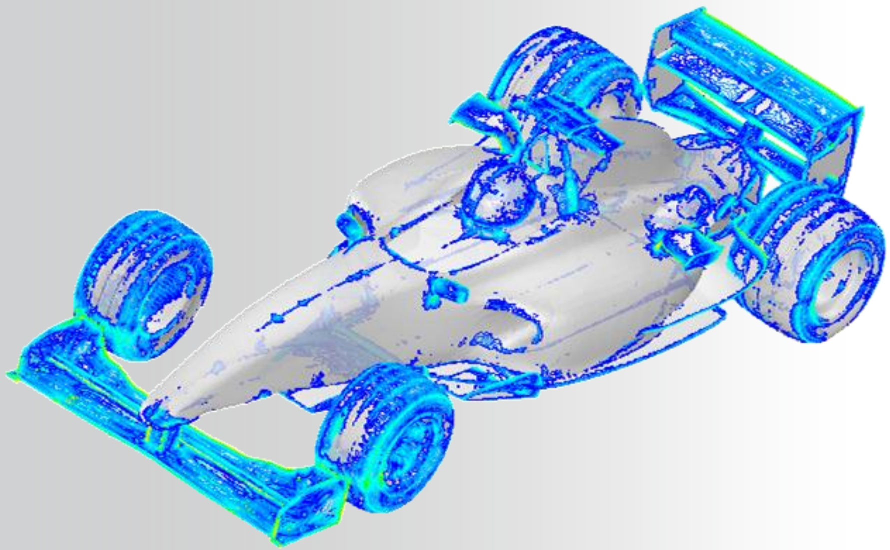
- INT64 version of MeTiS → improved partition quality on large meshes (β)
- More efficient HPC solver startup
- Improved parallel diagnostic output format
- Support for Intel MPI
- MPICH2 version for Cray XE (β)

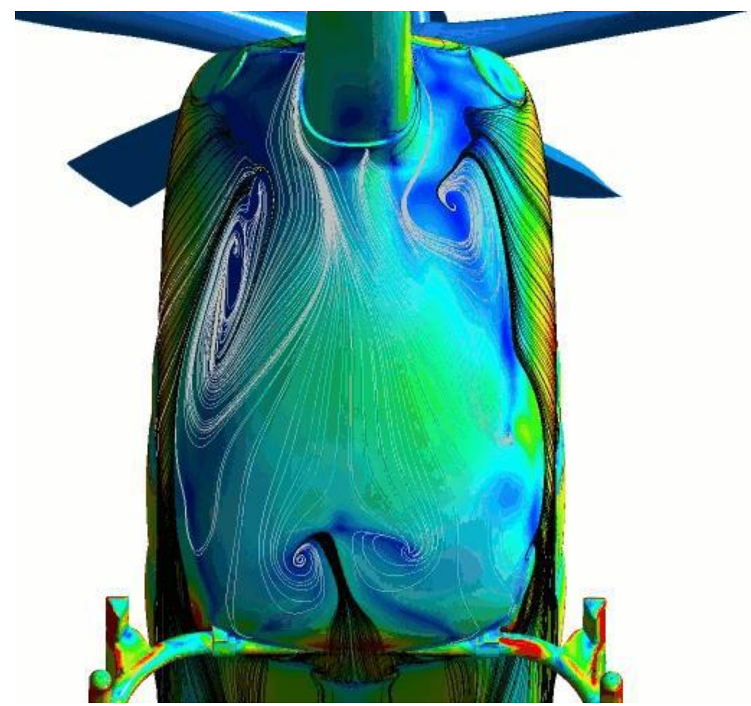




  January 28, 2014

Advanced Solver Technology



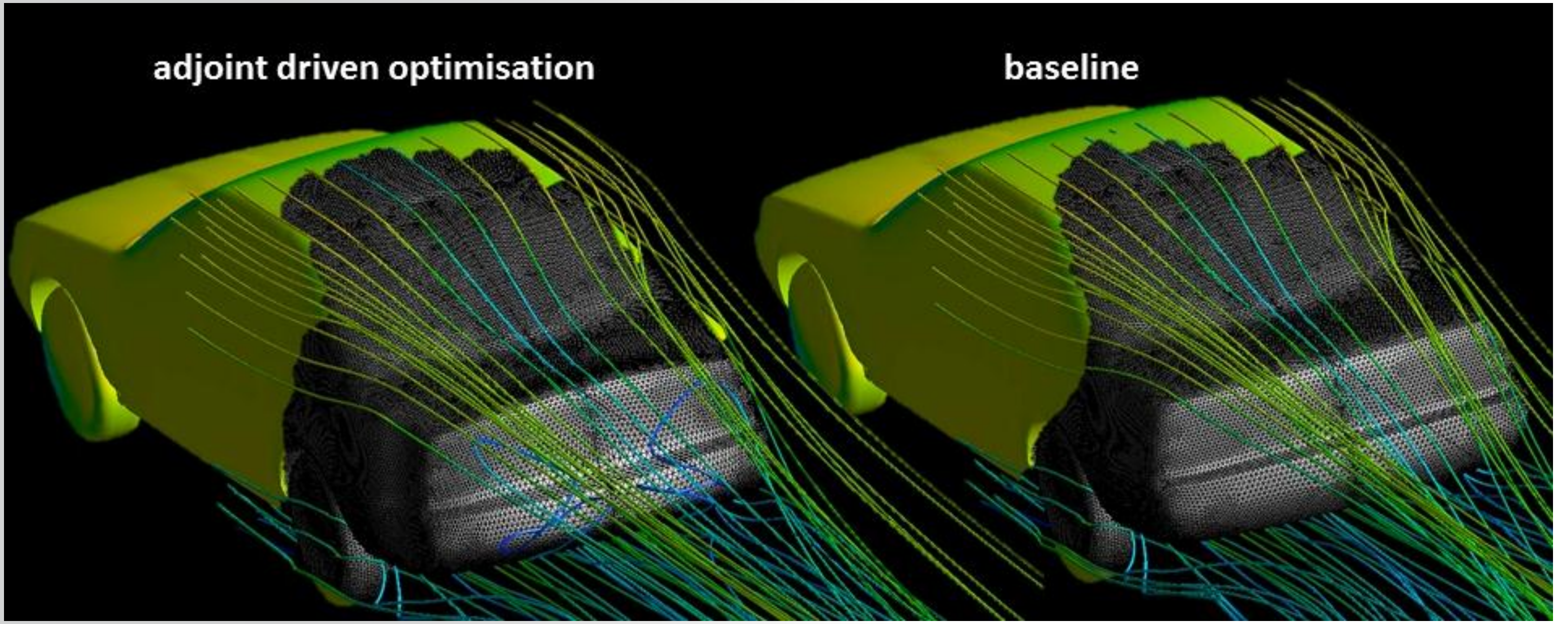


Robust, accurate, and advanced CFD solver technology from ANSYS provides the basis for fast assessment and optimization of designs and their performance.

Adjoint Solver

(Fluent)

- **Expanded adjoint solver capabilities**
  - Support for larger scale problems
    - Up to 30 millions cells
  - Ability to solve the adjoint equation for energy
    - Observables as integrals of heat flux and temperature



Mesh Morpher Optimizer

(Fluent)

- **Improved control point selection**
  - Select multiple control points with RMB click
  - Select control point based on I,J,K coordinates
- **Significant productivity gains (β)**
  - New NEWUOA algorithm requires fewer design iterations

	Old	NEWUOA
airfoil	440	97
airfoil-mult-DR	177	127
bend	55	45
helix	31	20
nozzles	35	34
ring	1744	920
sedan	90	93

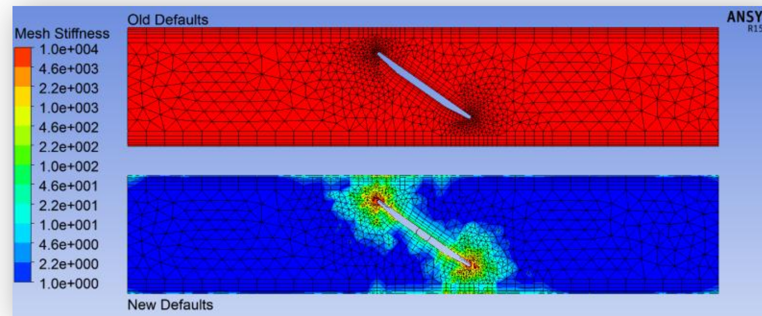
Final objective function values:

	Old	NEWUOA
airfoil	13.4	12.3
airfoil-mult-DR	5.7e-7	1.8e-12
bend	0.459	0.459
helix	0.500	0.500
nozzles	0.333	0.333
ring	0.341	0.360
sedan	284.8	284.9

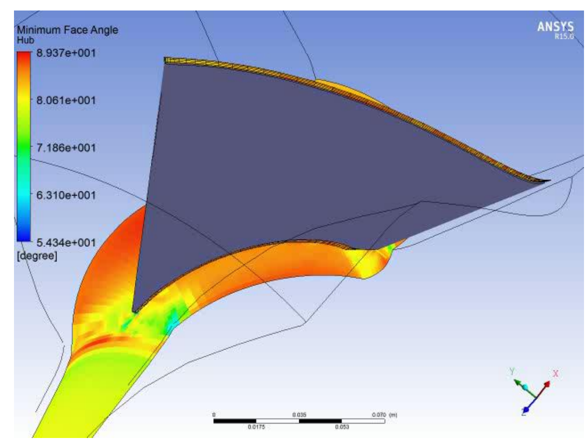
Moving and Deforming Mesh

(CFX)

- **Improved robustness**
  - Better defaults for stiffness
  - Additional options
    - Blended stiffness (β)
    - Jacobian Multiplier (β)
- **Improved sliding mesh on surfaces of revolution**
  - More robust for radial machines
  - Reduced parallel memory overhead
- **Alternative model for periodic mesh motion (β)**



Example showing mesh stiffness resulting with old defaults (top) vs. new defaults (bottom)



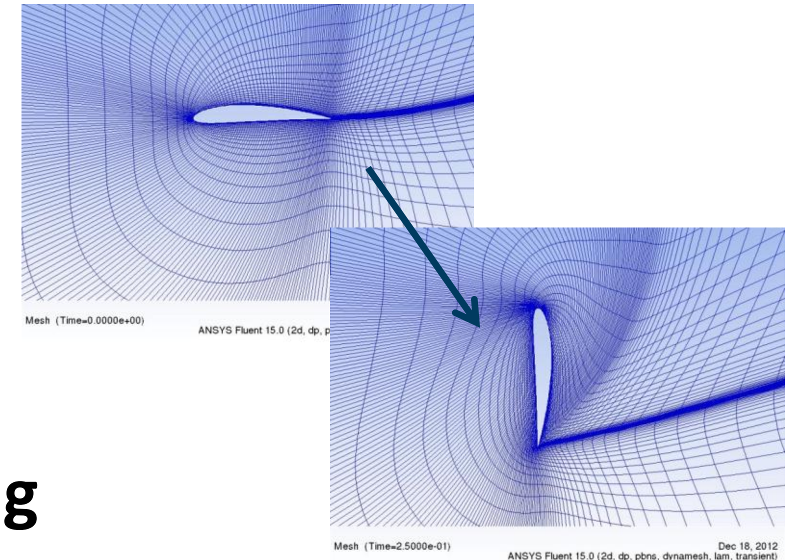
Hydraulic runner test case : rotation by +/- 5 deg showed stable mesh deformation with default settings, with significant improvement in R15

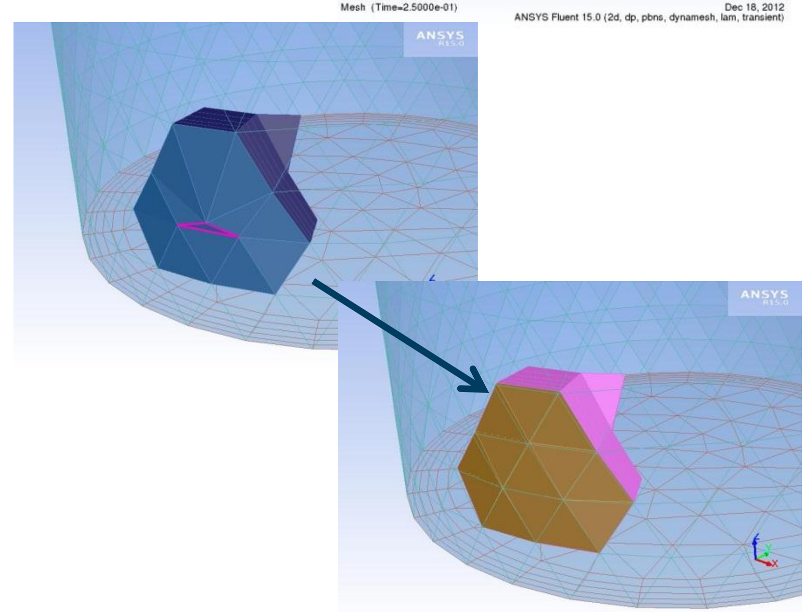
ANSYS

Moving and Deforming Mesh

(Fluent)

- **Increased temporal accuracy**
  - 2nd order temporal discretization with layering and re-meshing
- **Improved accuracy and robustness for mesh smoothing**
  - Node-based solver for diffusion smoothing
  - Linearly elastic solid smoothing
- **Increased flexibility with local re-meshing**
  - Detection and re-meshing of attached boundary layers

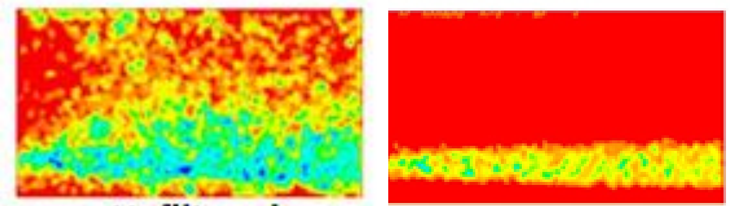




Solver Accuracy and Robustness

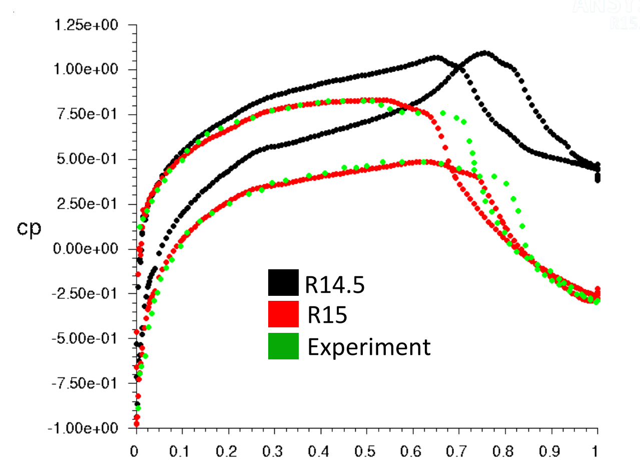
(Fluent)

- **Enhanced accuracy and robustness**
  - Limiter filter
    - Recovers 2nd order accuracy and improves convergence
  - Improved accuracy of node-based gradients near boundaries
  - Improved defaults for single-phase steady state robustness
  - Auto-adjust mode allows Fluent to select best solver settings based on physics (β)



Unfiltered Filtered

Contours show that limiters are active far from the mixing layer (red: active, blue: not active)



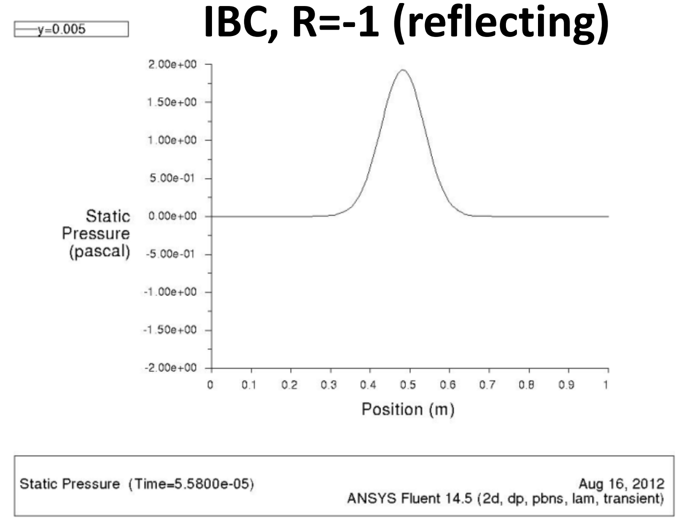
Improved pressure coefficient for transitional flow simulation over transonic 3D DLR-F5 Wing

Other Solver Enhancements

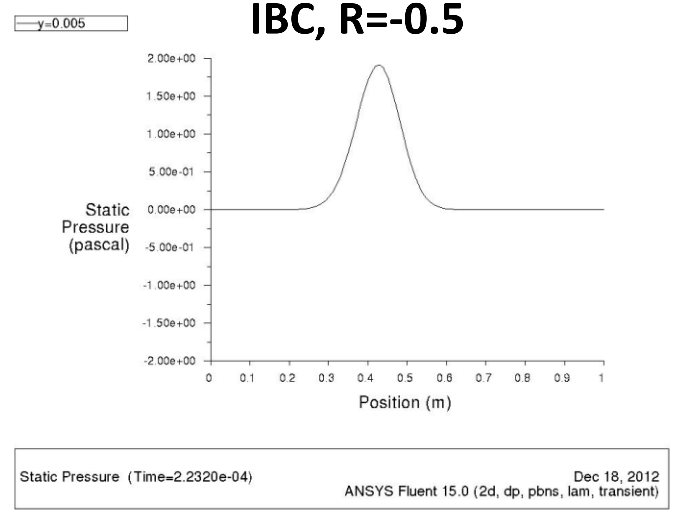
(Fluent)

- **New impedance boundary conditions**
  - Model the impact of pressure reflections from outside the domain of interest
- **Time-step specification with DBNS**
  - More physically meaningful then specifying the CFL number
- **Improved performance for polyhedral conversion**

IBC, R=-1 (reflecting)

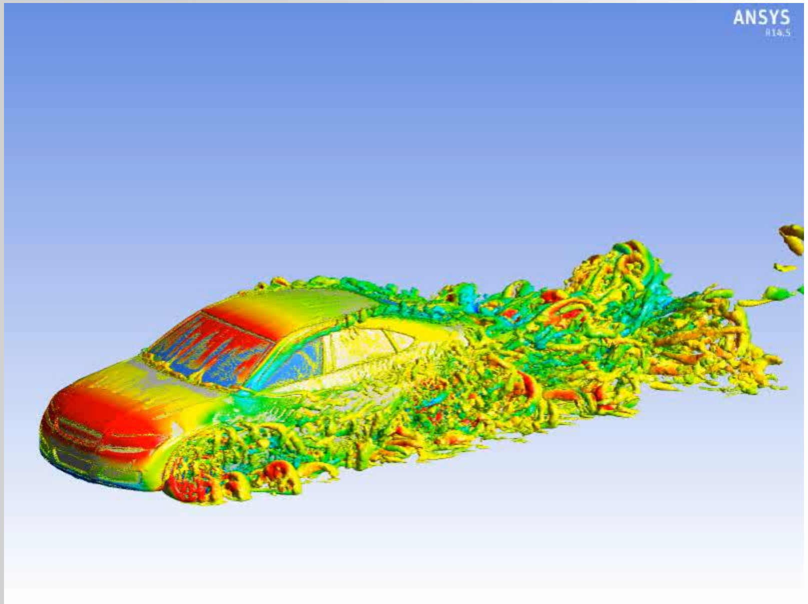


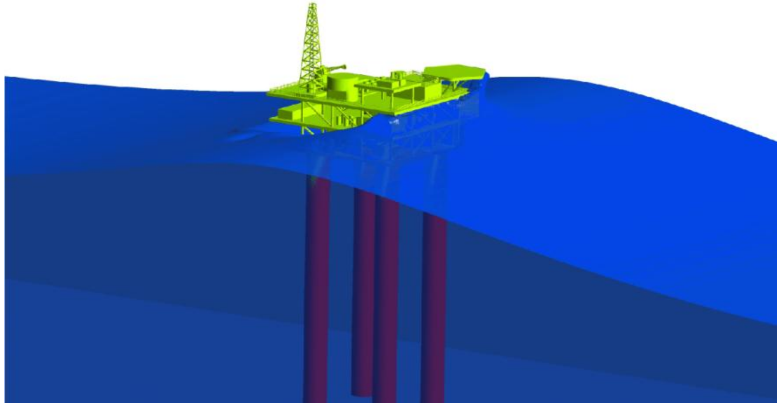
IBC, R=-0.5



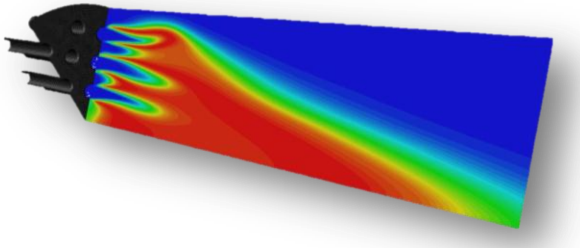
1-D wave at a boundary

Comprehensive Physics Modeling





FLUENT and CFX deliver a comprehensive range of physics modeling capabilities, giving users the detailed insight needed to make design decisions for applications that involve complex physical phenomena.

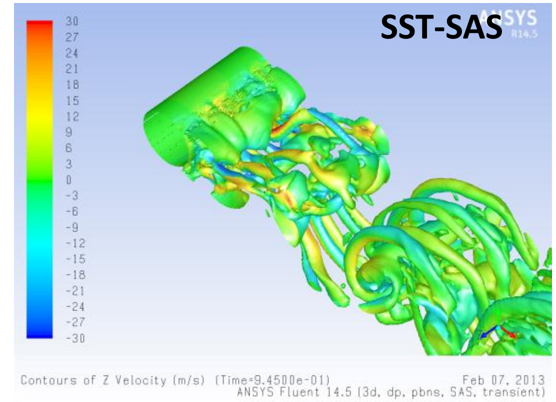


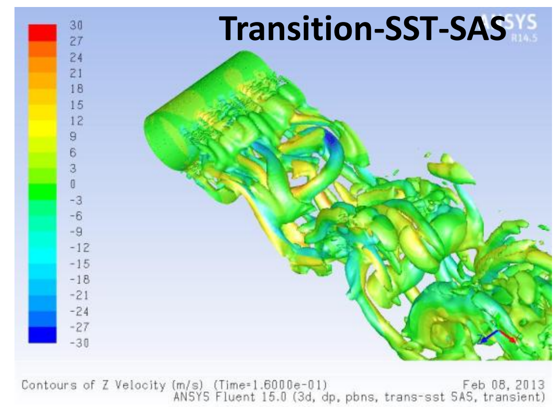
Turbulence

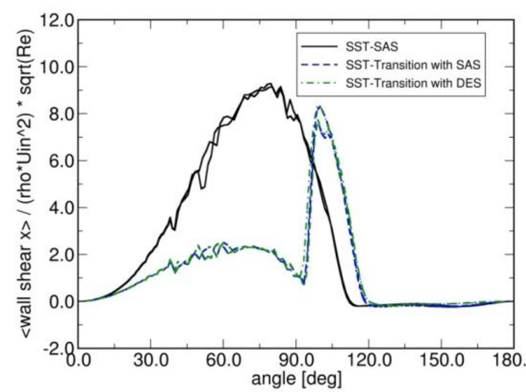
(Fluent)

- **Transition SST model with SAS and delayed DES**
  - Increased flexibility for modeling transitional flows
  - Benefits external flows

- **New WMLES S-Omega model formulation**
  - Offers improved accuracy and broader range of applicability



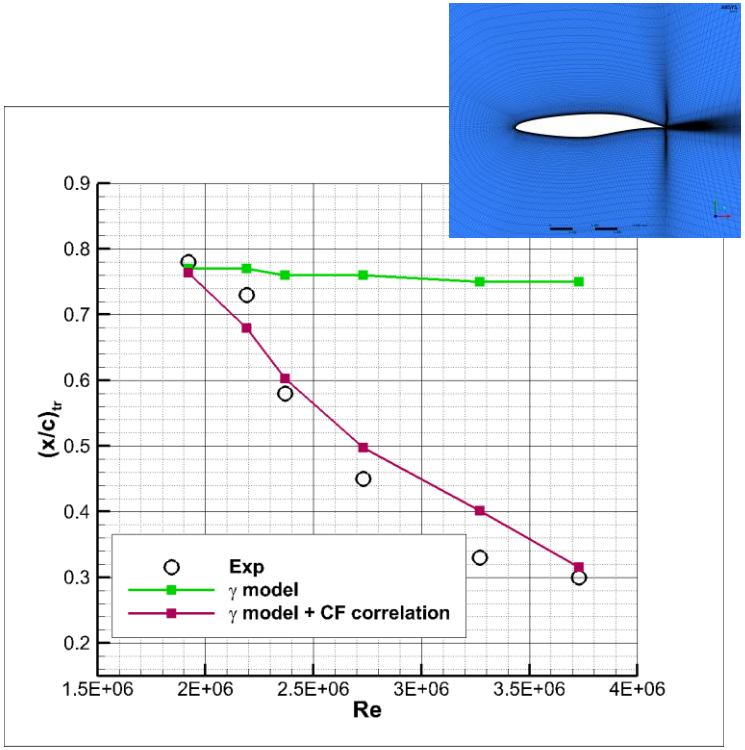




Turbulence

(CFX)

- **One-Equation Intermittency-based Transition Modelling (β)**
  - Evolution of the γ-Re_θ model
- **Alternative wall function calibration for omega-based models (β)**
  - Improved behaviour in the laminar limit
- **Delayed DES (DDES) model (β)**
  - Avoid switch to LES in boundary layer



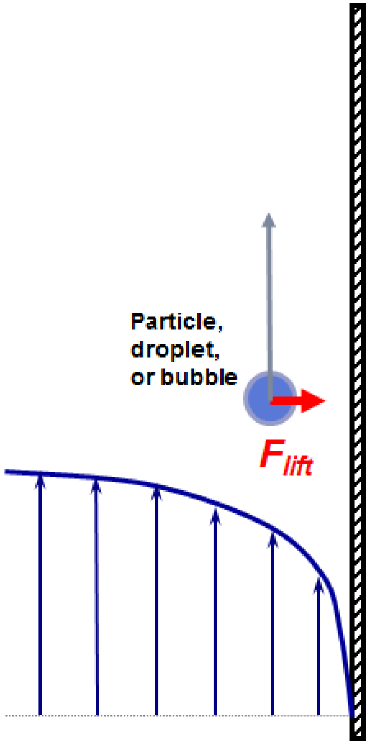
'Infinite' swept wing test case in which ability to capture transition due to cross-flow instability is critical

ANSYS

Eulerian Multiphase

(CFX)

- **Numerous enhancements and extensions**
  - Lift force for Algebraic Slip Model (β)
  - Improved turbulent dispersion for large turbulent Stokes number (β)
  - Different correlations for RPI wall boiling sub-models (β)
  - Bulk adiabatic boundary condition for heat transfer at a wall (β)



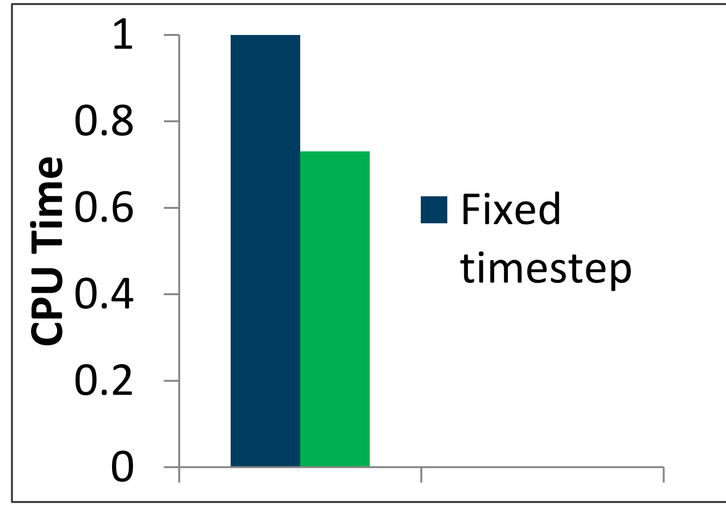
Lift forces lateral to direction of travel can be important when dispersed phase are subject to shear

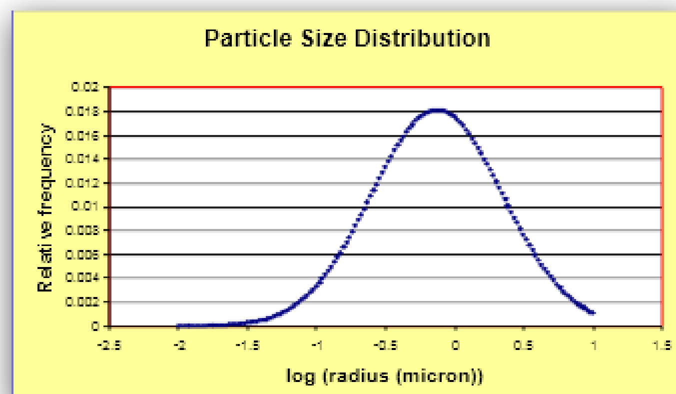
ANSYS

(Fluent)

Eulerian Multiphase

- **Faster and more robust multiphase calculations**
  – Adaptive time-stepping
- **Log-normal initial particle size distribution for population balance**
  – Quicker set-up and more accurate approximation of particle size
- **New interphase heat transfer models**
  – Better prediction of heat transfer between phases

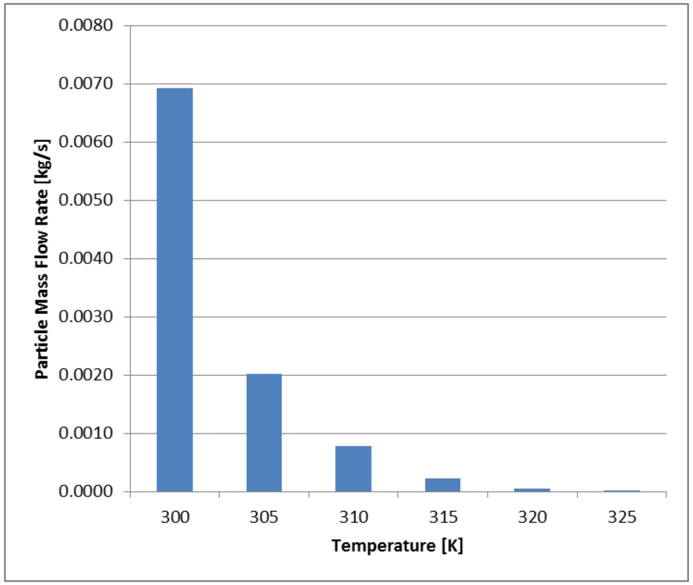




ANSYS *(CFX)*

Lagrangian Particle Tracking

- **New model options and diagnostics**
  - Additional variables for particle histograms
  - Ability to monitor particle mass flow and energy flow at boundaries
  - Ability to output characteristic numbers for particles
    - Ohnesorg, Weber, etc.
  - Particle diagnostics for >10^6 particles



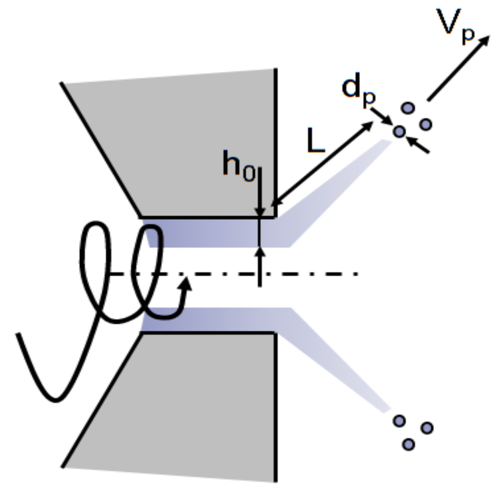
Distribution at a boundary of particle mass flow rate associated with temperature bands

ANSYS

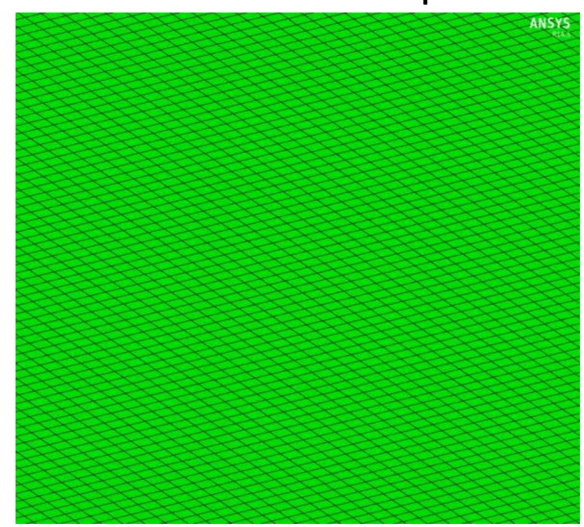
(CFX)

Lagrangian Particle Tracking

- **New model options and diagnostics**
  - Output of model quantities for diagnostics of primary breakup model (LISA, E-TAB, …) (β)
  - Additional wall film model (β)
  - Limits/bound on particle integration timestep and particle temperature (β)
- **Additional modeling flexibility**
  - Use of particle boundary data in CEL expressions (β)
    - E.g. erosion rate and wall mass flux



Schematic of LISA model quantities

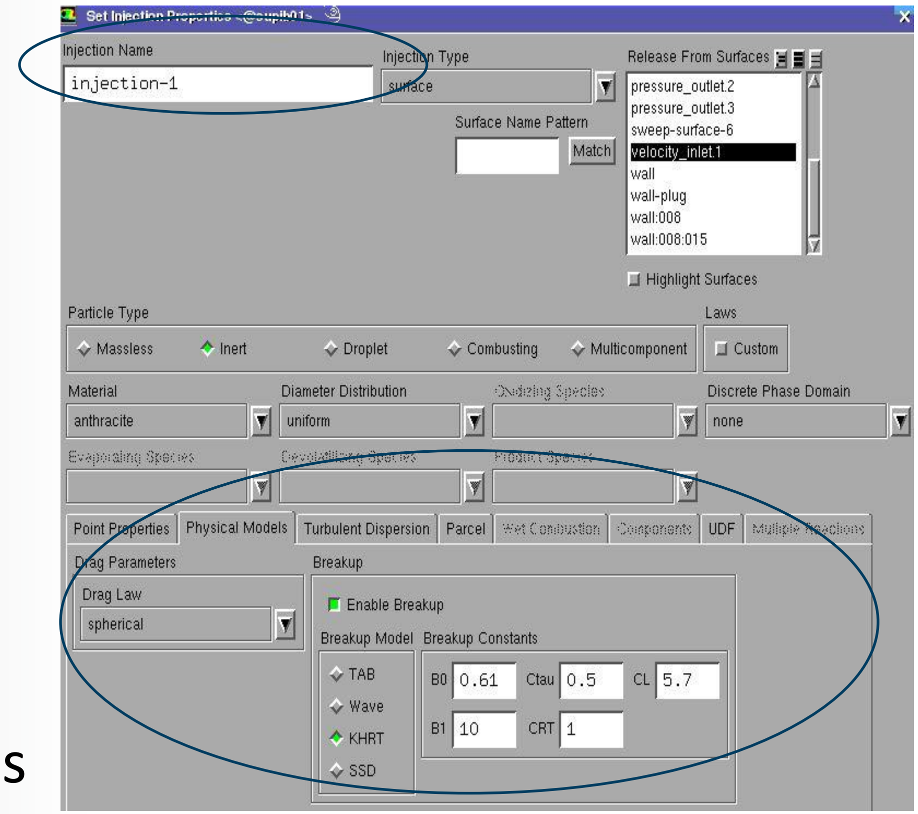


Animation of wall deformation based on particle erosion

Lagrangian Particle Tracking

(Fluent)

- **Increased flexibility and usability**
  - Different drag and break-up laws for different injections
  - Particle density and specific heat as a function of temperature
  - Compute transient statistics for discrete phase model variables
  - Volume seeding of fluid zones (β)

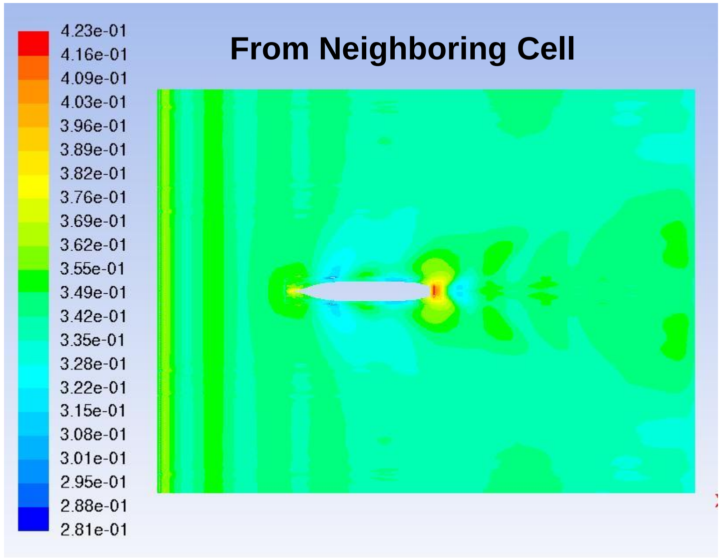


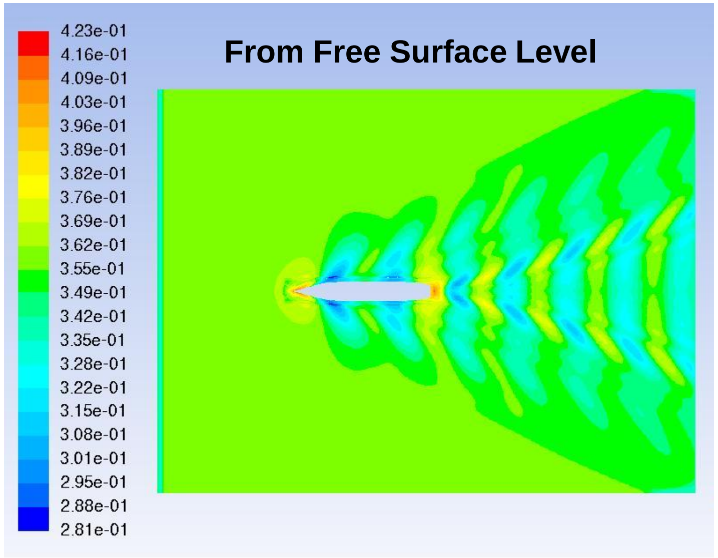
ANSYS

Free Surface Modeling

(Fluent)

- **Faster VOF simulations**
  - Speed-ups of 4% - 36% over a range of cases
- **Open channel flow enhancements**
  - Suppression of numerical reflection at inlet boundary
  - Transient profiles for free surface and bottom level
  - Numerical beach improvements
  - Better modeling of oblique waves

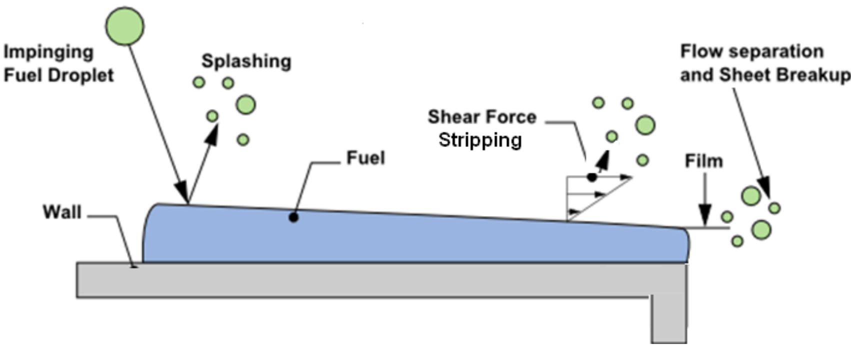


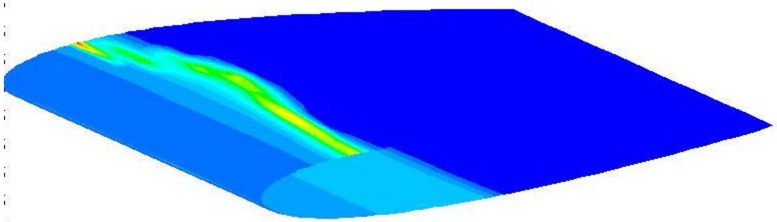


Eulerian Wall Film

(Fluent)

- **Several enhancements and extensions**
  - Compatibility with moving walls and MRF
  - Robust implementation of splashing model
  - Evaporation and condensation with Eulerian and mixture multiphase models
  - Mass flux reporting at boundaries



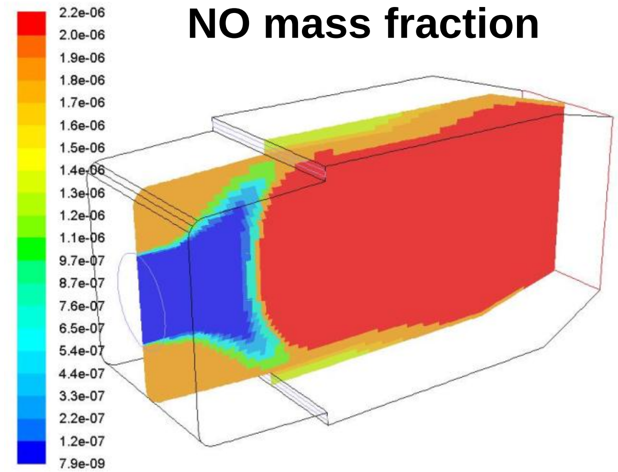


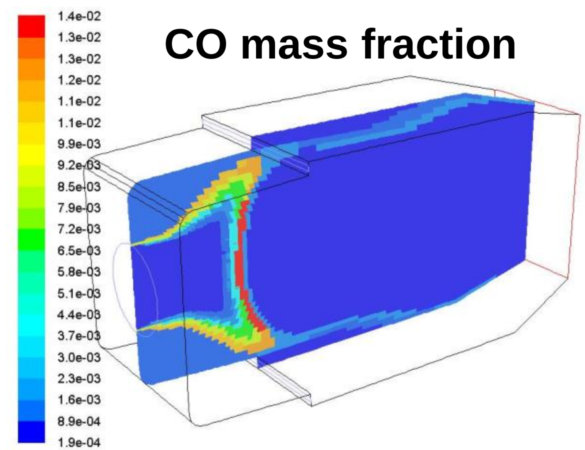
Predicted wall film thickness on a NACA 0012 airfoil verification case

Reacting Flow

(Fluent)

- **Enhanced modeling of detailed chemical mechanisms**
  - Species limit increased from 50 to 500
  - Dynamic mechanism reduction
    - 2-10x faster speed-up depending mechanism size
  - Reactor network model for rapid 3D simulations with detailed mechanisms
  - FGM (Flamelet Generated Manifold) model for diffusion flames



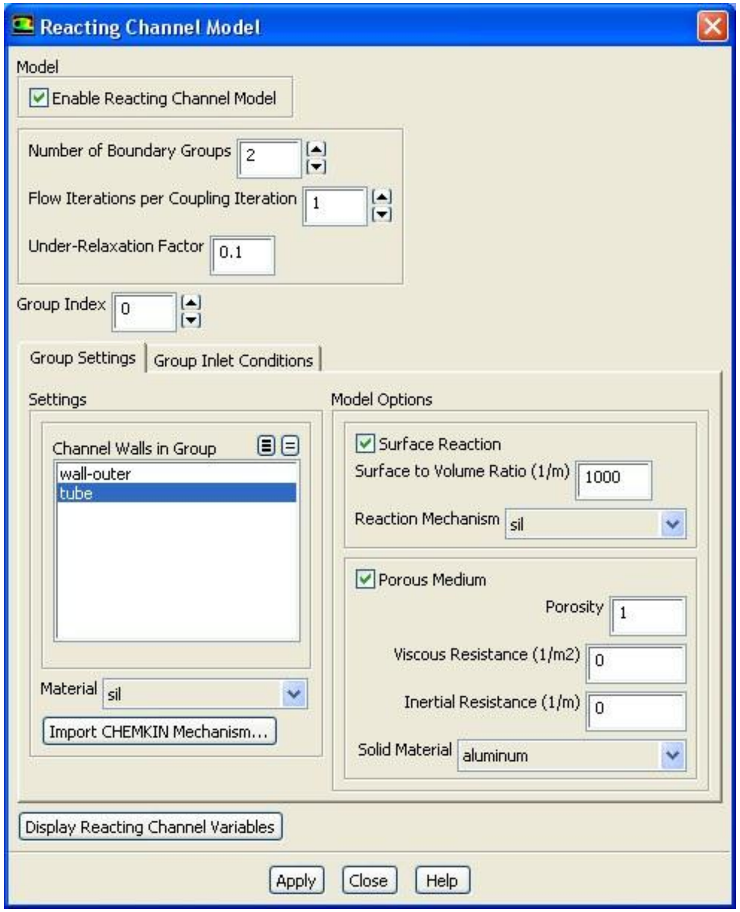


Gas turbine modeled with 20 reactors 325 reactions, 53 species

Reacting Flow

(Fluent)

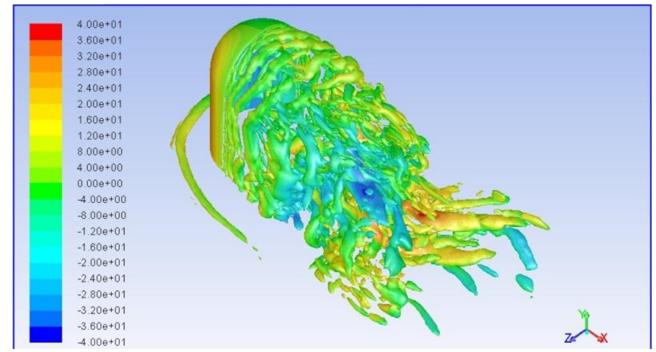
- **Enhanced reacting channel model**
  - Ability to include porosity and surface chemistry
- **Anisotropic species diffusion**
- **Improvements for IC engine modeling**
  - New spark model
  - Multiple unsteady flamelets
- **Generalized electrochemistry (β)**



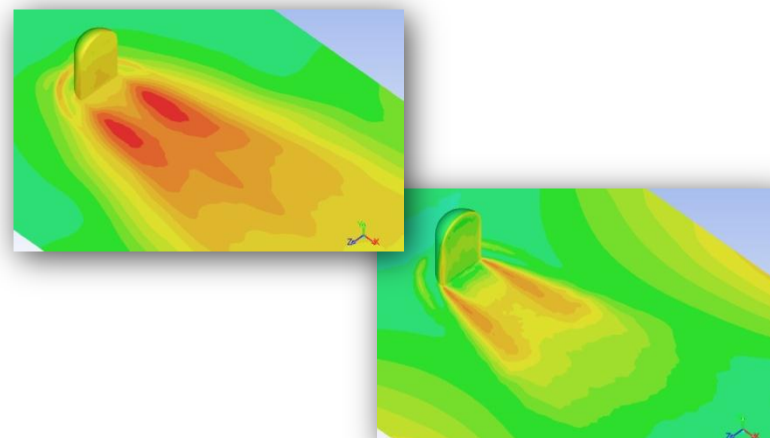
Heat Transfer and Acoustics

(Fluent)

- **Extensions for heat transfer and radiation**
  - New multilayer shell conduction model
  - Surface-to-surface radiation with non-conformal interfaces
  - Anisotropic heat conduction in solids
- **Improved acoustics analysis**
  - Banded analysis of acoustic sources (β)



Unsteady flow structures computed in simulation of generic car mirror

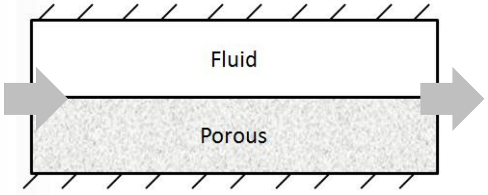


SPL for octave band centered at 63 Hz (left) and 500 Hz (right), showing areas of high noise generation

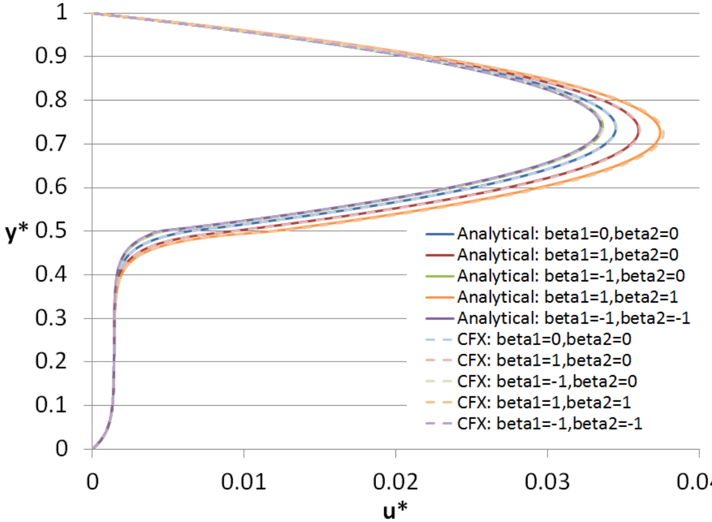
Porous Media

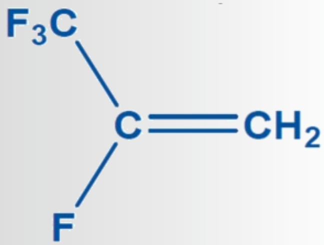
(CFX)

- **More robust and more accurate modeling of flow parallel to porous interface**
  - Improved treatment of pressure gradients at fluid-porous interface
  - Additional improvements for stress closure and stress jump (β)
- **Addition of R-1234yf to Materials Database**



Selected test case (periodic flow) to verify improved numerics shows good comparison to analytical solution



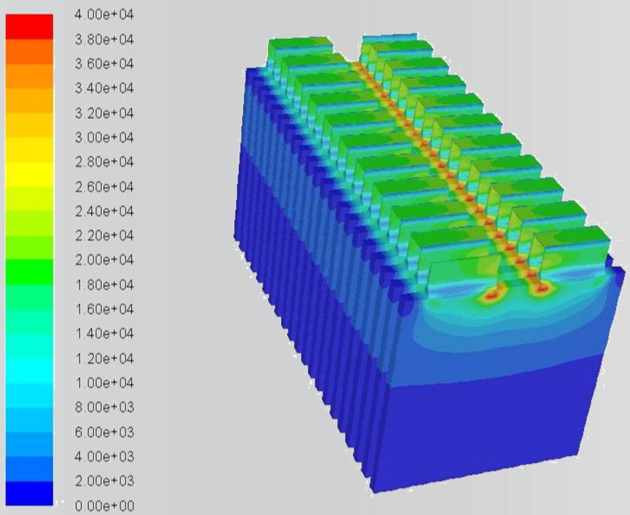


Battery Modeling

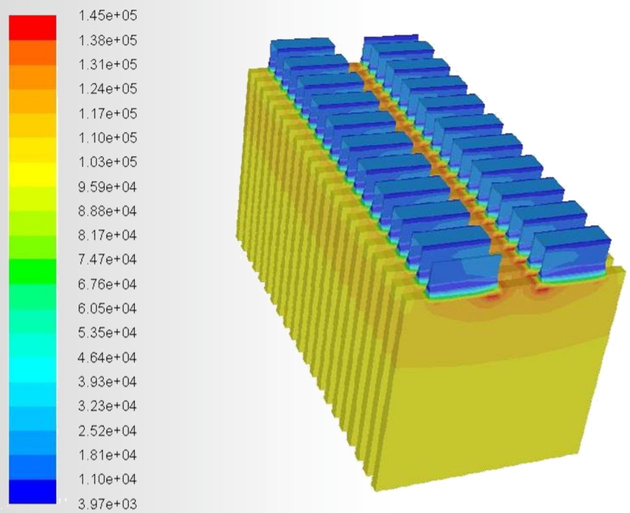
ANSYS

(Fluent)

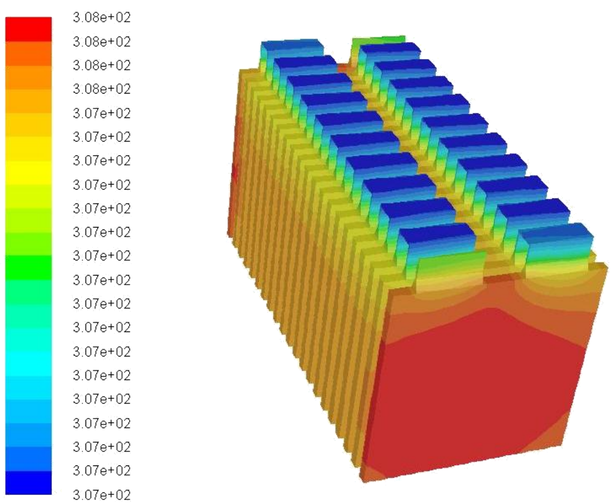
- **New multi-scale, multi-dimensional (MSMD) battery model**
  - Single battery cell or multiple cell battery pack
  - Fully coupled flow, thermal and electrochemistry
  - Fully parallelized



Ohmic heat generation

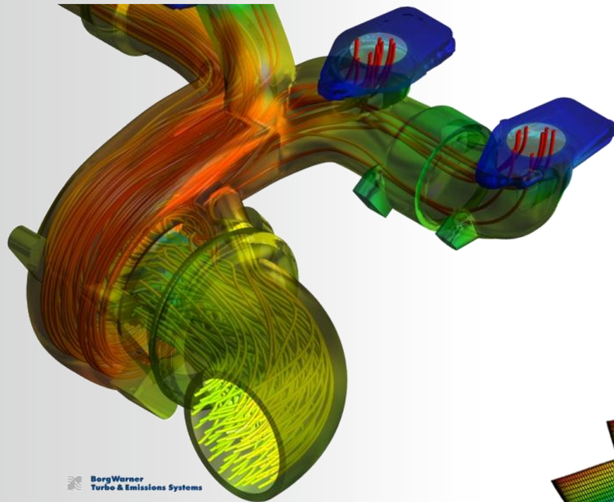


Total heat generation

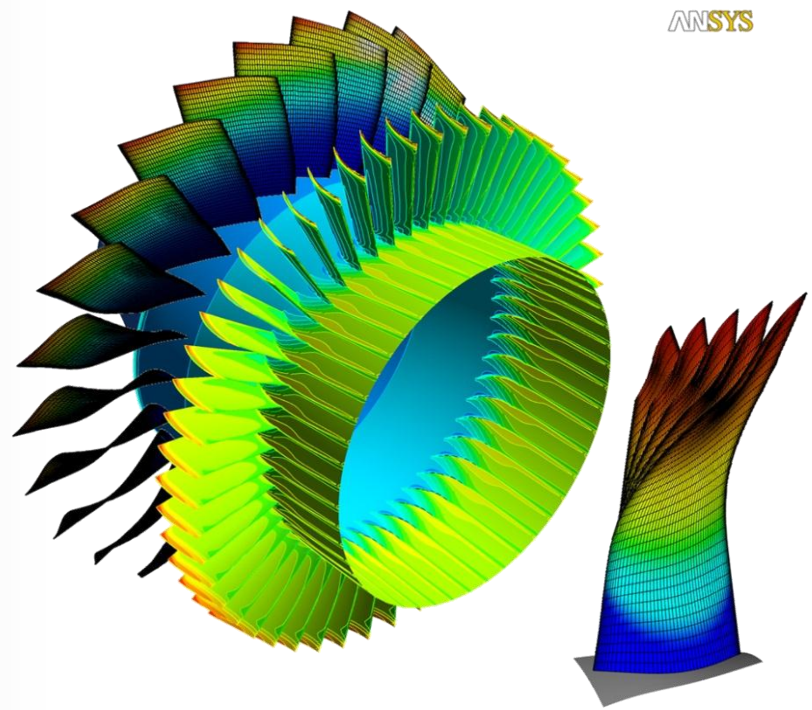


Temperature

Turbomachinery



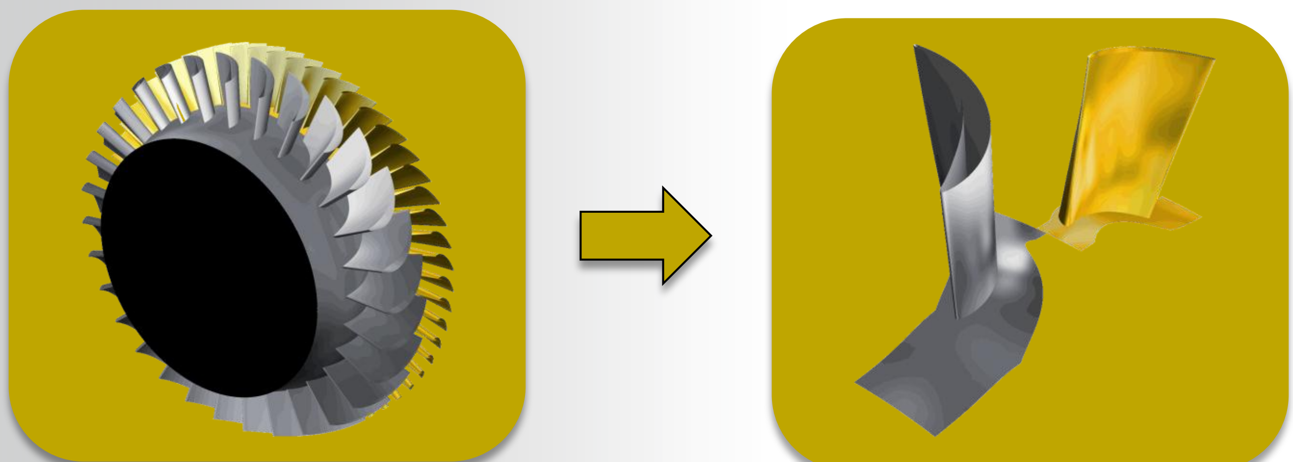
The ANSYS suite of customized tools for turbomachinery design and analysis enables users to work efficiently and effectively as they improve and optimize machine performance.



ANSYS
(CFX)

Turbomachinery

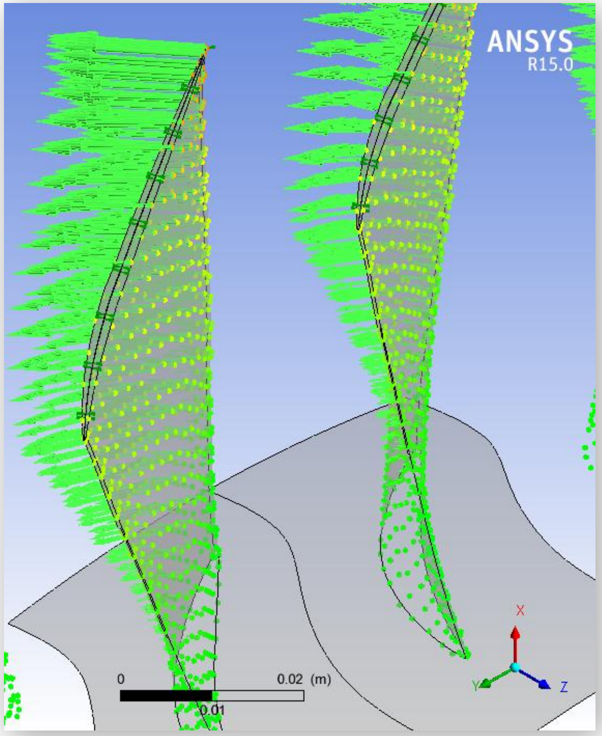
- **Recent focus on developing methods to accurately and efficiently simulate transient phenomena in blade rows → Transient Blade Row (TBR) models**
  - New models minimize number of simulated passages
  - Enormous efficiency gains!
  - Reduced infrastructure requirements



Turbomachinery

(CFX)

- **Easier and more automated workflow for Transient Blade Row (TBR)**
  - Profile vector visualization in CFX-Pre for clarity of flutter motion being applied
  - Definition of integer parameters to enable parametric studies in Workbench (e.g. vary nodal diameter)
    - Applicable any integer parameter
  - Built-in aero-elastic damping calculation & monitoring for blade flutter

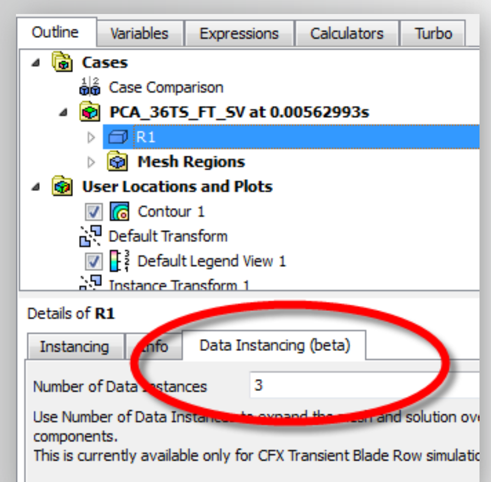


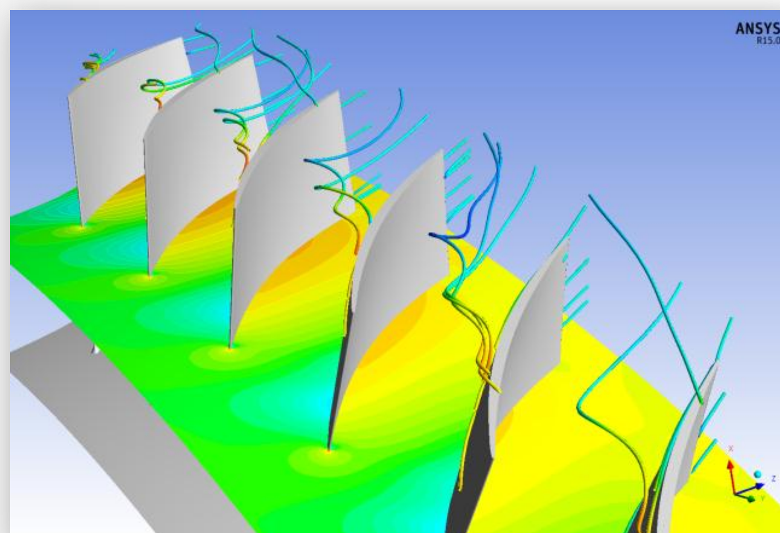
ANSYS

Turbomachinery

(CFD-Post)

- **Instancing and Expansion of TBR solutions for post-processing**
  - Full range of plots and quantitative analysis with data instancing
    - Points, lines, planes, surfaces (incl. turbo), volumes, …
    - Iso-clips, streamlines, probes, volume rendering, vortex cores, highlighting, …

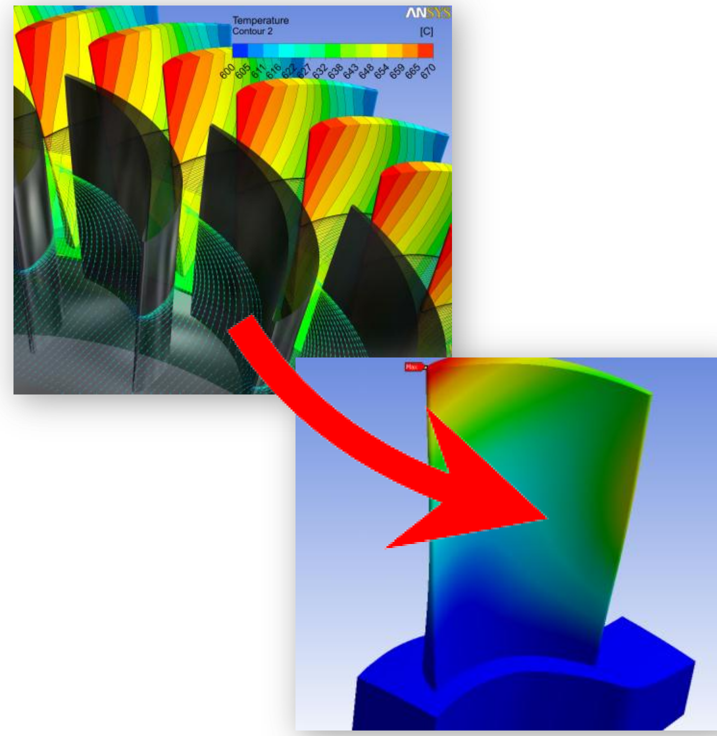




ANSYS *(CFX)*

Turbomachinery

- **Support for Forced Response**
  - Complement blade flutter analysis with ability to export complex pressures from transient flow simulation for load application in ANSYS Mechanical



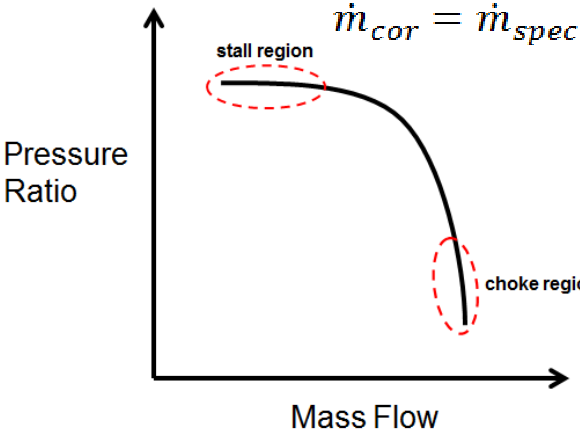
- **Continued R&D to extend and improve TBR and related capabilities**
  - R&D on enhanced robustness, improved initialization, new methods
  - Mesh motion improvements for flutter (as per separate slide)

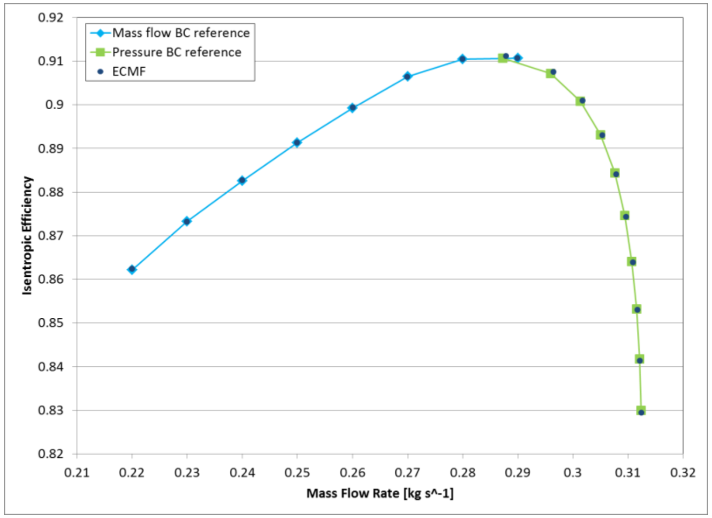
Turbomachinery

(CFX)

- **New boundary condition for use across full speed line**
  - "Exit-corrected mass-flow outlet" applicable from deep choke to stall
    - Avoid set-up change along speed line, ensure continuity
    - Gives desired mass flow specification in stall region, and includes effect of pressure variation in choke region
  - Can improve robustness, e.g. at start-up

$$\dot{m}_{cor} = \dot{m}_{spec} \frac{\sqrt{T_{01}/T_{ref}}}{P_{01}/P_{ref}}$$



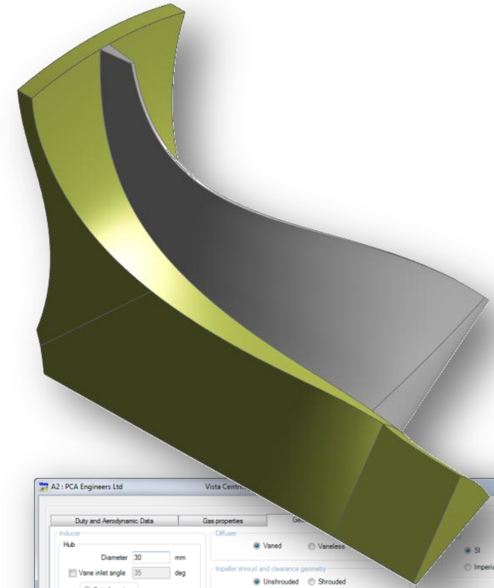


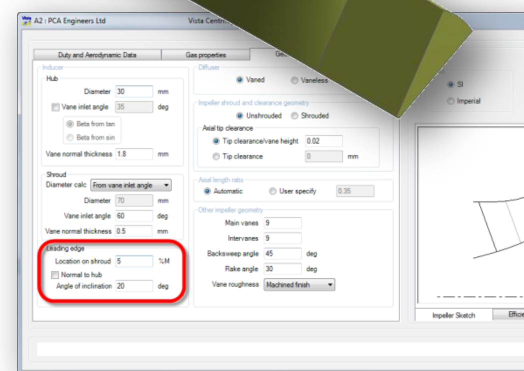
Speed line for compressor test case showing consistency of results between new exit-corrected mass flow BC and other BCs

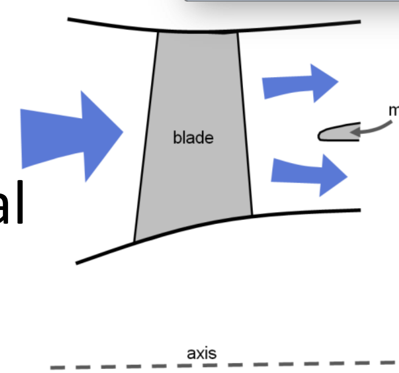
TurboSystem

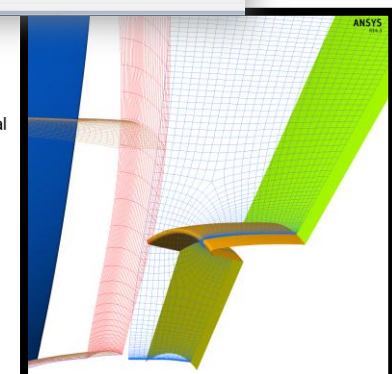
(BladeModeler, Vista Tools, TurboGrid)

- **BladeModeler**
  - Streamlined optimization workflow
  - Blade hub sector cut for FSI
  - Improved TurboGrid export (β)
  - Radial Element Blades (β)
- **Vista Tools**
  - Enhancements to Vista CCD and Vista RTD
- **TurboGrid**
  - Topology template for meridional splitters

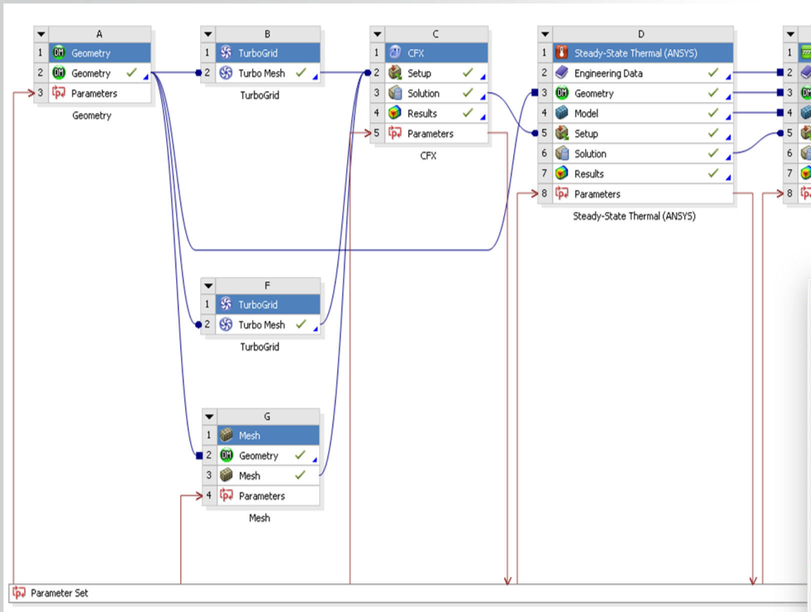


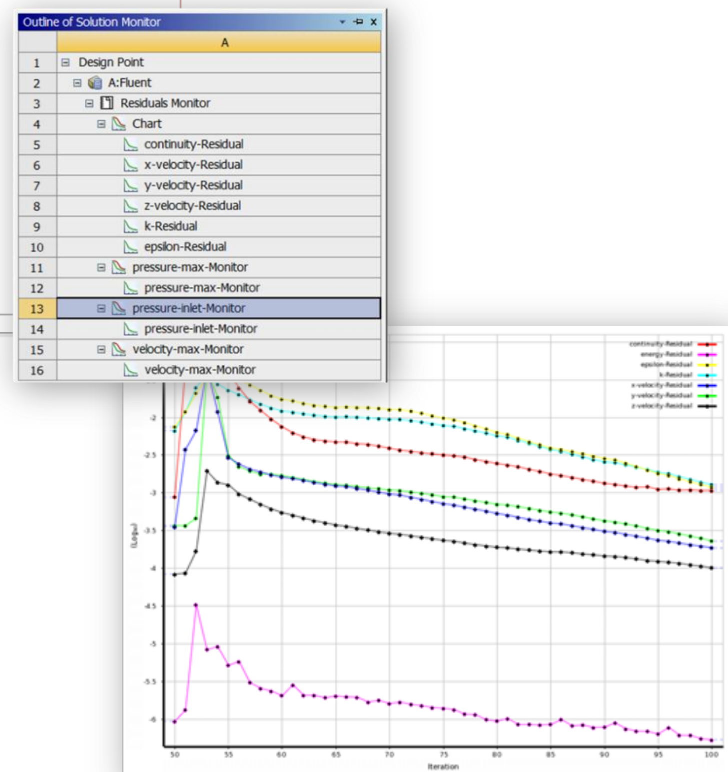






Enhanced Usability



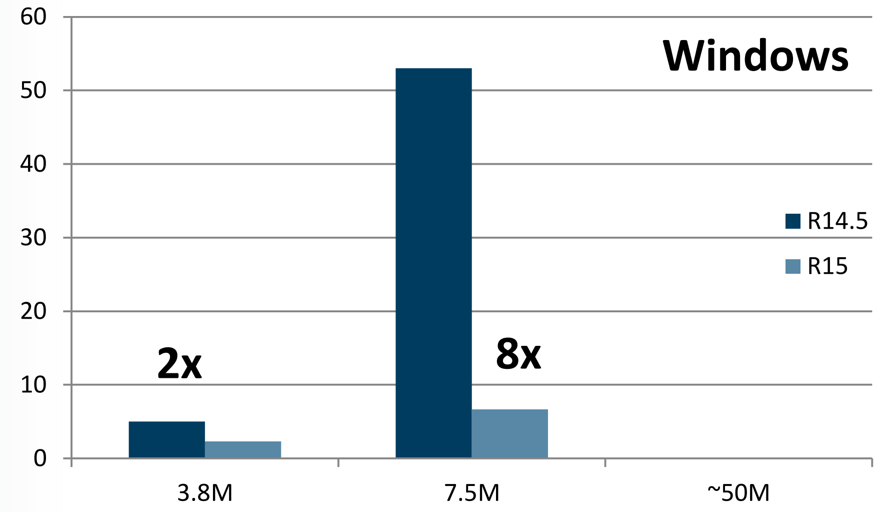


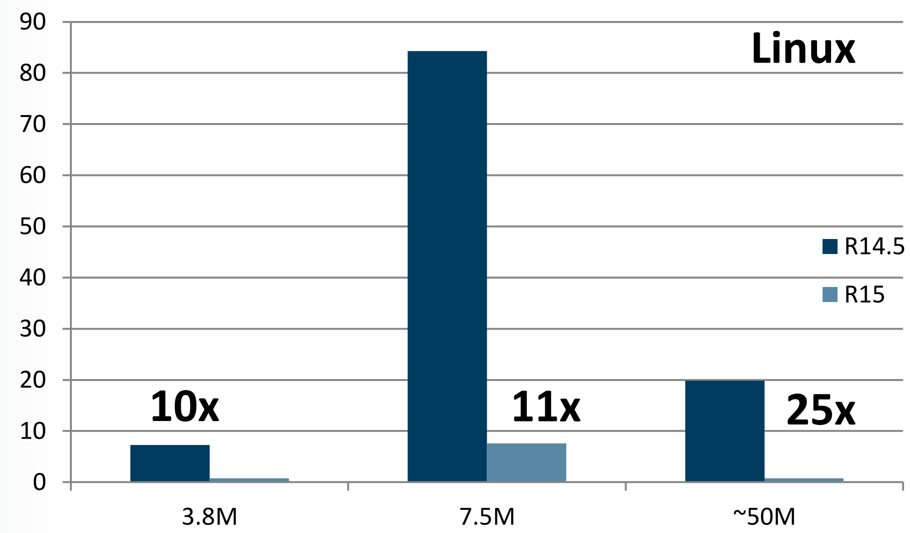
ANSYS constantly strives to help users increase their productivity and streamline their workflow by improving the ease-of-use and intuitiveness of our simulation tools.

Fluent User Environment

(Fluent)

- **Enhanced solution monitoring**
  – Track forces, moments, surface and volume averages over user-specified intervals
- **Significant performance improvements**
  – Rotating the model in the graphics window is between 2-25x faster!
  – More efficient loading of cases with large numbers of zones



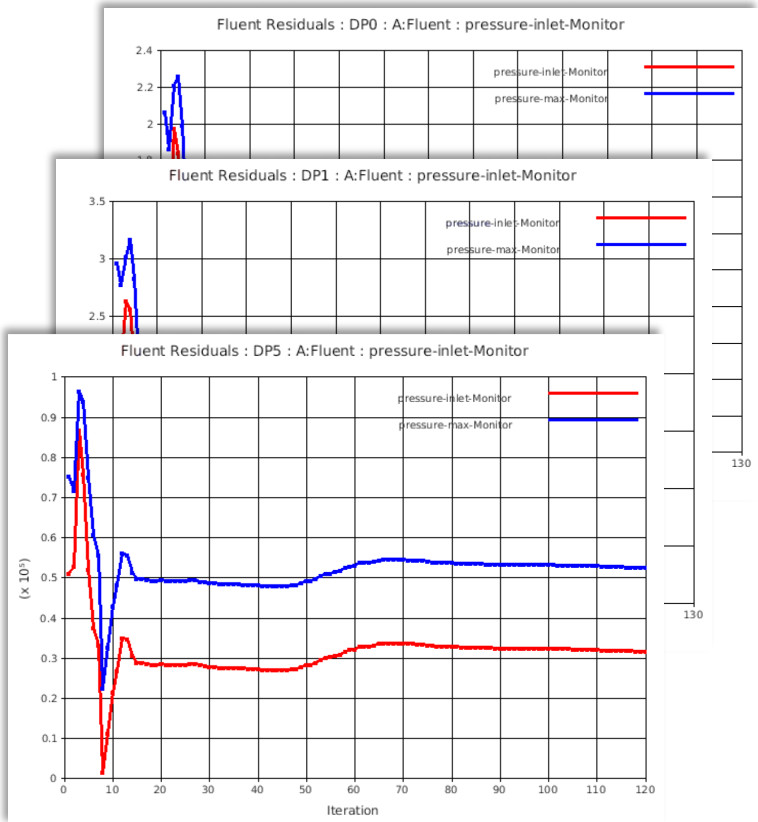


ANSYS

Workbench Integration

(Fluent)

- **Expanded solution monitoring**
  - Plotting of surface, volume and force monitors
- **Advanced parameter options**
  - Use parameters in conjunction with UDFs and scheme function
- **New data interpolation option (β)**
  - Initialize on updated meshes with prior related results
  - Boosts performance

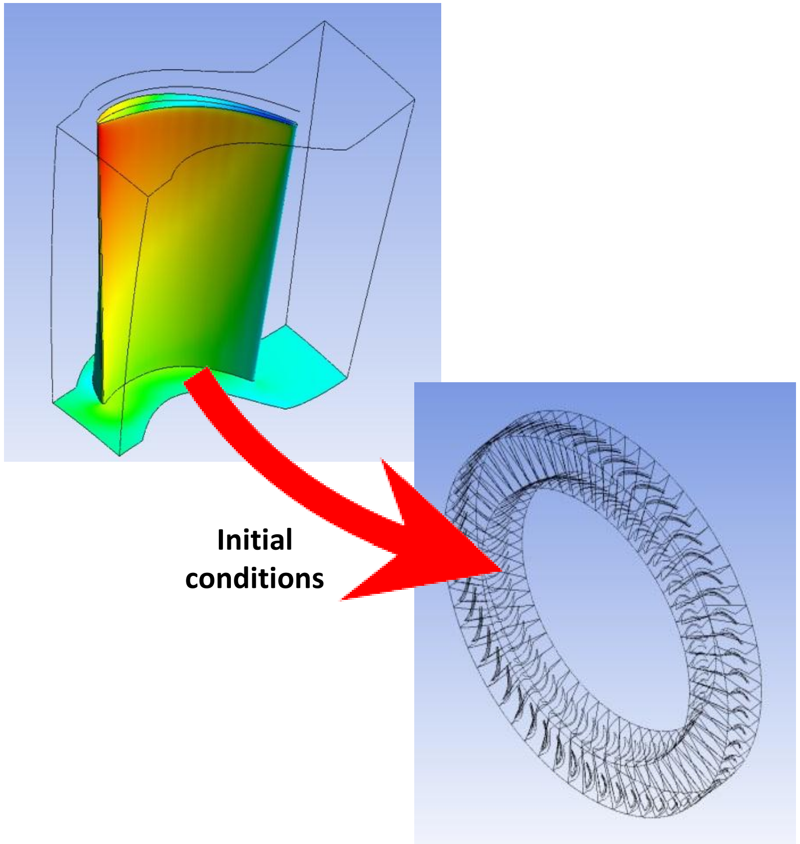


Surface Monitors saved in WB for different Design Points

Solution Initialization

(CFX)

- **Ability to use single passage turbomachinery solutions to initialize solution on multiple passages**
  - Efficient means to generate start conditions for large model
- **Additional controls for specific applications**
  - Multiphase, particle tracking, and mesh motion



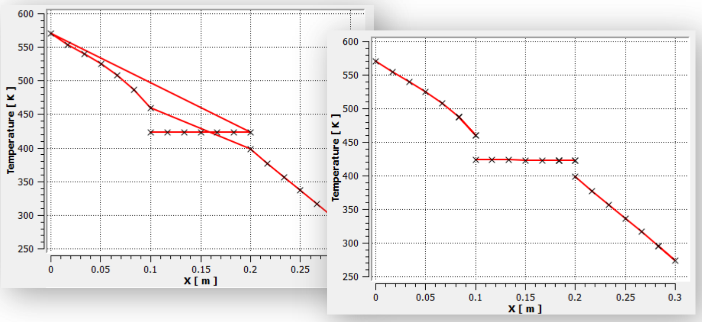
General Post-Processing

(CFD-Post)

- **Improved performance**
  - Efficiency in expression handling
- **Improved cut line ordering for multi-domain cases**
  - Includes proper display of discontinuities
- **Project-wide reporting**
  - Ability to include CFD-Post content in project-wide in ANSYS Workbench reports

Action	R14.5	R15.0 P1
Select Expression Tab	38 [s]	< 1[s]
Double-click an expression	20 [s]	< 1[s]
Change an expression value	20 [s]	< 1[s]

Performance improvement in expression handling in R15 shown on a case with approx. 800 CEL expressions

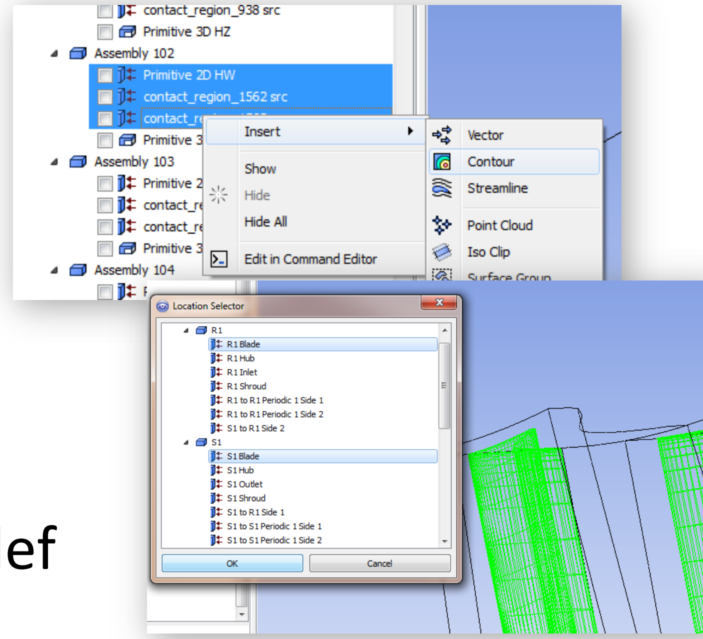


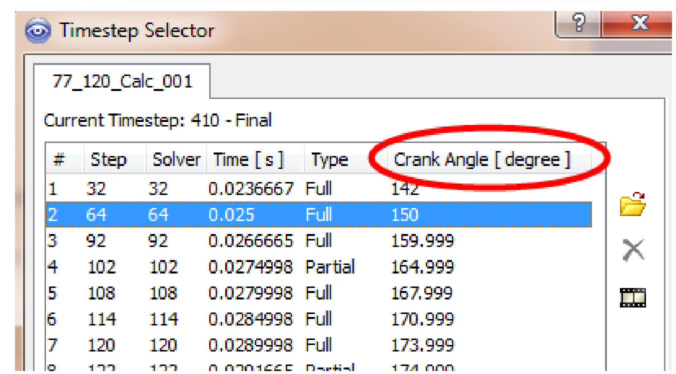
R14.5 (left) vs. R15 (right) chart of a cut line in a multi-domain case, showing discontinuity correctly

General Post-Processing

(CFD-Post)

- **Ability to use domain hierarchy for location multi-select**
  – Much improved usability for multi-domain cases with many regions
- **Direct Import of *.msh file**
  – Eliminate need to create *.cas or *.def file to analyze mesh in CFD-Post
- **Timestep selection based on crank angle**
  – For IC engine applications

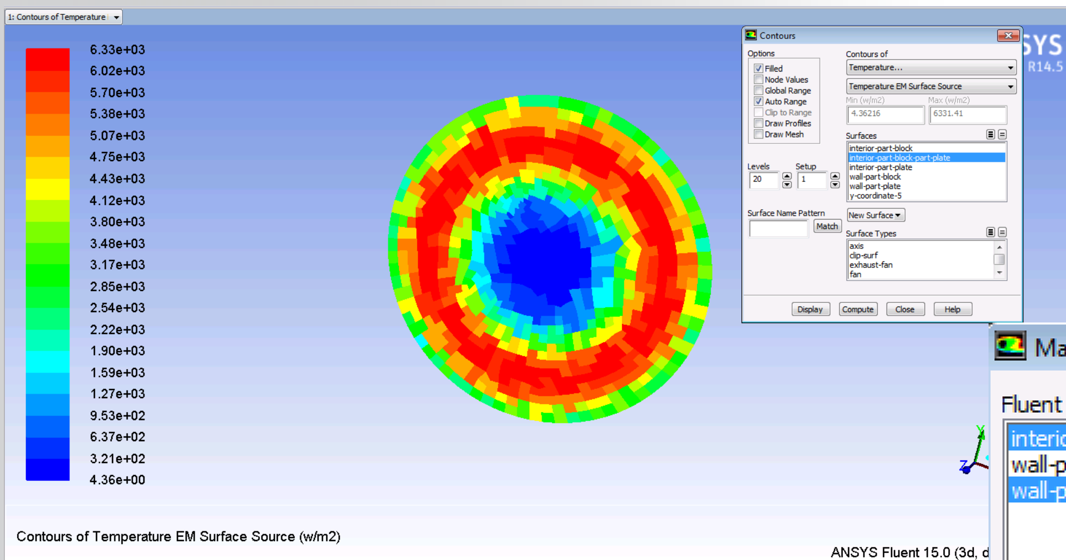


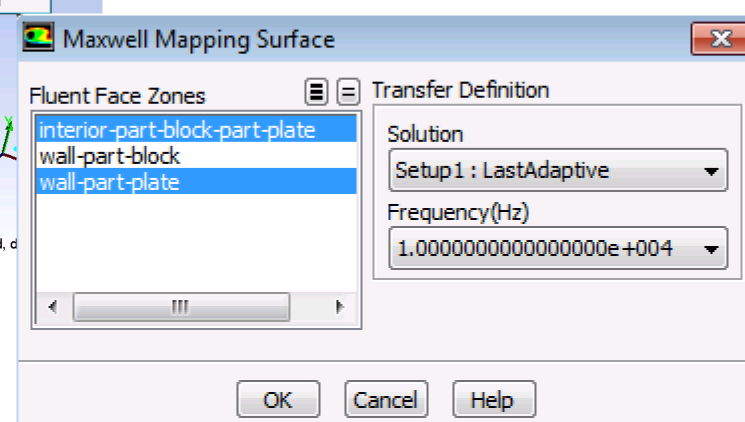


Electro-Magnetics Coupling

(Fluent)

- **Improved surface mapping capabilities for coupling between Fluent and Maxwell**
  - Support for surface losses on interior zones



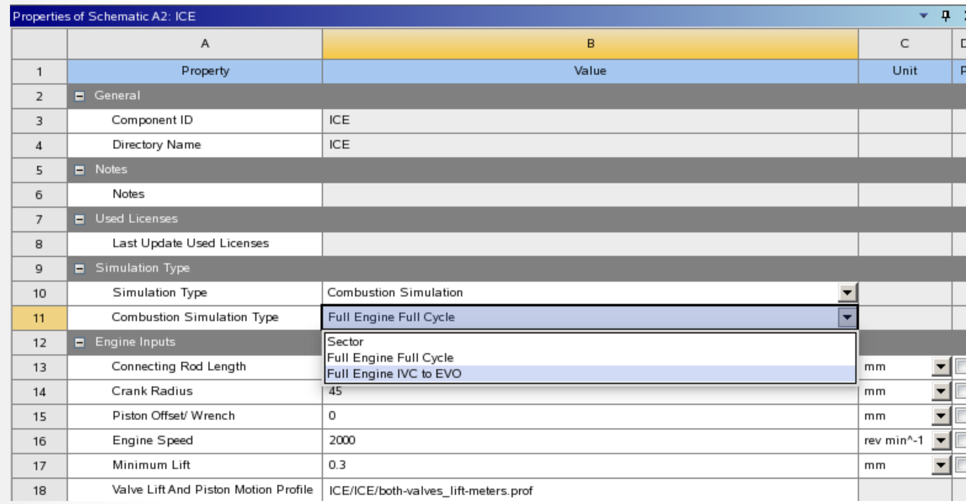


Contours of the mapped surface loss in Fluent (Above)
Maxwell mapping surface panel (Right)

Workbench IC Engine System

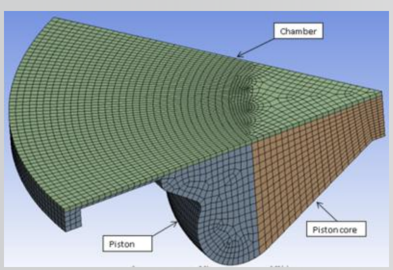
(Fluent)

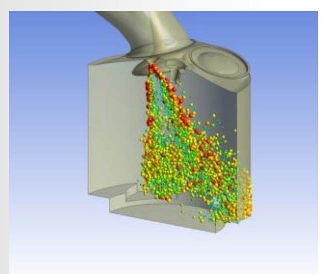
- **Expanded capabilities for WB-IC Engine**
  - Complete setup for gasoline and diesel combustion simulations from geometry to reporting

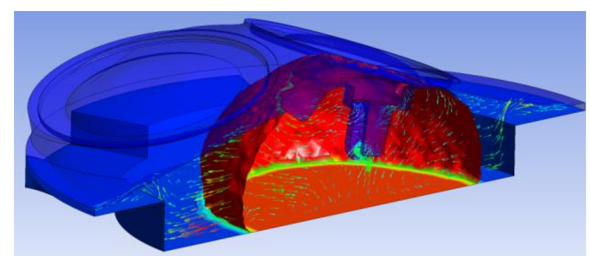


Properties of Schematic A2: ICE

	A	B	C	D
1	Property	Value	Unit	P
2	General			
3	Component ID	ICE		
4	Directory Name	ICE		
5	Notes			
6	Notes			
7	Used Licenses			
8	Last Update Used Licenses			
9	Simulation Type			
10	Simulation Type	Combustion Simulation		
11	Combustion Simulation Type	Full Engine Full Cycle		
12	Engine Inputs	Sector Full Engine Full Cycle Full Engine IVC to EVO		
13	Connecting Rod Length		mm	
14	Crank Radius	45	mm	
15	Piston Offset/ Wrench	0	mm	
16	Engine Speed	2000	rev min^-1	
17	Minimum Lift	0.3	mm	
18	Valve Lift And Piston Motion Profile	ICE/ICE/both-valves_lift-meters.prof		





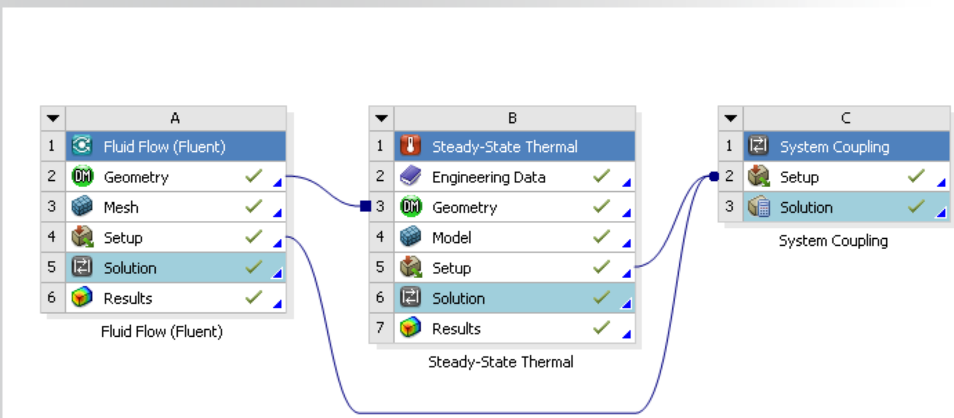


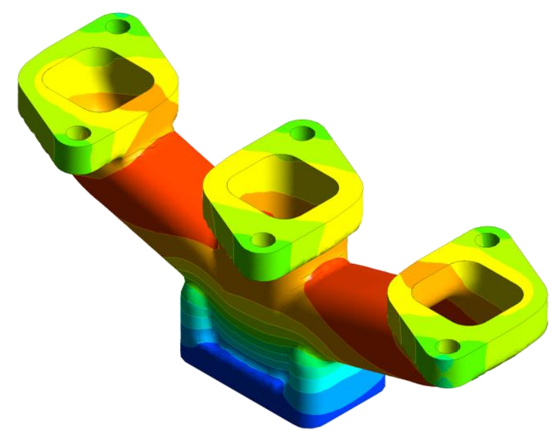
ANSYS

Fluid Structure Interaction

(Fluent)

- **Two-way coupling between Fluent and Mechanical**
  - Surface thermal FSI
  - Surface thermal and structural FSI





Two-way transfer of surface temperatures and convective film coefficients to solve for the temperature field in an exhaust manifold.

Summary

- **Many significant improvements and enhancements coming for fluids customers with ANSYS 15**
  - Please consult documentation for full details!

Thank You

