Ansys Fluent

Past, Present and Looking Forward

Adam Norman – Manager, Application Engineering

Year after year, Ansys Fluent's **cutting-edge innovations** have helped engineers overcome the most challenging design obstacles imaginable when dealing with fluid-dynamics product problems

2022
- PyFluent
- Multi-GPU Solver

2021
- VOF ⇔ DPM ⇔ EWF
- AI Turbulence Tuning

2020
- 3D Battery Electrochemistry
- AIAD Transition

2019
- GEKO Turbulence Model
- Mosaic™ Meshing

2018
- Spray Break-up Model

2017
- PUMA Adaptation
- SBES Turbulence Model

2016
- Supercomputing Record 170,000 Cores

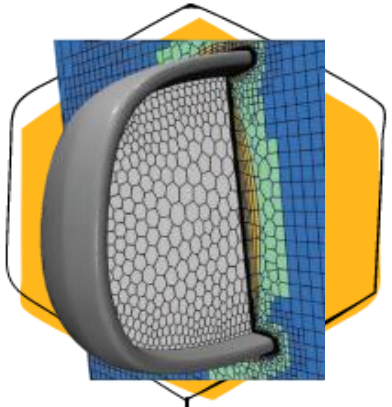
2014
- GPU Accelerated Offloading
- Adjoint Solver

2006
- Fluent Inc. was acquired by Ansys Inc.

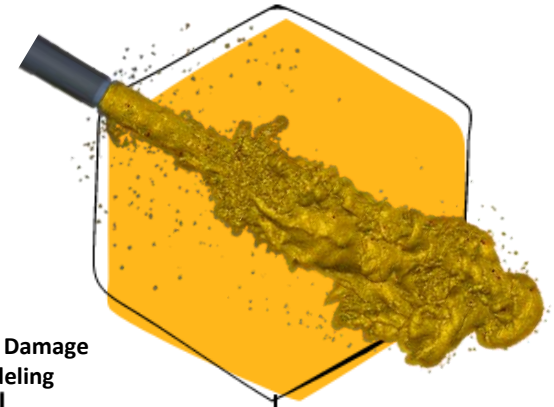
1980s
- Fluent was the first commercial CFD software to have a graphical user interface and workflow rather than a command-line input

2018 : New Capabilities Highlights

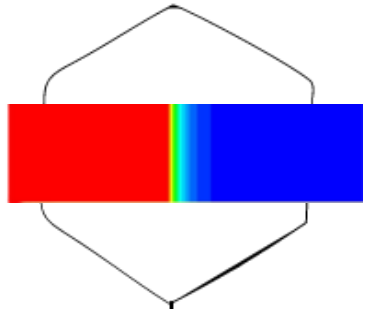
Mosaic™ Meshing



Spray Break-Up Model



Capillary Pressure Modeling in Porous Media



+ PERFORMANCE
+ ACCURACY
+

Erosion Damage Modeling

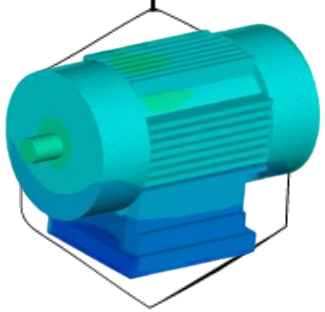
Expressions

Parallel Flamelet Generation

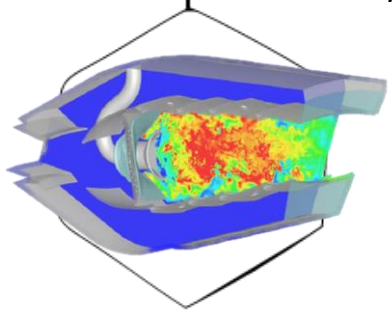
Surface Integrals as Adjoint Observables

AGR for Reacting Flows

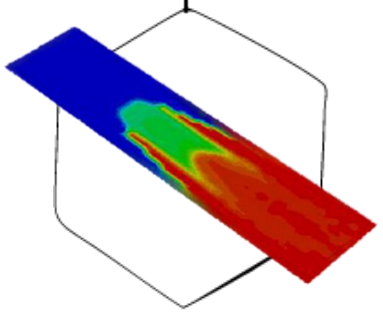
ROM Builder



Load Balancing for Parallel CHT Simulations



DPM Domain Method Enhancements



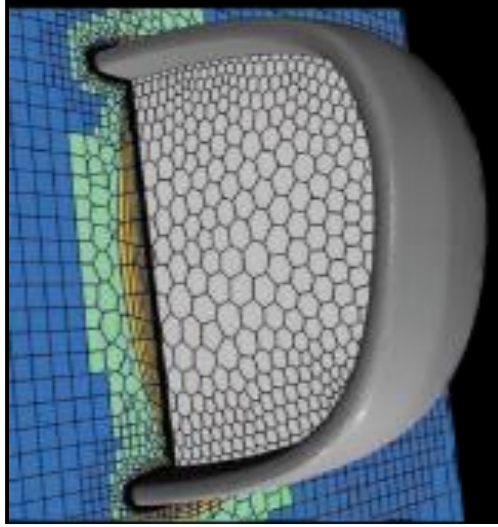
LWF and EWF Enhancements

Bringing to the market unique and unprecedented innovations in mesh generation

Ansys

2018 : Increased Performance Highlights

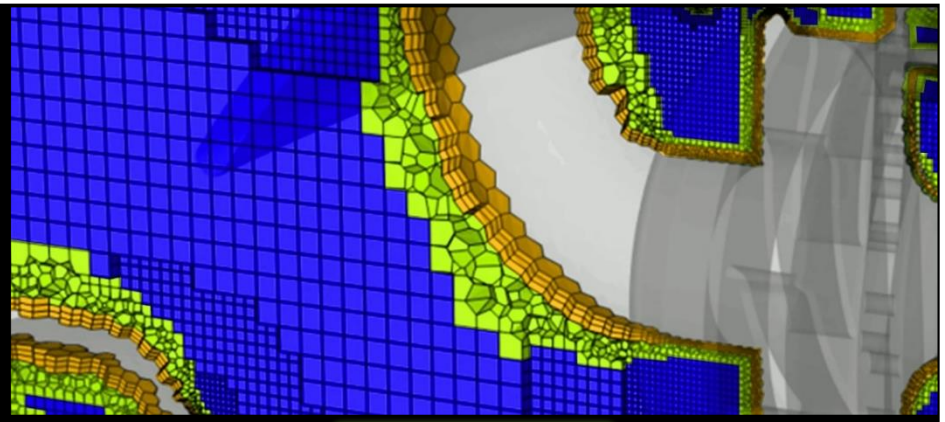
Mosaic™ Meshing



Ansys's unique Mosaic™ Meshing combines a hexcore bulk volume with polyhedral prism and transition cells at the boundaries for accurate flow resolution

Benefits

- Up to 50% cells reduction compared to conventional Hexcore
- Higher cell-quality than all-poly or Hexcore meshes
- Lower memory consumption than all-poly
- Higher solver performance than all-poly or Hexcore



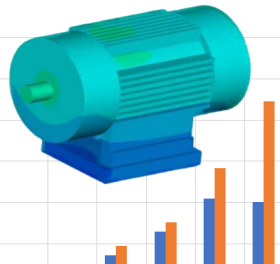
Poly Prism
High quality
Significantly Fewer cells than tri-prisms

Mosaic
Conformally connects poly prisms to hexcore

Hexcore
High quality
Fast solve time

Load Balancing for Paralell CHT Simulations

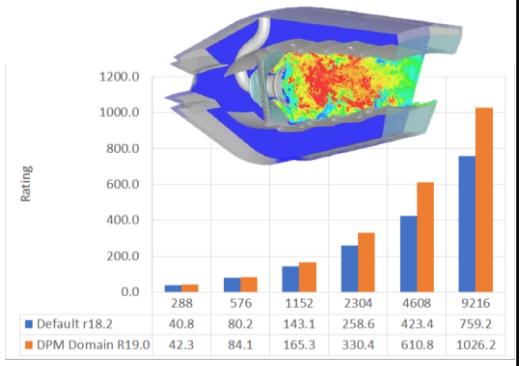
CHT cases now benefit from special load-balancing using the Model-Weighted Partitioning that is performed by default during re-partitioning



	72	144	288	576	1152	2304	3456
190	210.9		715.9	1284.6	2085.3	2017.8	
190 with Load Balancing	256.2	493.5	940.7	1509.6	2827.0	4453.8	5280.9
Improvement	21%		31%	18%	36%	121%	

DPM domain method enhanced

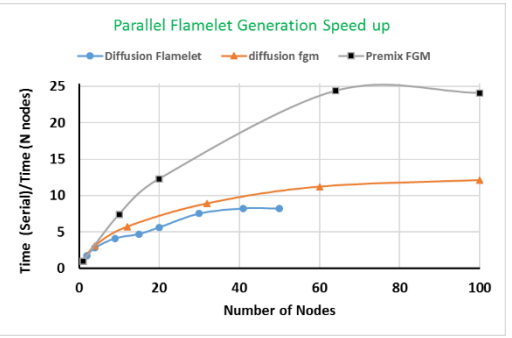
The DPM domain method performance and compatibility has been enhanced resulting 1.5X faster at 4608 cores



	288	576	1152	2304	4608	9216
Default r18.2	40.8	80.2	143.1	258.6	423.4	759.2
DPM Domain R19.0	42.3	84.1	165.3	330.4	610.8	1026.2

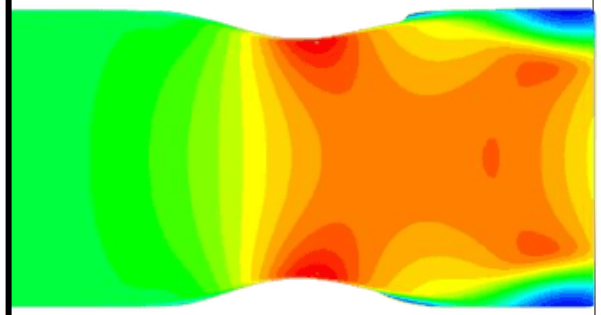
Parallel Flamelet Generation

The parallel flamelet generation speed up is now 10-20X faster for reacting flows simulations

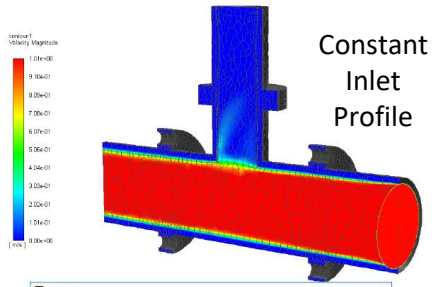


2018 : Increased Productivity Highlights

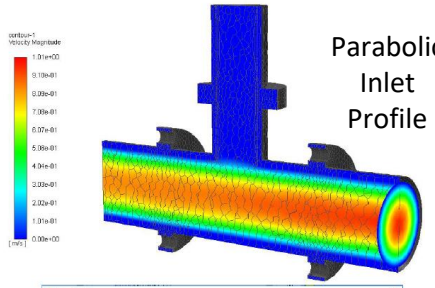
Expressions



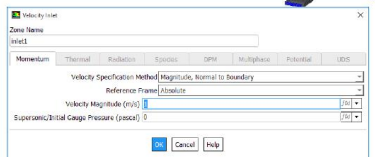
Expressions enable to specify inputs as functions of location, time or solution variables without the need of User Defined Functions (UDFs) compiled in C language

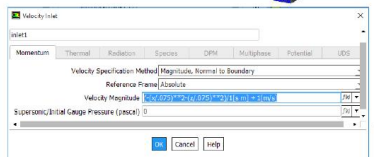


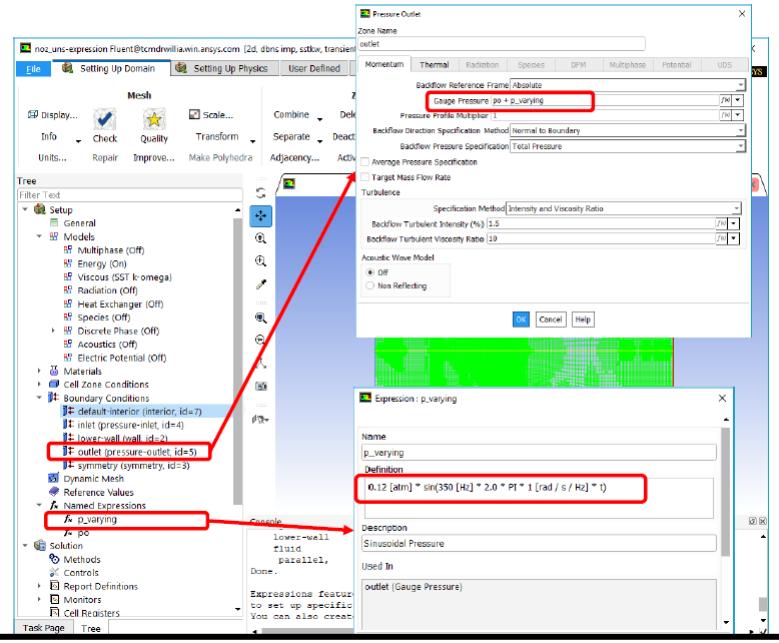
Constant Inlet Profile



Parabolic Inlet Profile



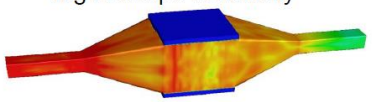




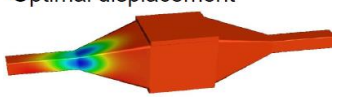
Surface Integrals as Adjoint Observables

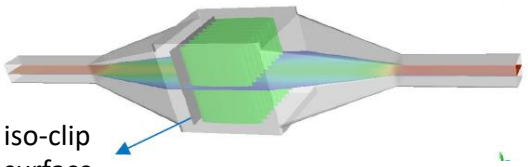
Iso-clip surface integrals (e.g., average-weight variance of the mass flux) can now be used as an observable in the Adjoint Solver

log10 shape sensitivity



Optimal displacement



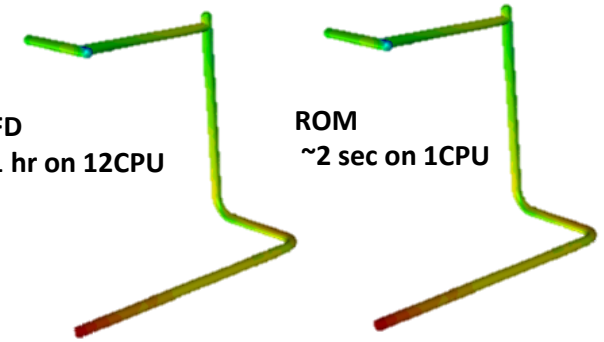


iso-clip surface

Observable	Before morphing	After morphing	Expected Change	Real Change
Velocity variance kg2/m4-s2	2112155.5	2066756.5	-2%	-2.15%

Reduced Order Model (ROM) Builder for CFD simulations

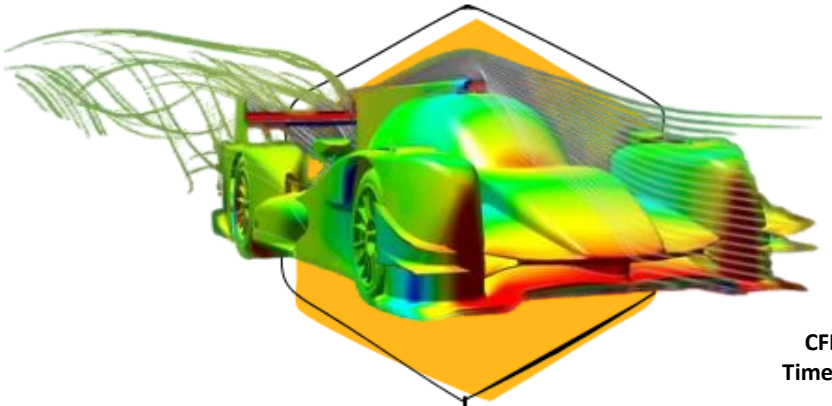
ROM Builder in Workbench to create steady-state Reduced Order Models for Cavitation, Corrosion Rate, Erosion Rate and Conjugate Heat Transfer



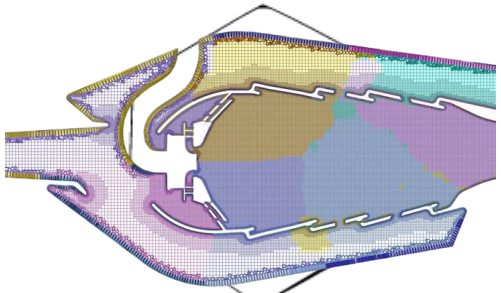
~1-3% difference in results

2019 : New Capabilities Highlights

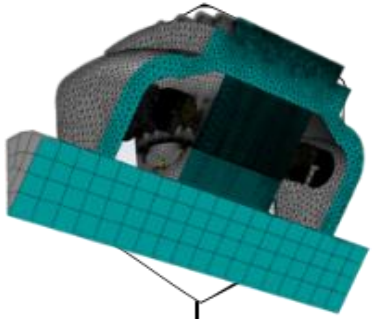
GEKO Turbulence Model



Parallel Mosaic™ Meshing



Coupled CFD-Electromagnetics for Electric Machines

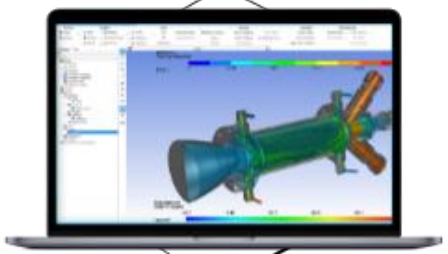


CFL Based Time Stepping

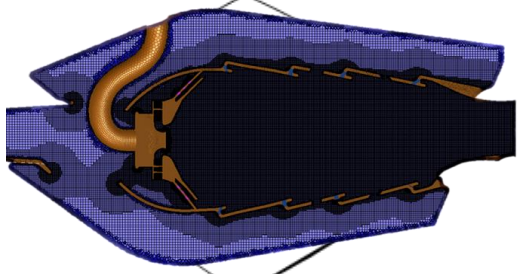
Pressure Based Solver Transient Speedup

BCD in Density Based Solver

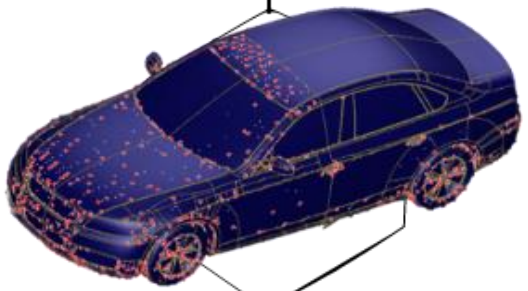
+ PERFORMANCE
+ ACCURACY
+



Single-Window Workflow Experience



Watertight Workflow for Clean Geometries

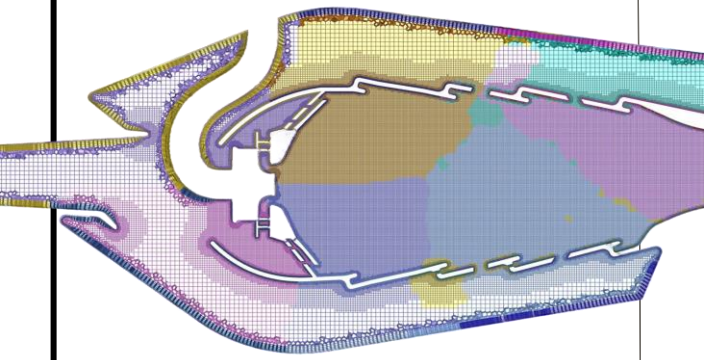


Fault Tolerant Workflow for Dirty Geometries

A revolution in user experience along with innovations and improvements for performance and accuracy

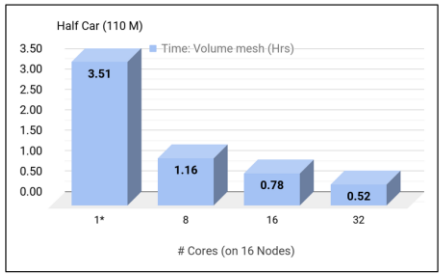
2019 : Increased Performance Highlights

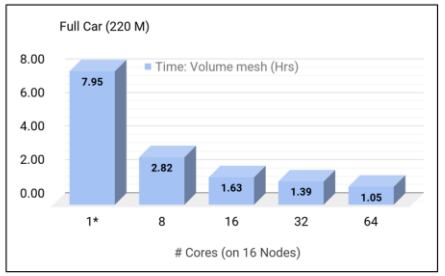
Parallel Mosaic™ Meshing

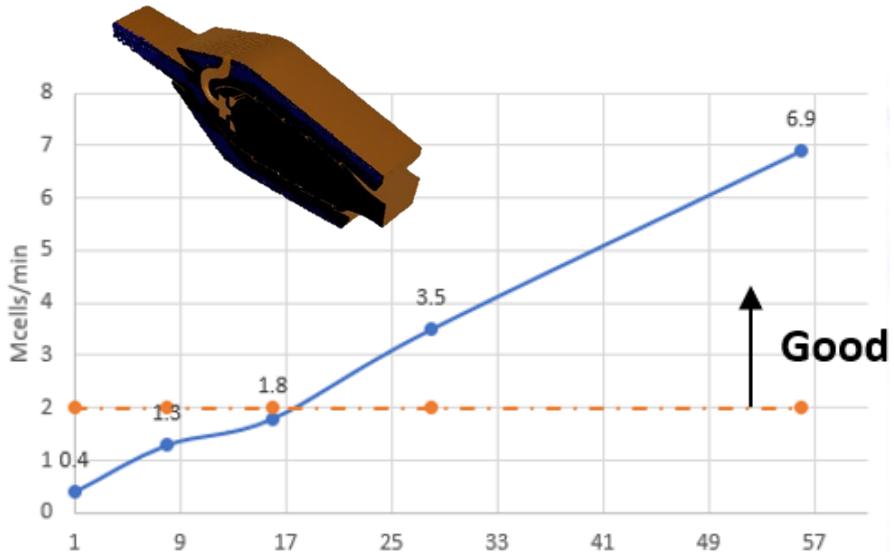


Mosaic Meshing ™ now provides parallel mesh generation with speedups up to 10X

- Particular benefit for meshes larger than 10-20Million cells
- Up to 2.5 Million cells/min with 16-way parallel
- Typical memory requirement: ~3GB / Million cells

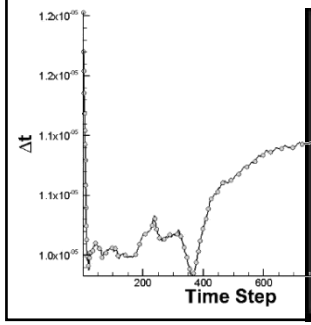


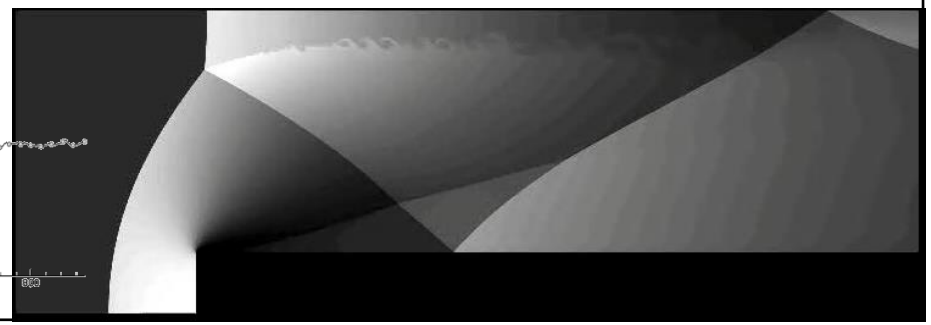




CFL based Time Stepping

The CFL based time stepping is particularly useful when mesh adaption is used to capture the fine details that are present in transient flows in a more efficient manner

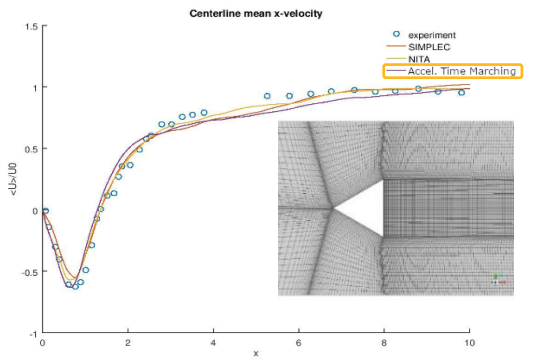




Pressure-based Solver Transient Speedup

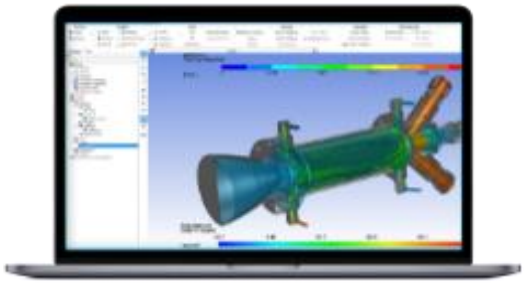
Accelerated Time Marching scheme for the NITA method, which further improves speed for LES turbulence cases without boundary layer meshes providing up to a 5X speed up compared to the SIMPLEC iterative method, with comparable accuracy

Solver	Time	Speed up
SIMPLEC	15 h	1
NITA	6 h	2.3
NITA - Accelerated Time Marching	2.6h	5.8



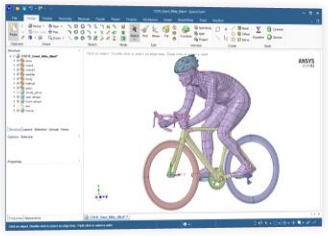
2019 : Increased Productivity Highlights

Single-Window Workflow Experience



Single window user experience providing a task-based workflow offering only relevant choices and options and prompting the user with best practices

- Tailored to the task at hand
- Best practices ensure good results
- Offers only relevant choices
- Automates repetitive activities
- Full access to Fluent's power and flexibility



Space Claim

Cyclist geometry courtesy of Centre for Sports Engineering Research, Sheffield Hallam University

Single-Window Task Based Workflow

Geometry Part Management	Identify Regions	**Mesh Controls**	Volume Mesh	Setup & Solve	Post

Watertight Meshing Workflow (WTM)

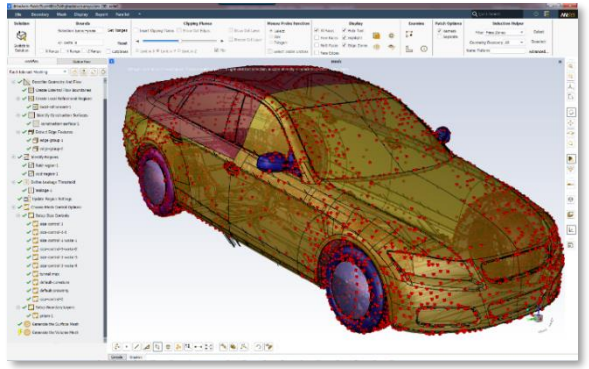
Task-Based Fluent Meshing Workflow for clean CADs/Geometries to create high-quality meshes with minimal training and effort thanks to intelligence and automation behind the scenes with up to 50% less hand on time.

Guided workflow based on meaningful tasks with status, warnings, etc. User can choose to modify task-list and save as a custom workflow.

Fault Tolerant Meshing Workflow (FTM)

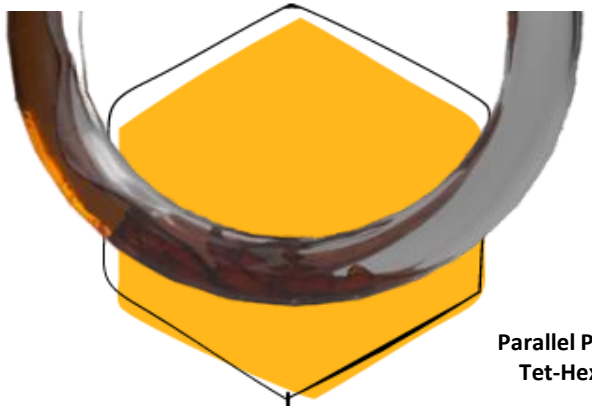
Task-Based Fluent Meshing Workflow capable to generate meshes for even dirtiest CADs/Geometries with a dramatic 2x speedup.

The streamlined wrapper-based workflow can create high-quality meshes sealing leakages and holes (typical of external aero simulations, underhood / installation simulations, etc.) with an intuitive drag and drop part management.

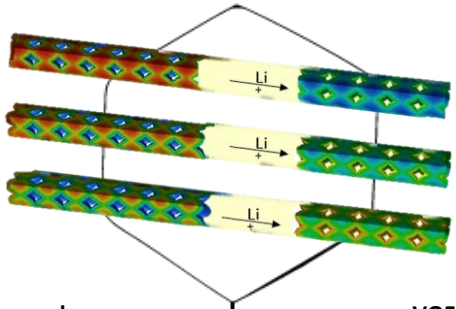


2020 : New Capabilities Highlights

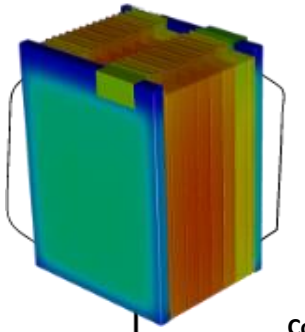
AIAD Regime Transition



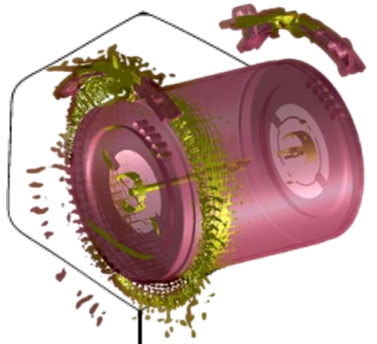
3D Battery Electrochemistry



Capacity Fade and Battery Life Effects



Sliding Mesh Performance Speedup



Parallel Poly-Hexcore and Tet-Hexcore Speedup

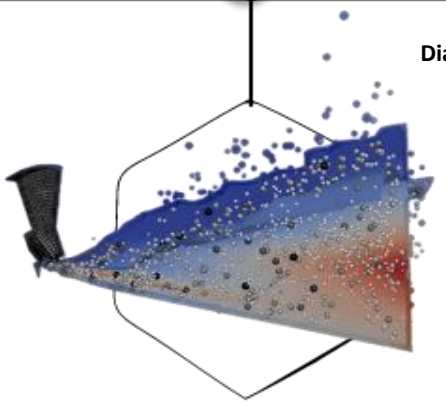
VOF Robustness Improvements

Coupled Solver Performance Improvements

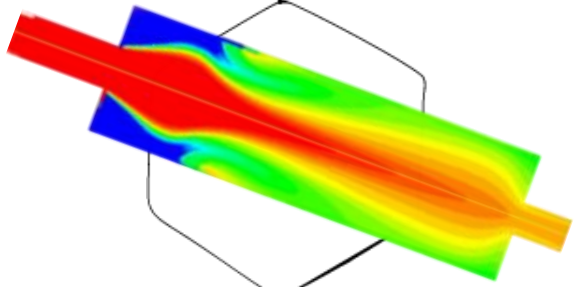
+ PERFORMANCE
+ ACCURACY
+

Diagnostics of Expressions

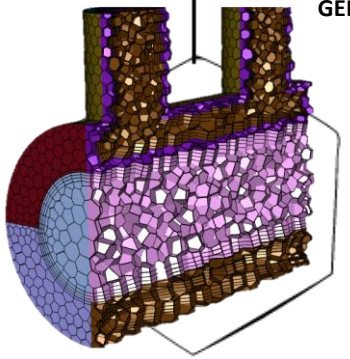
GEKO Turbulence Model in Adjoint Solver



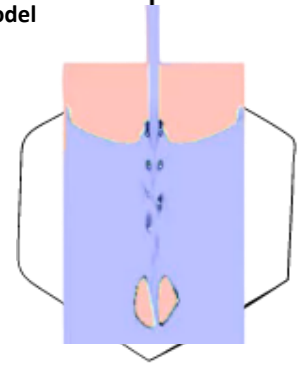
Post-processing Enhancements



Nonadiabatic FGM with Premixed Flamelets



Full Boundary Layer Flexibility



GENTOP Transition Model

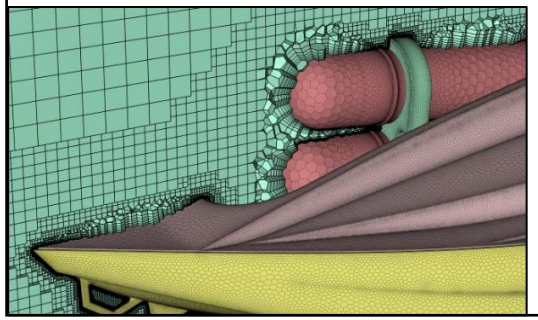
Focusing on performance while delivering innovations for multi-phase modelling and battery simulations

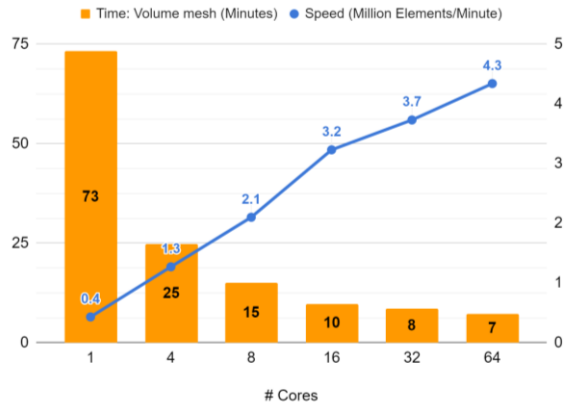
Ansys

2020 : Increased Performance Highlights

Parallel Poly-Hexcore Speedup

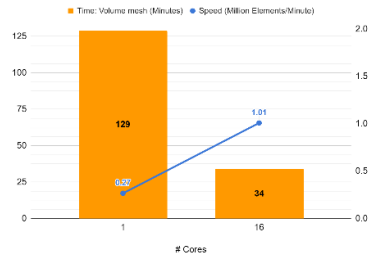
Parallel Poly-Hexcore quality, robustness, scalability improvements with overall speedup in mesh generation up to 10X

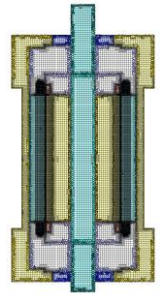




Sequential Meshing

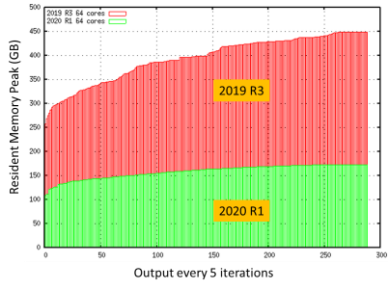
Multi-object distributed parallel volume sequential meshing with overall speedup up to 3.8x for industrial electric motor





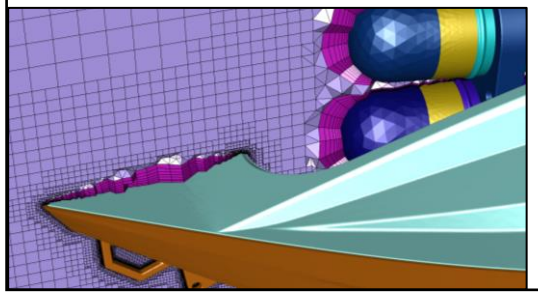
Overset Mesh Enhancements

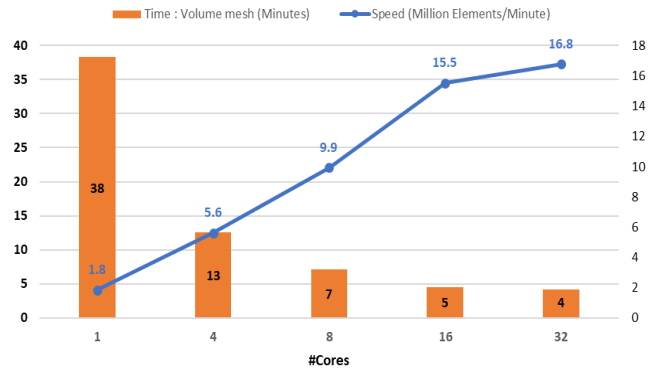
New cell exchange model is used with improved parallel memory efficiency for overset mesh simulations



Parallel Tet Hexcore Speedup

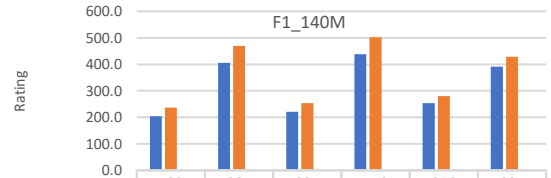
Parallel Poly-Hexcore quality, robustness, scalability improvements with overall speedup in mesh generation up to 9.5X





Enhanced Coupled Solver Performance

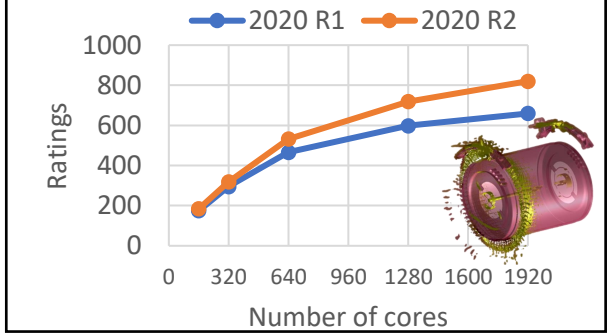
Intel worked with Fluent adding an ILU smoother to MKL with overall speedup up to 15%



	192	384	224	448	256	384
	CLX 8260		CLX 8280		EPYC 7542	
Baseline	203.6	405.0	220.4	438.0	252.9	390.7
MKL ILU	236.9	469.9	253.7	502.1	280.3	427.9
Enhancement	16.4%	16.0%	15.1%	14.6%	10.8%	9.5%

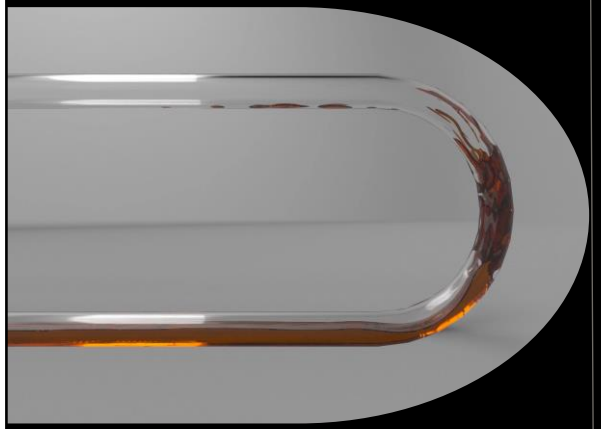
Sliding mesh enhancement

Wall distance calculation has been enhanced with resulting speed-up of sliding meshes up to 34%



2020 : Increased Accuracy Highlights

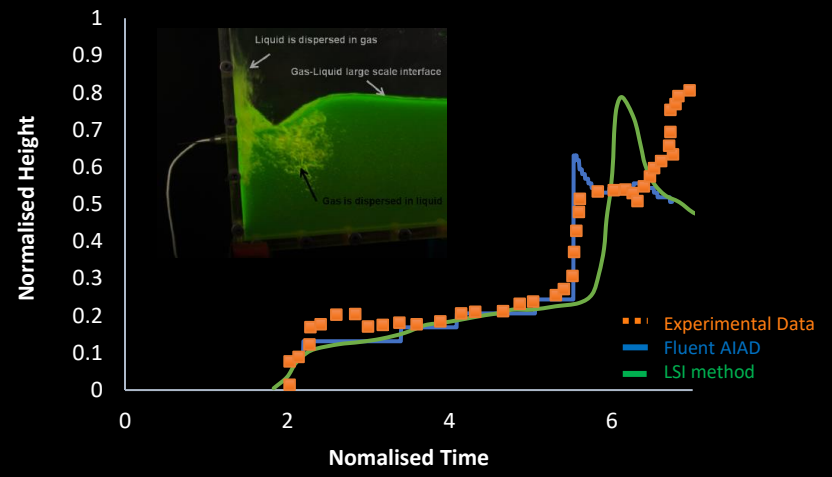
AIAD Regime Transition



Eulerian multi-phase transition method based on Algebraic Interfacial Area Density (AIAD) approach

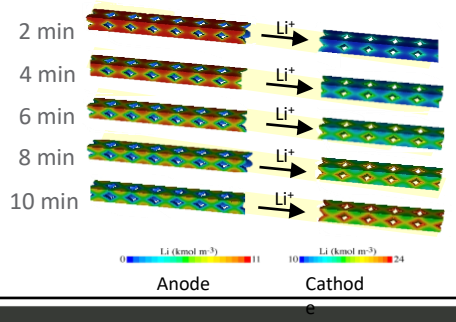
- Suitable for a range of applications including loss of coolant scenarios in pressurized-water reactors
- Incorporation of sub-grid turbulence contribution for accuracy
- Mass transfer mechanisms to account for entrainment/absorption
- With population balance can simulate dispersed phase size distributions

Fluent AIAD produces better match to experimental results



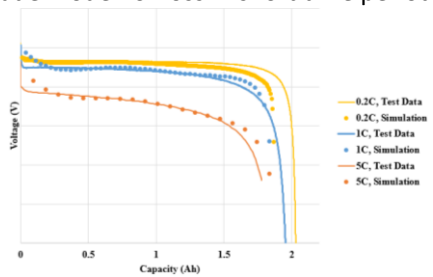
3D Battery Electrochemistry

Transient simulation of Li-ion transport during battery discharging/charging



Capacity Fade and Battery Life

Life model for battery's capacity loss during a long time period and capacity fade model for loss in short time period



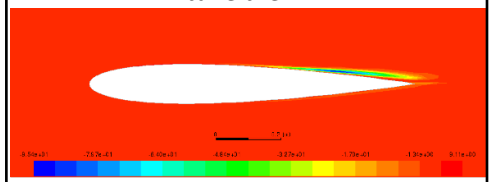
GENTOP Transition Model

GENTOP (Generalized TwO Phase) model includes additional phase that can behave either as continuous or dispersed based on phase volume fraction critical bubble diameter.

GENTOP phase (resolved continuous/hybrid gas) flowing alongside the dispersed modeled phase

Adjoint GEKO Turbulence Model

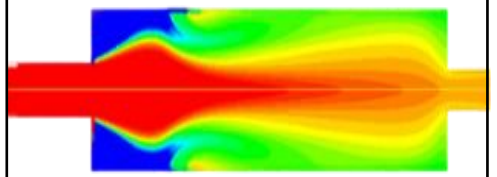
GEKO sensitivity in adjoint solver shows how solution is affected by changes in GEKO parameters and help tune them



Sensitivity to GEKO Blending Function

Nonadiabatic flamelets FGM

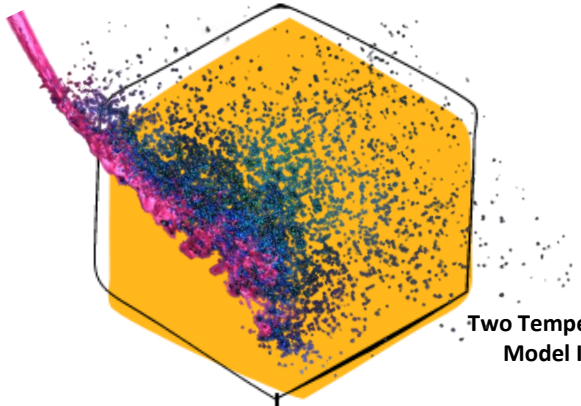
Improves accuracy in cases where heat loss/gain due to presence of cold walls, participating media can't be neglected



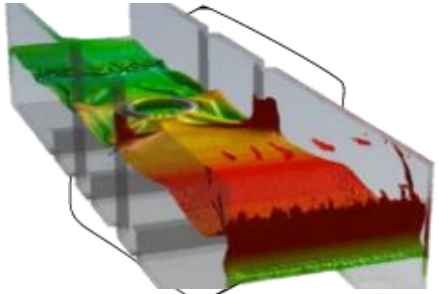
Premixed Flamelets

2021 : New Capabilities Highlights

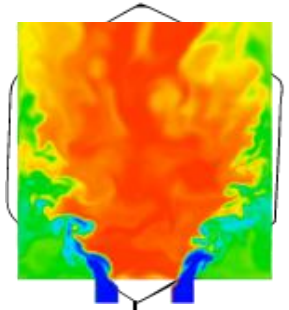
Bidirectional VOF ⇔ DPM ⇔ EWF



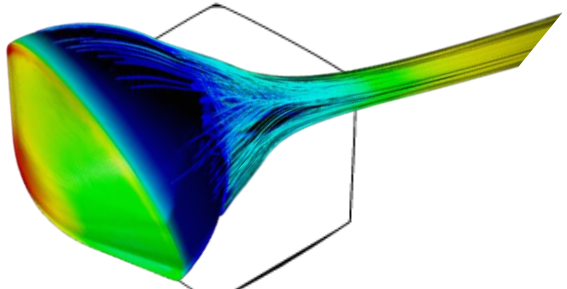
Solver and Meshing Speedup for Overset Meshes



Strained FGM Combustion Model



Surface Ablation Model



Two Temperature Equations
Model Improvements

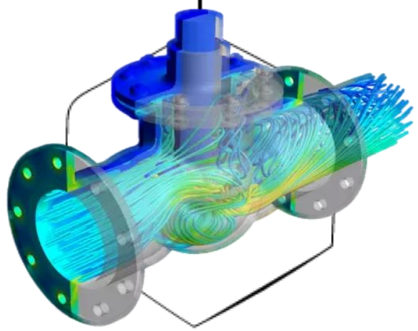
High-speed
Numerics Improvements

+ PERFORMANCE
+ ACCURACY
+

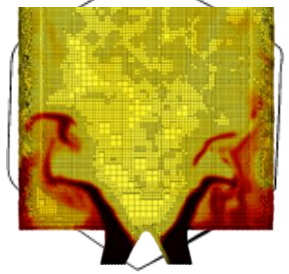
Dark Theme

Embedded Windows

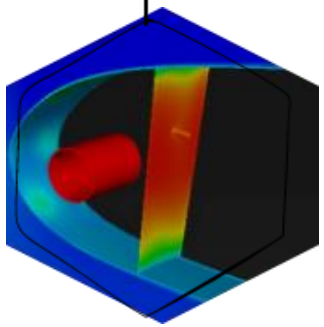
Improved Treatment of Reaction Sources in Density Based Solver



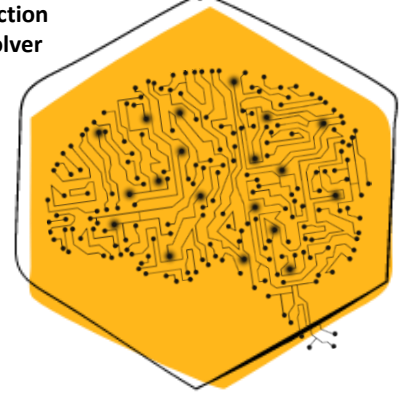
Ansys CFD Pro



Best Practices Embedded in Mesh Adaption



Dedicated Icing Workspace



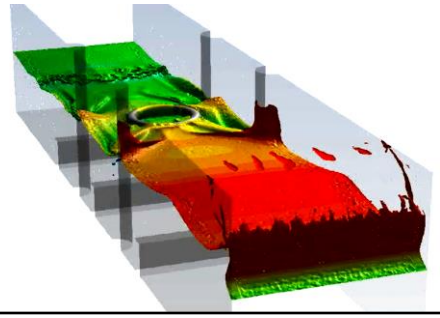
AI/ML Turbulence Tuning

Delivering features that enable easier workflows as well as extending the breadth and depth of solver capabilities

2021 : Increased Performance Highlights

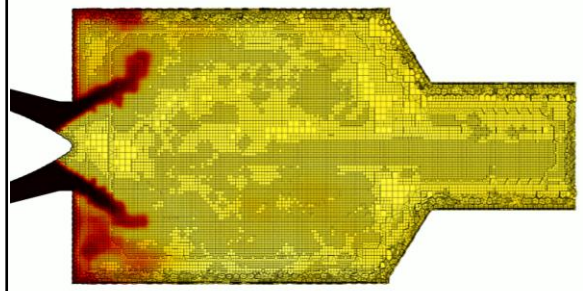
Overset Meshing and Solving Speedup

PUMA-based dynamic mesh adaption technique extended for overset problems providing 3X speedup for industrial gearboxes using VOF



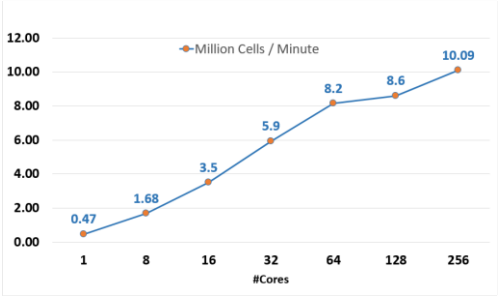
Best Practices in Mesh Adaption

Best practices embedded in the automated mesh adaption setup for combustion and multi-phase applications with cell count reductions up to 70%



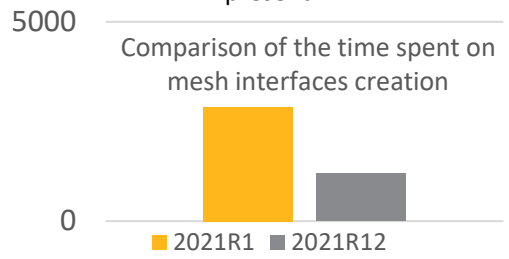
Mosaic Meshing Improvements

Parallel poly and poly-hexcore now fully supported for WTM and FTM and parallel enhancements provide speedup up to 21x



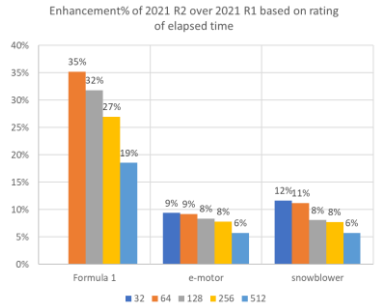
Mesh Interfaces Performance & Robustness

Improved gradient method at fluid-fluid interfaces with more robustness and accuracy for poorly matching interfaces when many are present



Sliding Mesh Performance

Further enhancement up to 35% in sliding mesh cases at each time step after the mesh slides



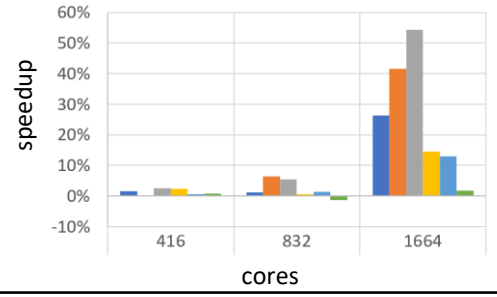
Adjoint Speed and Memory Improvements

Partial coupling adjoint solver: the adjoint continuity and momentum equations solved coupled while other equations solved segregated with up to 2x speedup

Decouple the shape sensitivity calculation from the AMG allocation, which reduces the memory cost of postprocessing and design tool calculation considerably up to 30%

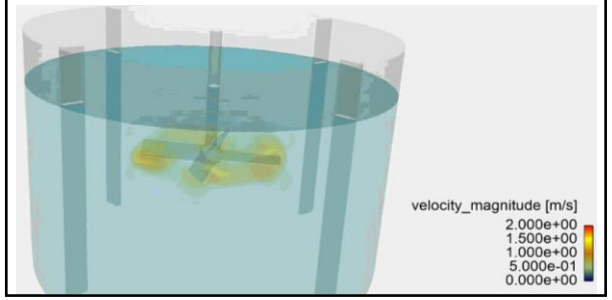
Job Scheduler / MPI Support

Added SLURM support both at command line and launcher and INTEL MPI flags for parallel scalability resulting in speedups up to 54%

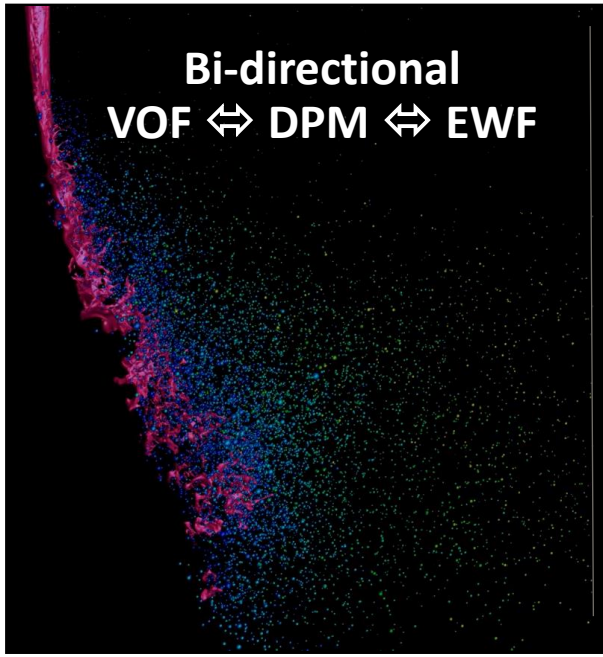


Instability Detector Improvements

Speed-up of Hybrid NITA while using instability detector. 20% reduction in wall-clock time for stirred tank vortex case.



2021 : Increased Accuracy Highlights



Bi-directional
VOF ⇔ DPM ⇔ EWF

Enables the transition from DPM to VOF (and viceversa) for multi-phase simulations

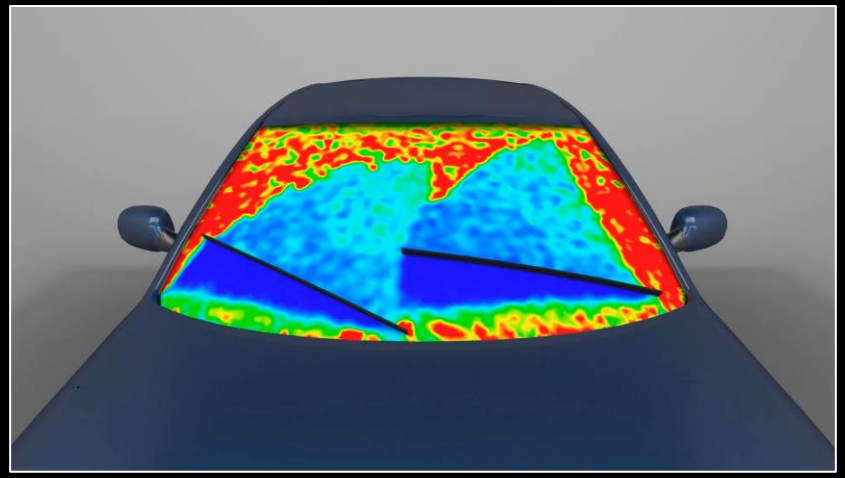
Ansys Fluent 19.0 introduced model transition **VOF -> DPM**

Distinct droplets in the VOF simulation are detected and replaced by DPM (i.e., mass-point) particles, and mesh is coarsened accordingly

Ansys 2021 R2 makes this transition bi-directional with DPM -> VOF

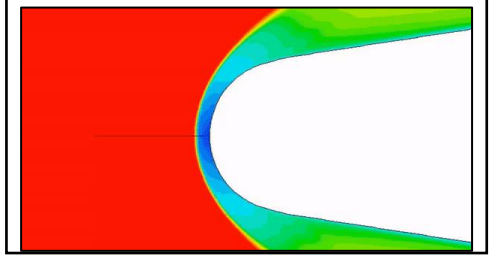
DPM particles falling onto a free liquid surface transition back into VOF formulation; mass-point particle replaced by mesh-resolved VOF liquid

adding also support for **EWF <-> VOF**



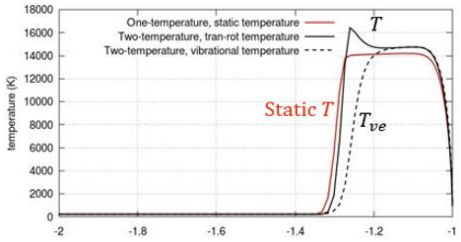
Surface Ablation Model

Built-in workflow for wall recession improves accuracy of hypersonic re-entry and rocket engines simulation



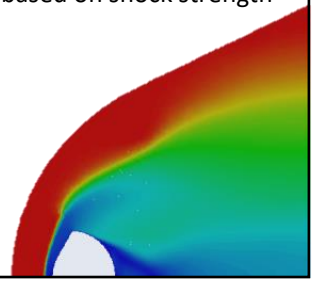
Two Temperature Equation

Improve prediction of high-speed aerothermodynamics with additional equation for electro-vibrational energy



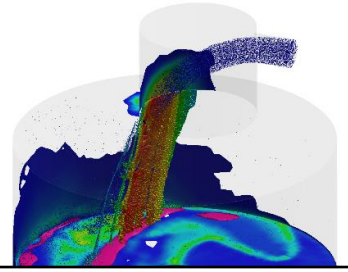
Adaptive High-speed Numerics

Adaptive method detects shocks and applies numeric correction to stabilize the solution based on shock strength



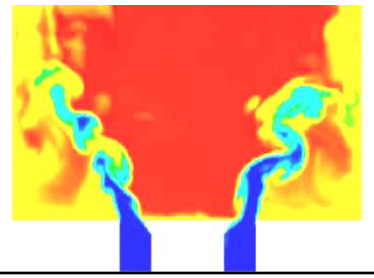
Adaptive Transition Time Stepping

Time step size gets updated from all cells until the VOF interface is detected for better model transition



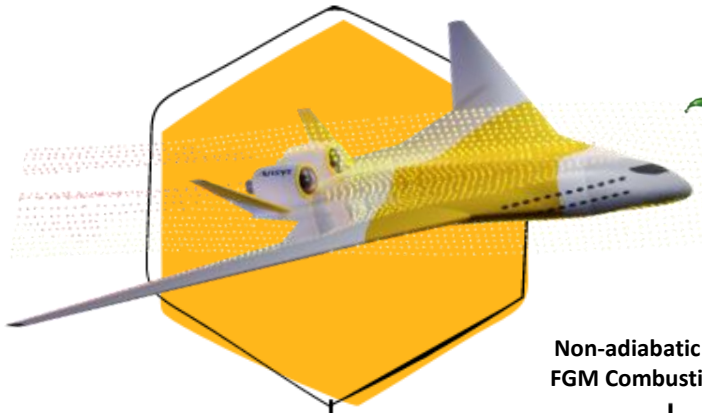
Strained FGM Combustion

Addition of a strained FMG model for better flame stability particularly at lean conditions like lean blow out

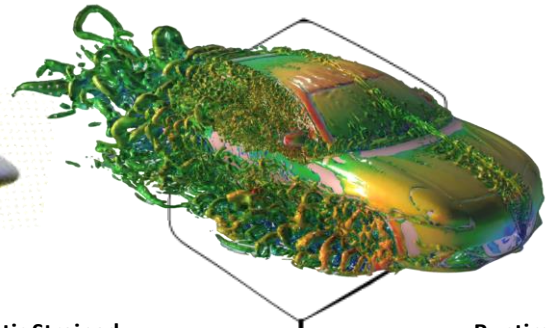


2022 : New Capabilities Highlights

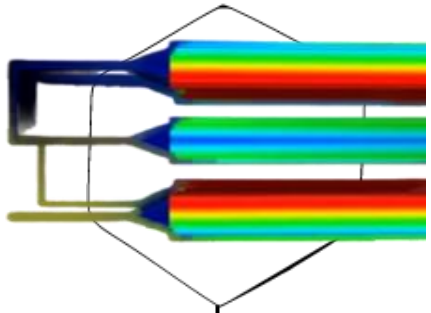
Live GX Multi-GPU Solver



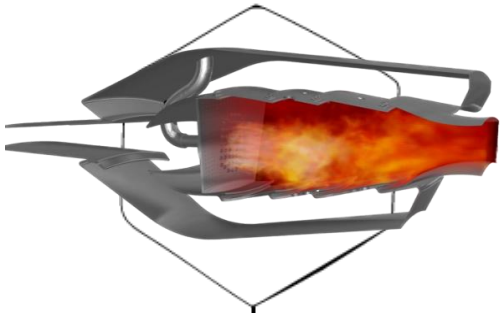
Sound Composition



Electrochemistry Battery Swelling



Hydrogen Combustion Validated Models



Non-adiabatic Strained FGM Combustion Model

Runtime Discrete Fourier Transform

Green Hydrogen Production Model

Hypersonic Partially Catalytic Wall BC

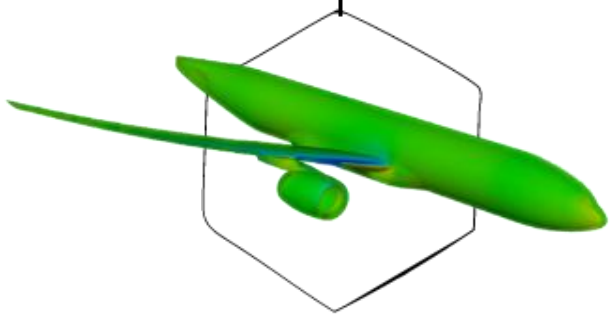
+ PERFORMANCE

+ ACCURACY

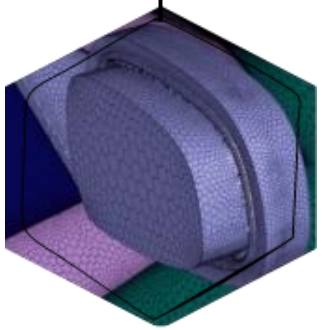
+

Acoustic Sponge Layers

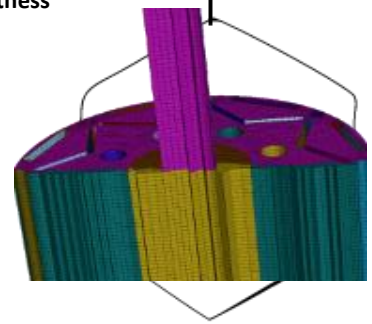
Improved High-Speed Flow Robustness



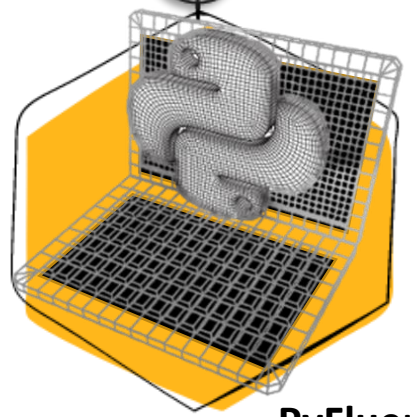
Dedicated Aerospace Workspace



Part Replacement in FTM



Automated Structured Meshing in WTM



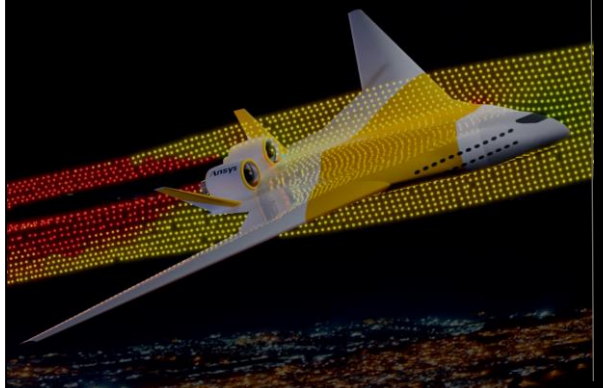
PyFluent

Major GPU technology advancements and open source assess to more sustainable CFD simulations

Ansys

2022 : Increased Performance Highlights

Live-GX Multi-GPU Solver



Fully native multi-GPU Fluent solver to accelerate steady-state and transient simulations

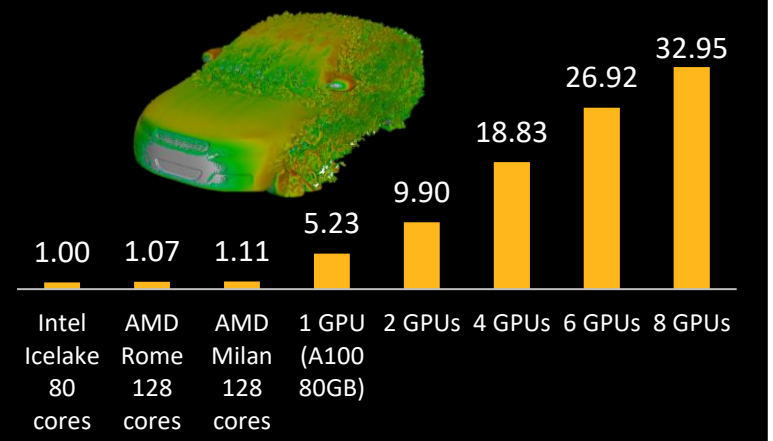
Capabilities

Single / multi-GPU	Materials : constant & polynomial
All mesh types and interfaces	Majority of turbulence models
Steady & transient scale resolving	Solid conduction and CHT
Subsonic compressible flows	Moving Walls and MRF
Ideal gas, Bousinnesq, Constant ρ	Porous Media

Target Applications

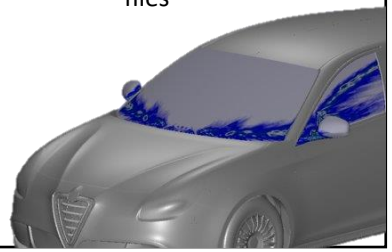
External Aero | Internal Flows | CHT

- **6 typical GPUs > 2,000+ CPU cores**
- **7x cheaper hardware purchase cost**
- **4x lower power consumption**



Runtime DFT

Evaluate at runtime the Discrete Fourier Transform of any variable in the entire domain without I/O of large files



Many Zones Cases Performance

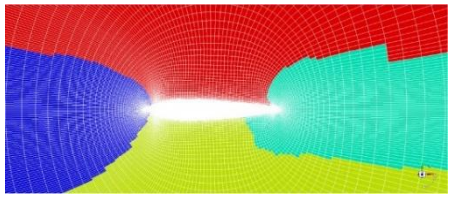
Large improvements in UI and I/O performance when 10k's of zones are present

Max speedup :

Mesh Display (73%)
Contour Display (83%)
Vector Display (80%)

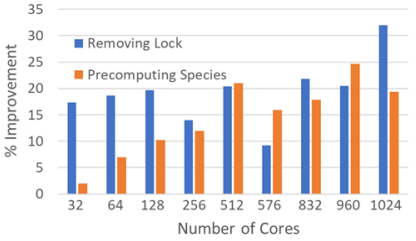
Partitioning Enhancements

Robustness improvements for extruded 3D cases and cases with very high-aspect ratio cells



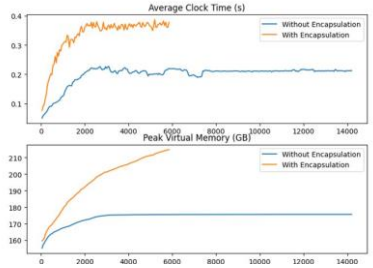
Improved Particle Tracking

Significant reduction in DPM solve time with improved robustness and accuracy in various scenarios



PUMA Performance

PUMA Improved performance and reduced memory



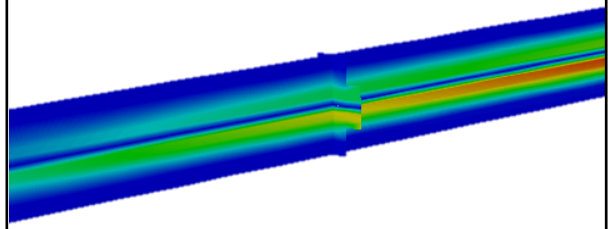
2022 : Increased Accuracy Highlights

Electrochemistry based Battery Swelling

Accurately predict battery swelling during charging due to electrochemistry, pressure and swell-related material properties

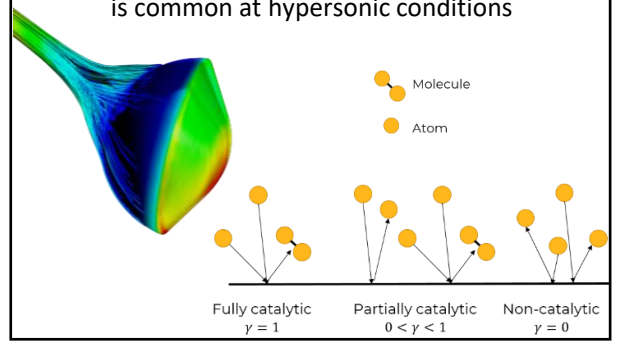
Green Hydrogen Production Model

New method to model green hydrogen production through electrolysis with the proton exchange membrane (PEM) model



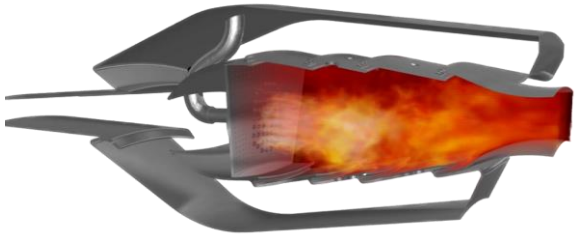
Hypersonic Partially Catalytic BC

Boundary condition that accounts for recombination of atoms near vehicle walls, which is common at hypersonic conditions



Hydrogen Validated Combustion Models

Validated hydrogen and hydrogen blends combustion models for accurate prediction of knock, detonation, flame stability and flashback

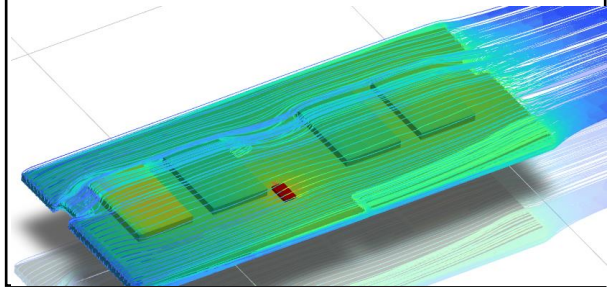


Acoustic Sponge Layer

New Sponge Layer option to damp acoustic pressure fluctuations at inlet/outlet boundaries reducing the cell count and simplifying setup

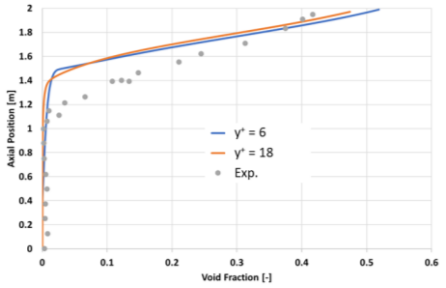
Advanced Electronics Cooling

A new printed circuit board (PCB) add on model enables to bring Ansys Icepak settings into Fluent to perform advanced electronics cooling analyses



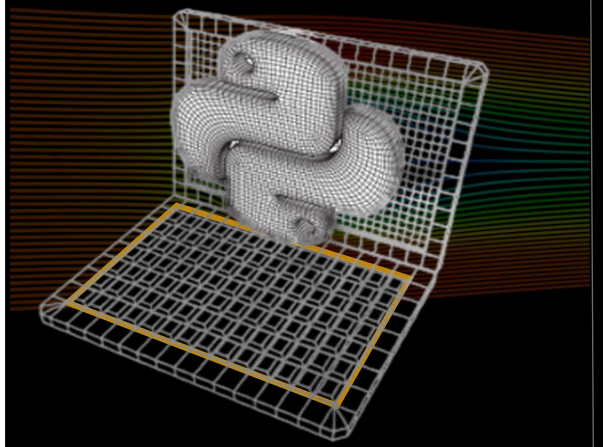
Improved Wall Boiling Accuracy

New model to accurately predict liquid temperature near wall independently of the mesh resolution reducing the pre-processing time



2022 : Increased Productivity Highlights

PyFluent



Open-source access to all Ansys Fluent commands via Python scripting

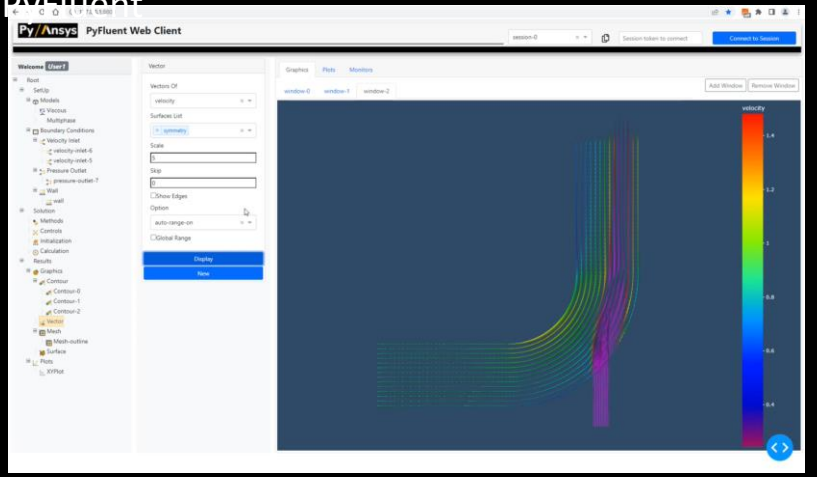
Capabilities

- Use Fluent within, or alongside, any other Python environment
- Access all Fluent TUI commands for meshing, solving, post
- Extract flow field data and use with standard Python tools
- TUI API mirroring the Ansys Fluent Text User Interface
- Settings API aligned with the Fluent object tree

Target Applications

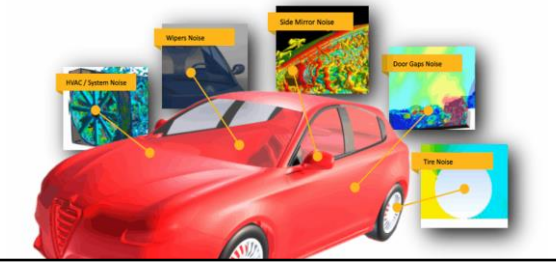
Flexible Automation | Flexible Distribution | Technology Integration

Craft a customized user experience with PyFluent



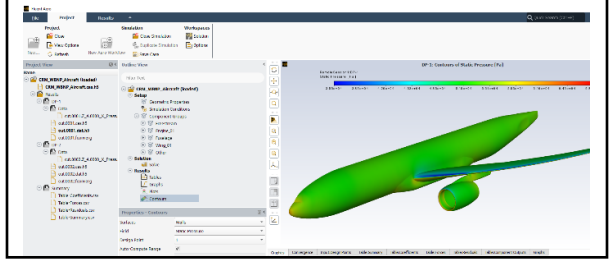
Sound Composition

Perform sound composition directly in Ansys Fluent and transfer acoustic files to Ansys Sound for in depth acoustics analyses



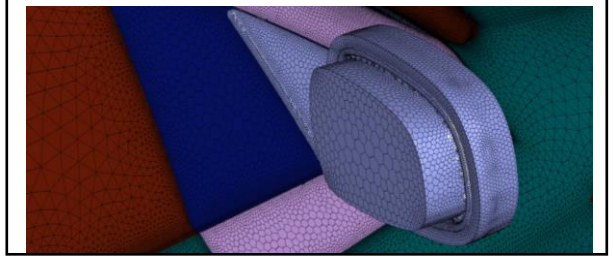
Dedicated Aerospace Workflow

A dedicated aerospace workspace tailors the Fluent user interface to external aerodynamics simulations streamlining pre- to post processing



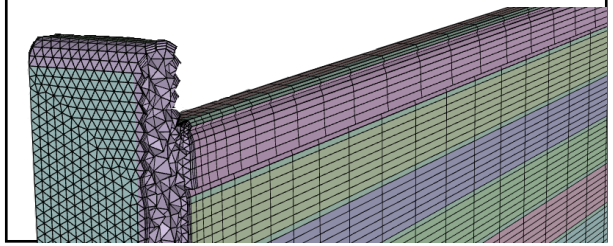
Part Replacement in FTM

Add, remove or replace geometry objects of your CAD without having to re-mesh (re-wrap) the complete model in Fault Tolerant Meshing



Automated Structured Meshing in WTM

Automated and efficient structured meshing (including mix multizone and unstructured meshing) is now available in Watertight Meshing



2023R2 Fluids Highlights

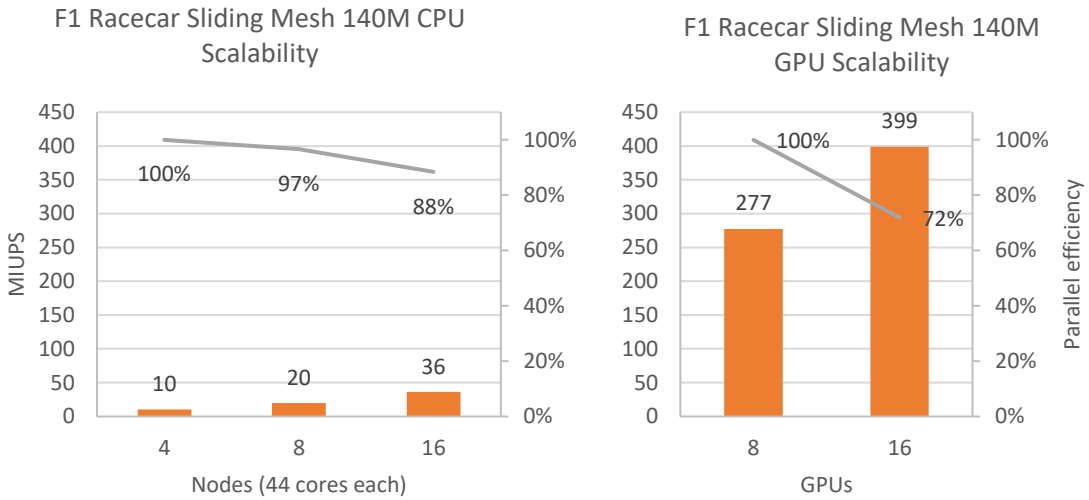
Ansys

GPU Solver: Sliding interfaces

Model rotating and stationary components

- An efficient and robust sliding mesh algorithm
- Enhanced robustness of dramatically changing intersections
- Test case: F1 140M rotating wheels, URANS SST

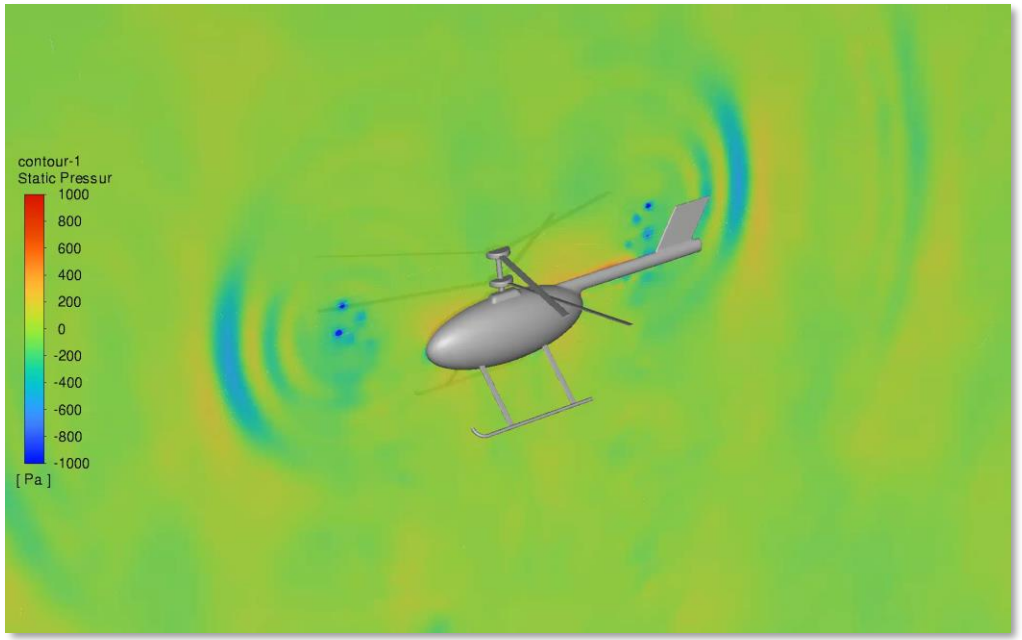
Intel Platinum 8168 dual-socket cluster
Nvidia A100 40GB 8-GPU per node

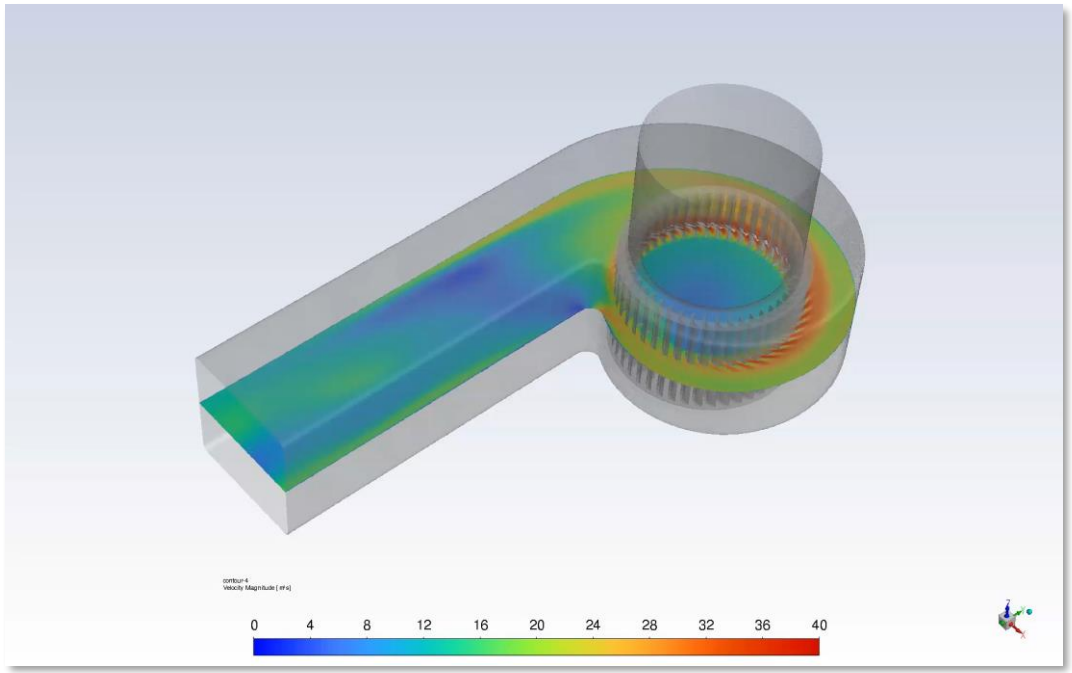


1 GPU ~ 16 CPU nodes (704 cores)
With 8 GPUs, solve 1000 steps in 40 minutes!

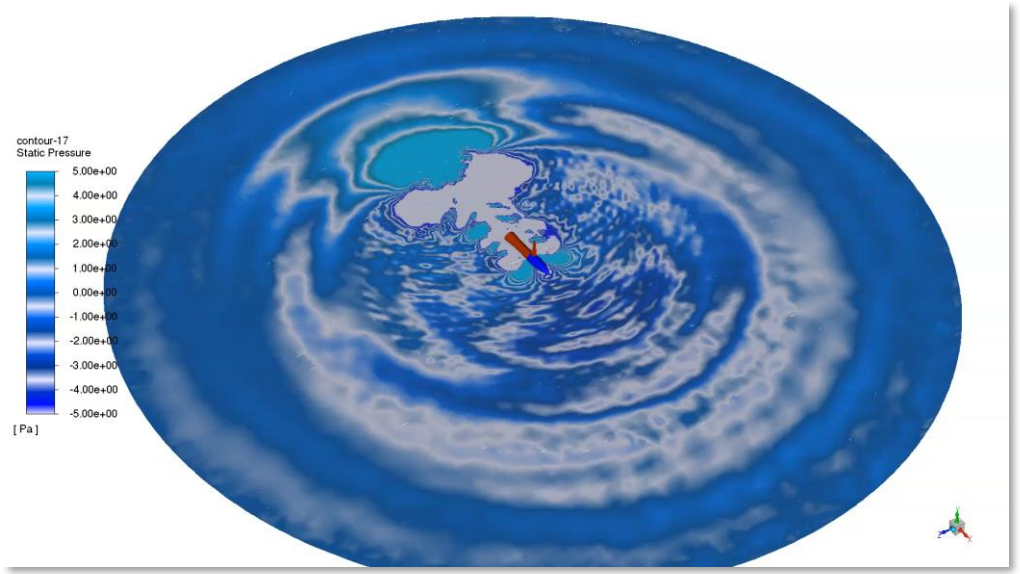
GPU Solver: Compressibility

The Fluent GPU solver now supports low-speed compressibility in addition to incompressible flow





Centrifugal blower with compressible flow and sliding mesh

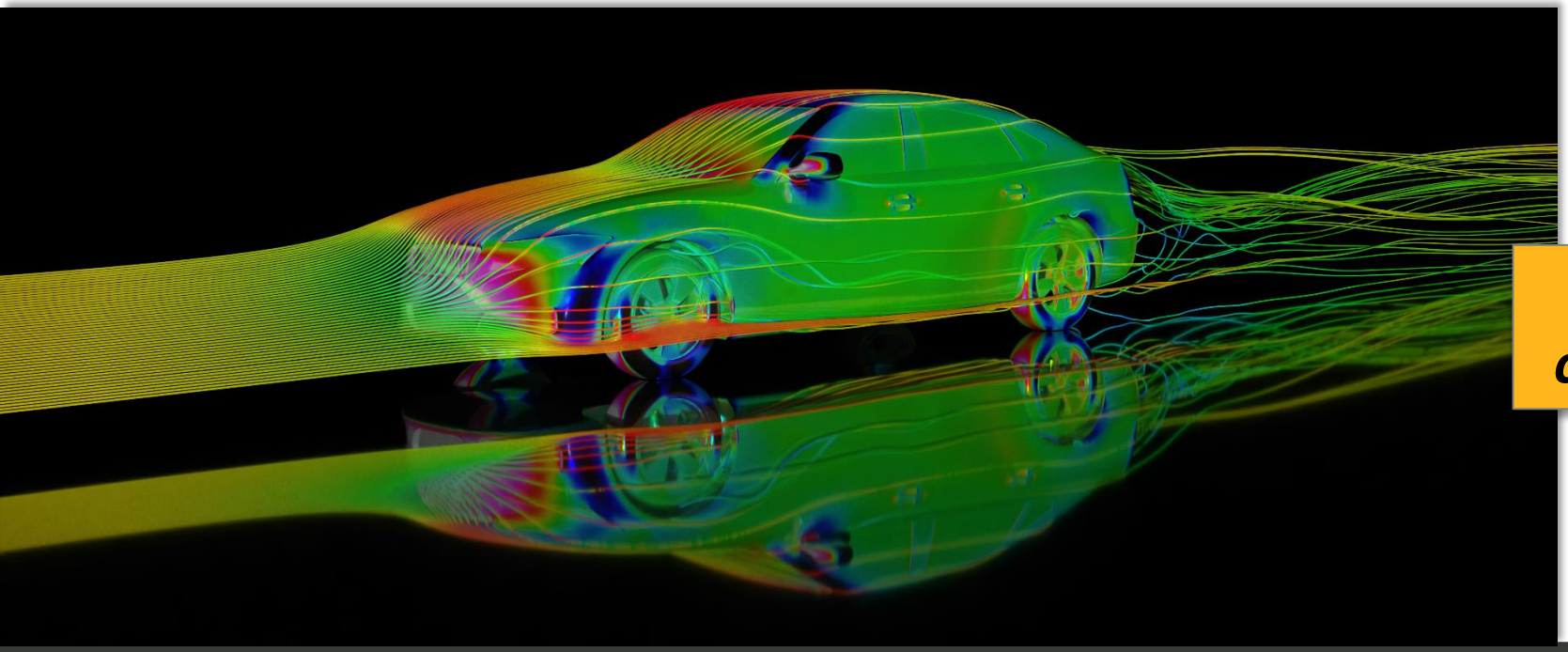


GPU Solver: Turbulence

- Enhanced robustness
- RANS equation solution frequency controls for SBES

400 steps	Timing standard	Timing when updating k+ω every 5th step only
CPU (Milan 128 cores)	11792.5 [29.5 s/dt]	6533.2 [16.3 s/dt]
GPU (DGX 8 x A100)	357.6 [0.9 s/dt]	**255.7 [0.6 s/dt]**
Speedup	32.8 (1 GPU ~ 512 CPU cores)	27.2 (1 GPU ~ 435 CPU cores)

Auto external aero case, 156M with SBES, DGX: 8 x A100 80GB, enables less than 1 day turn-around

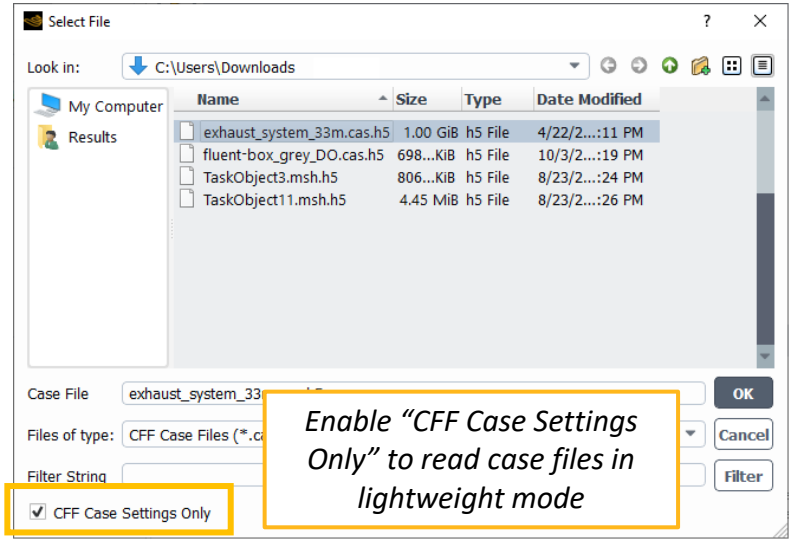


0.6 s per time step for 156M cell external aero calculation!

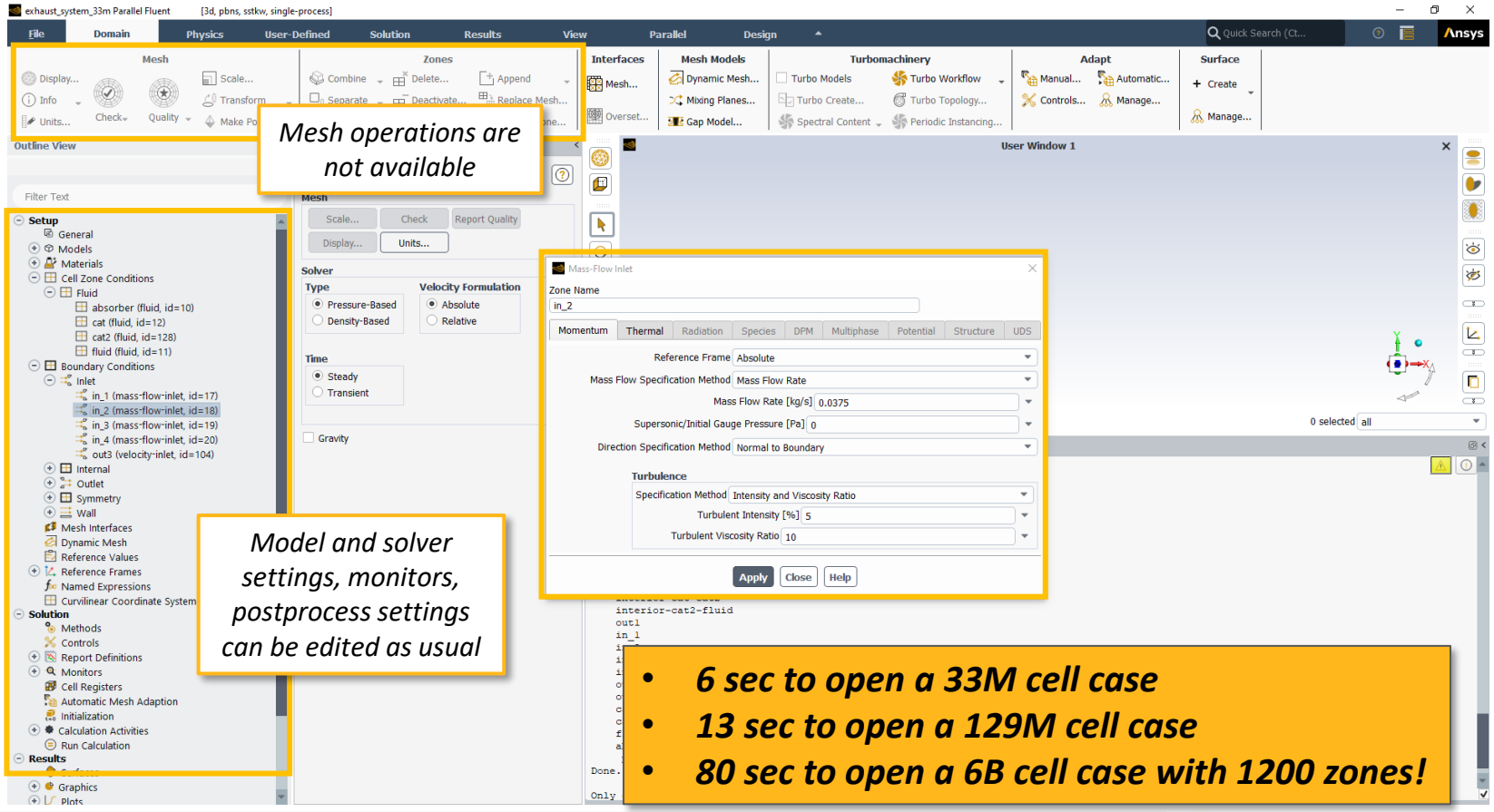
Lightweight Setup Mode

- Quickly view and edit your setup without loading the mesh
- Read/write are fast with minimal memory usage
- Support GPU Solver restart

- Supports command-line option `-lite`
- TUI command: `file read-case-settings test.cas.h5`



Enable "CFF Case Settings Only" to read case files in lightweight mode



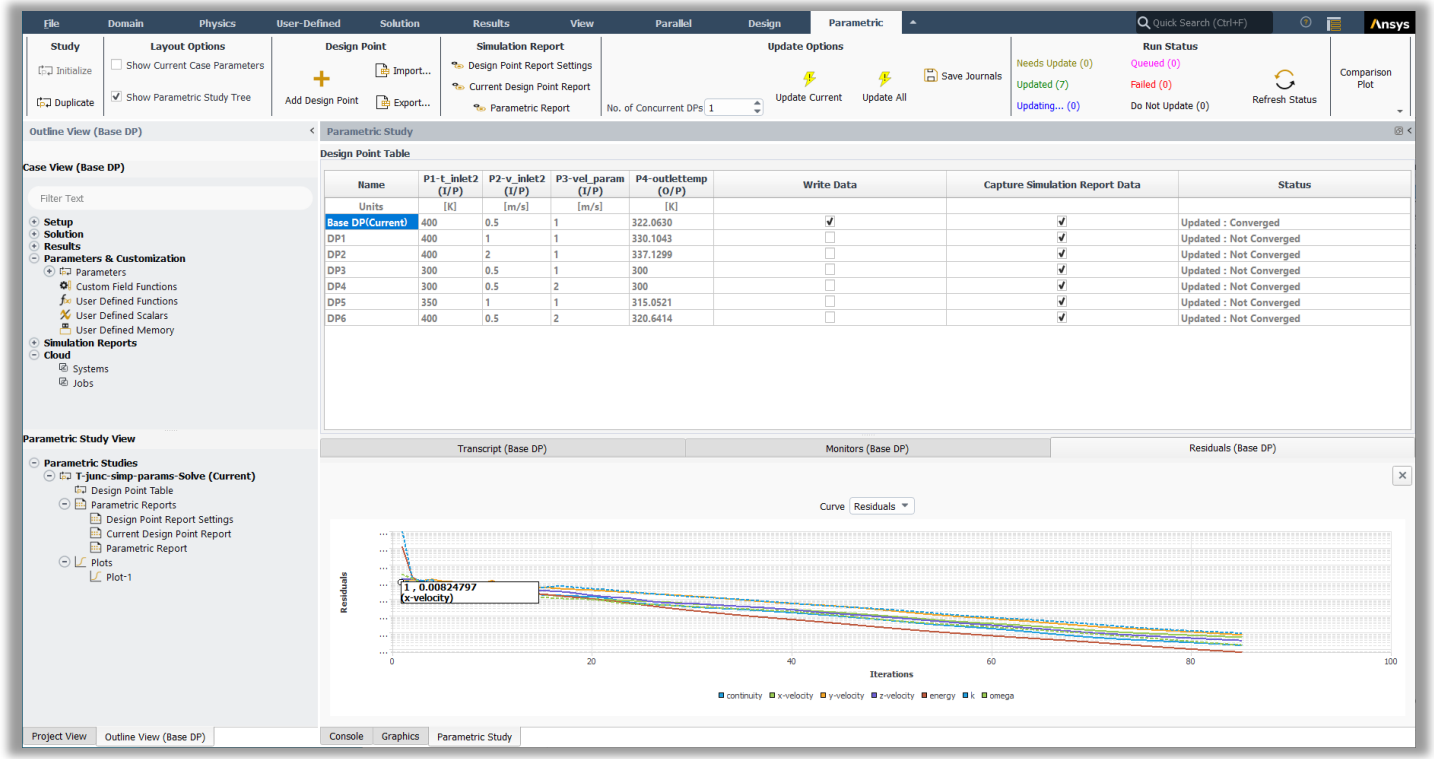
Mesh operations are not available

Model and solver settings, monitors, postprocess settings can be edited as usual

- ***6 sec to open a 33M cell case***
- ***13 sec to open a 129M cell case***
- ***80 sec to open a 6B cell case with 1200 zones!***

Parametric Workflows & Simulation Reports

- New Capabilities / Technologies
  - Sequential Design Point updates with GPU Solver
  - Report Export in Power Point Format
- Enhancements
  - Open Parametric Study in optiSLang
  - Initialize from Previous Design Point
  - Automatic Mesh Morph update
- Continuous Improvement
  - Table Validation and Column Sizing
  - Performance improvements



DBNS Solver and High-Speed Flow Applications

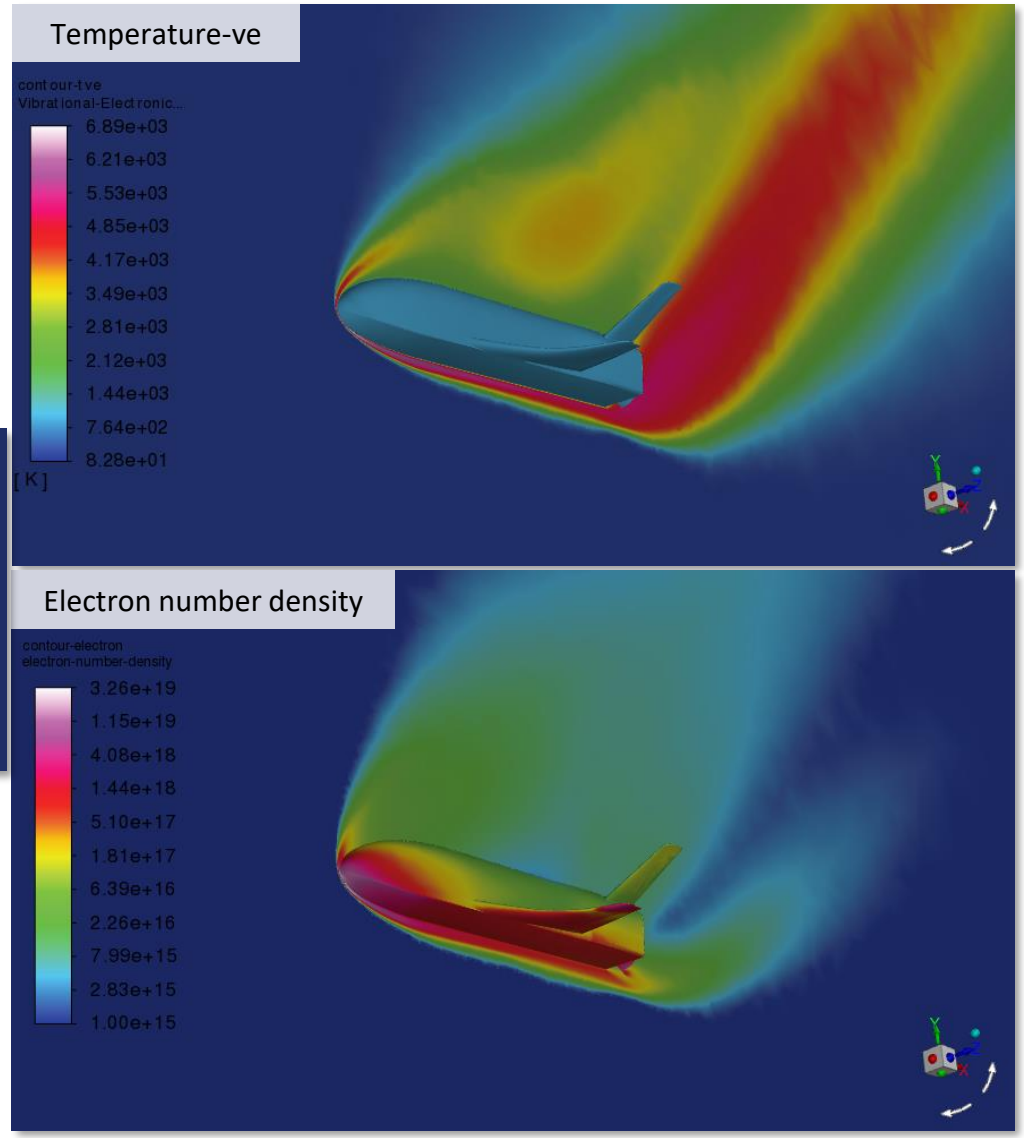
- **Two-Temperature Model**: you can now account for **weak Ionization** for improved solution fidelity at hypersonic speeds
  - At very high-speed, plasma starts to form around hypersonic vehicle
  - Crucial for predicting **communication blackout or degradation**

- **Built-In Gupta transport properties** for **one-Temperature** model (**Beta**)
  - To improve solution prediction at lower hypersonic speed when one-Temperature model is used
  - **Gupta curve fit** used for **Thermal conductivity** and **Viscosity**

    (previously using kinetic theory)

NASA X-37 with Two-Temperature Model

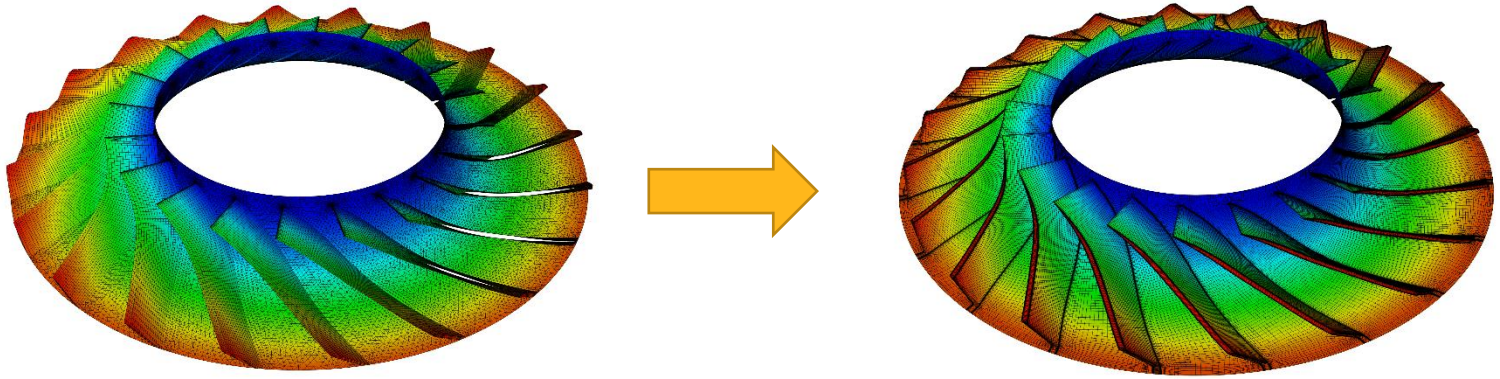
M=19.36
Alt= 65 Km
AoA=40 deg.
Two-Temp
Air-11-park93



Streamlined Aeromechanics Workflow

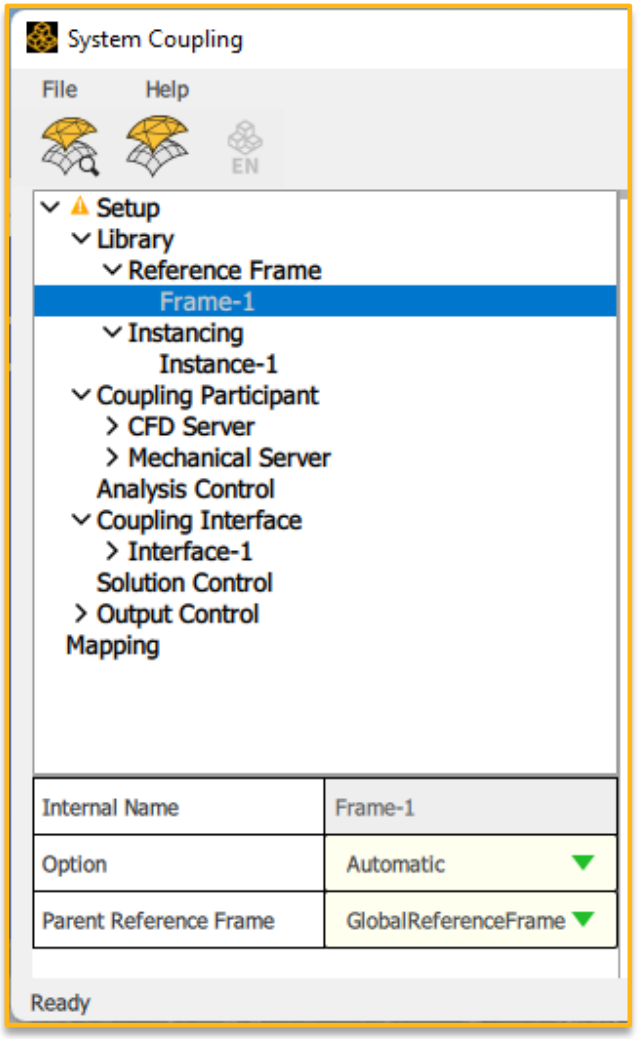
System Coupling provides a streamlined workflow for Aeromechanic analysis

- Easily transfer real and complex mode shapes from ***Mechanical Modal Analysis*** to ***CFD Aerodynamic Damping Analysis***
  - Accurate data mapping, with diagnostics
  - Visualize mapped results in Ansys EnSight
  - New in 23R2: Automatic geometry alignment between

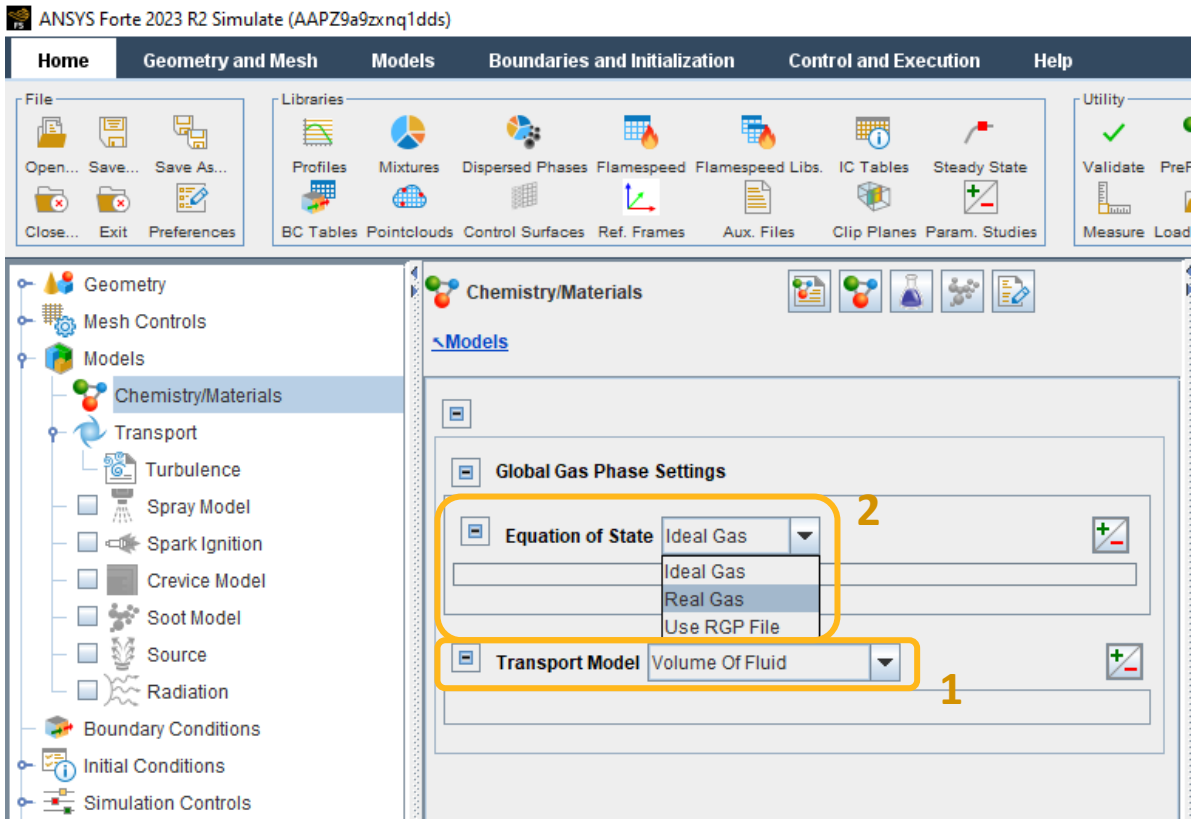


Source: Mechanical Modal Analysis

Target: CFX Analysis



Volume-of-Fluid option as Transport model



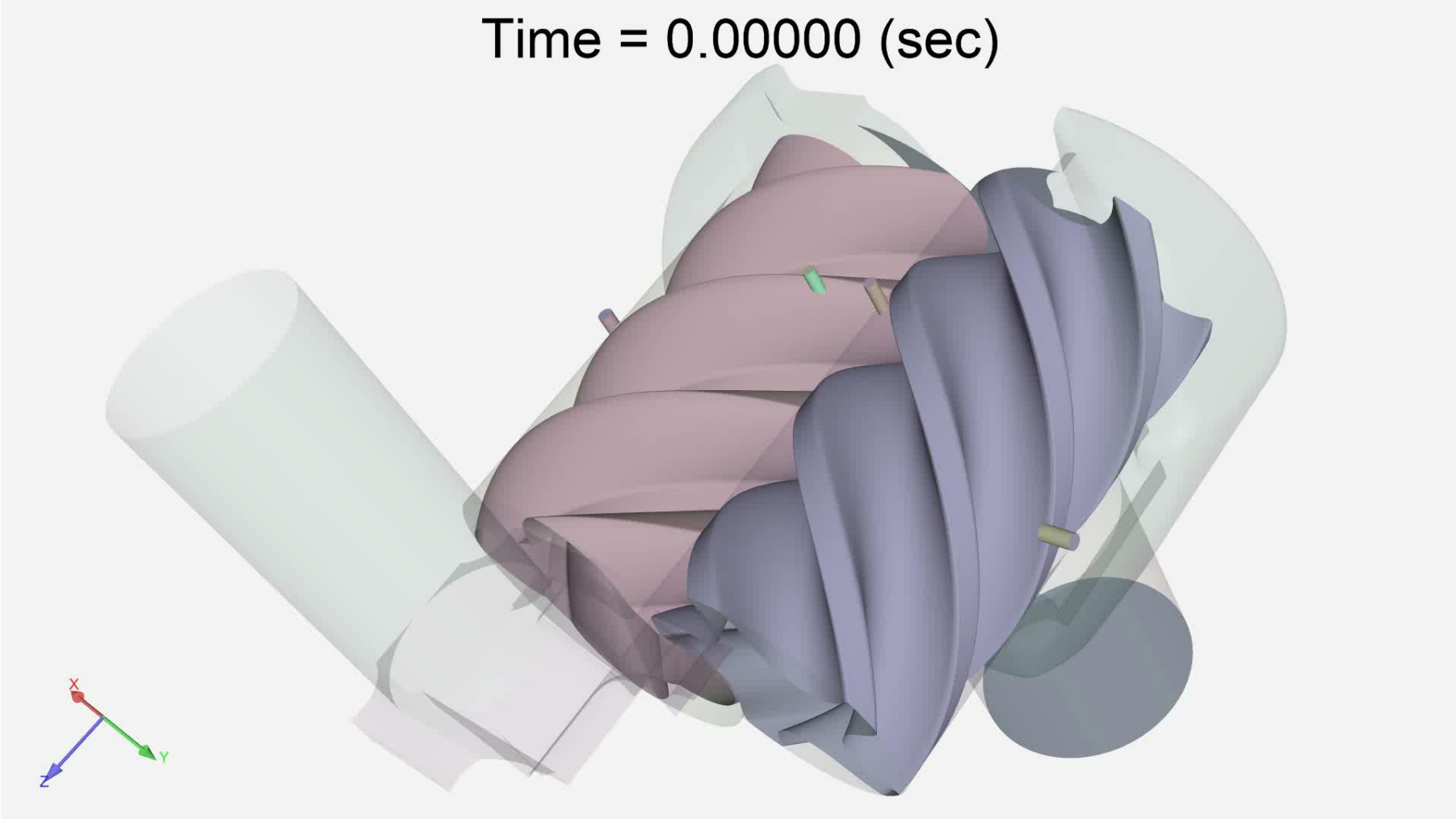
The **Volume-of-Fluid** option can be accessed on the **Chemistry/Materials** node:

1. Options in Transport Model include:
   - Mixture Eulerian 2-Phase
   - **Volume Of Fluid (new in 23R2)**

2. Equation of State options are now effective for the gas phase in Mixture 2-Phase and VOF simulations:
   - Ideal Gas
   - Real Gas
   - Use RGP File (Beta for VOF)

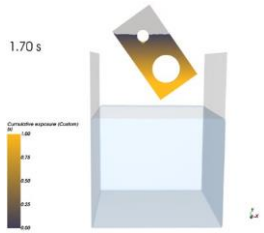
VOF simulation of oil injection in the compressor

- Balance of mass flow rates of air and oil are verified;
- Currently validating the magnitude of mass flow rate and power;
- Planned study: effects of oil cooling of compressed gas and oil sealing of flow leakage.



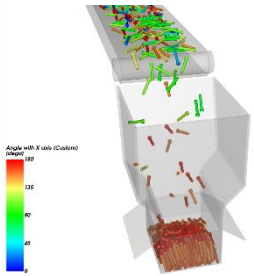
Iso-surface: liquid volume fraction = 10%

Ansys Rocky 2023 R2 Highlights



Smoothed-Particle Hydrodynamics (SPH)

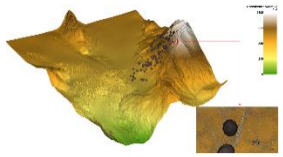
New cutting-edge **post-processing** capabilities, featuring **streamlines**, **flow tracers**, and innovative **boundary interaction statistics**.



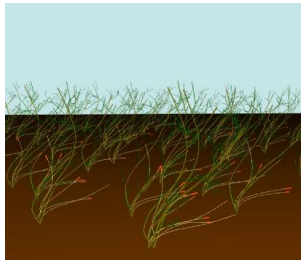
Multiphysics Coupling

The **drag force** induced by a secondary fluid phase on SPH simulations, and reorientation support on the **magnetic force** module.

Performance Enhancements



Advanced Algorithms for Rendering and 3D Manipulation ensuring smooth visualization of **massive STL (geometries)** files.



… And More

Flexible fibers modeling enhancement, plus **User interface and usability** improvements.