Ansys Fluent 2021 R2 Update

21/09/2021

Pedro Afonso

pedro.afonso@ansys.com

Fluids Release Update Series

Ansys Fluent 2021 R2 Update	
User Experience	
Meshing Workflows	Dedicated Webinar
Solver/HPC	
Combustion / Reacting Flows	
Turbomachinery	
Aerospace	Dedicated Webinar
Multiphase	
Heat transfer	
Turbulence	
Batteries and Fuel Cells	

pedro.afonso@ansys.com

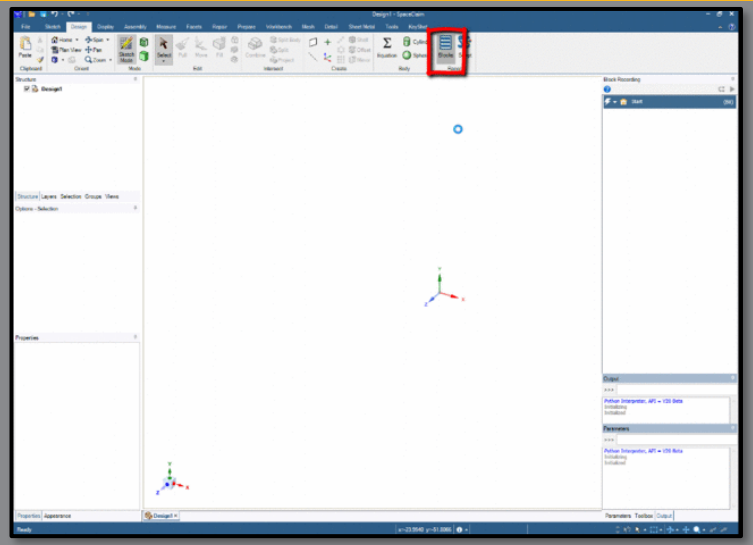
PreProcessing
Geometry

Ansys

Preprocessing: User Experience

NEW

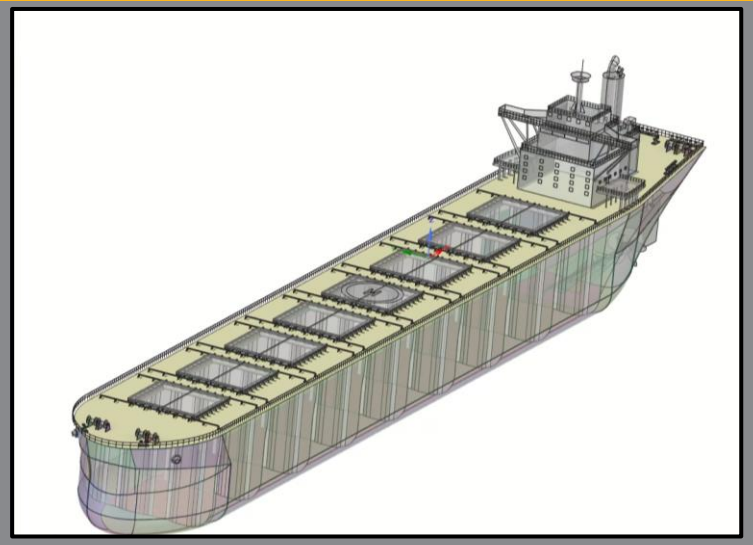
2021 R2 Release Highlights



Geometry Block Recording

Block recording is the baseline for parametric studies

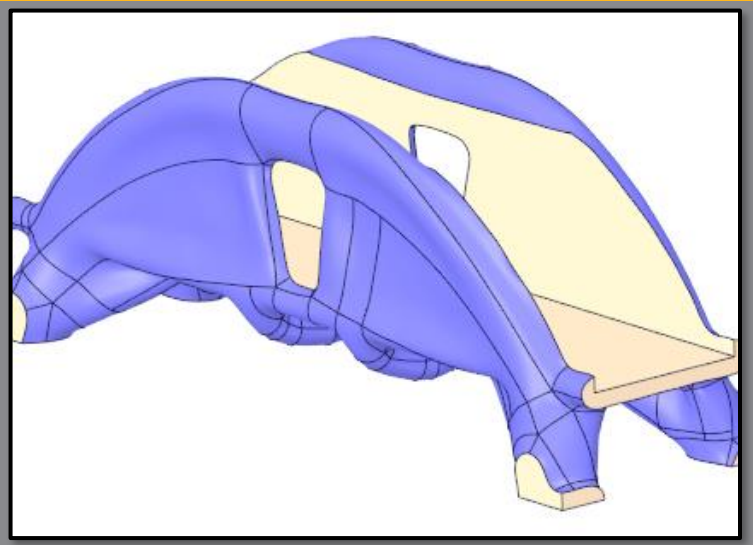
Benefit: Allows changes to be tracked within SpaceClaim while preparing for simulation. Connects bidirectionally from CAD <-SpaceClaim->Workbench. Supports parametric and unplanned changes from CAD



Geometry Scritping (Python)

Faster Geometry Cleaning: 2X to 10X faster geometry preparation

Benefit: Mostly on semi or full automated mesh generation. Customized body, surface or feature along with full control on geometry operations



BACK TO CAD

- STL Preprocessing & Deviation Tool: Tend to focus on Cleanup/Geometry Repairing.
- STL Preprocessing & Back to CAD: Essential to Non-Parametric Optimizations such as Adjoint (Shape Optimization) and/or Topology Optimization.

Benefit: Directly interact work with STL geometries end-to-end.

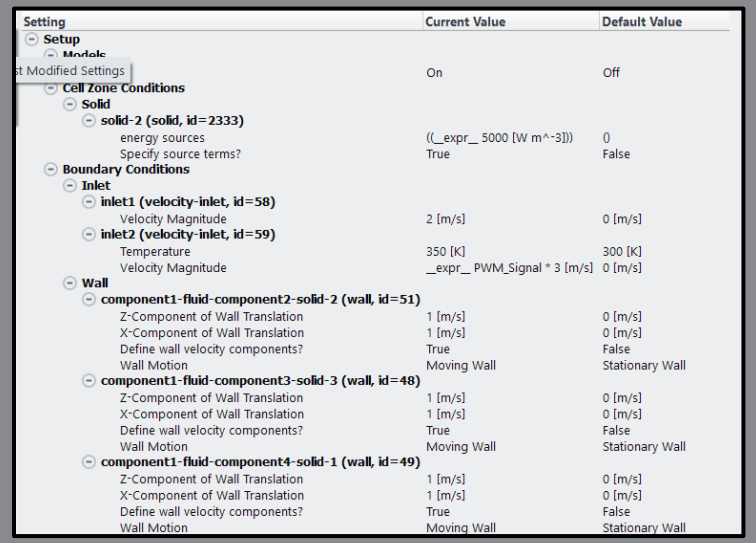
User Experience

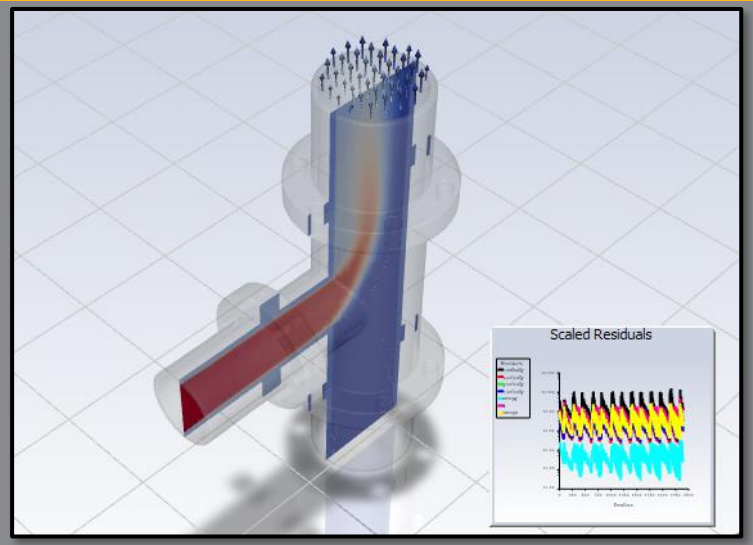
Ansys

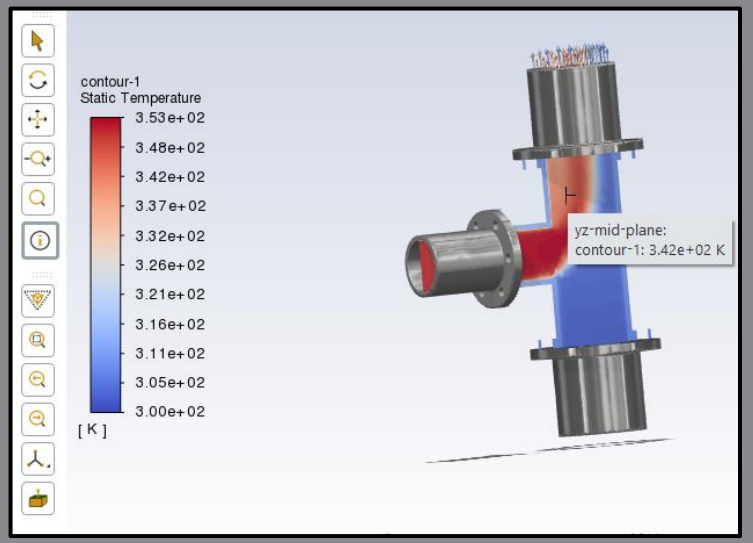
Ansys Fluent : User Experience

NEW

2021 R2 Release Highlights







View Modified Case Settings

Summarize case differences from default settings

Benefit: Compare modified settings to default setting to ensure no values were missed

Embedded Windows

Layouts of embedded windows will now be directly saved in case and data files

Benefit: Post process more efficiently by having monitors or residuals right next to surfaces, graphics and animations

Improved Post Processing

- Mouse probe value on post processing objects such as contour, vectors, path lines
- New colors and realistic rendering capabilities
- Additional color maps with improved lighting are now available

Benefit: Directly interact with your model while post processing and compare

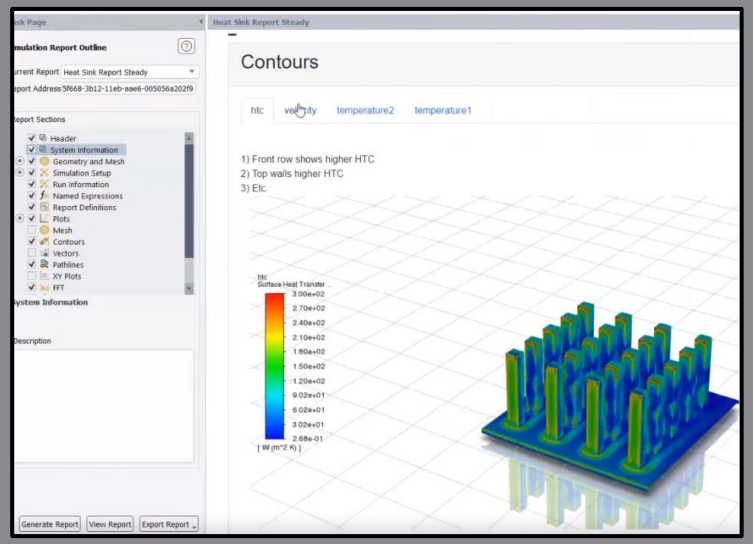
Ansys Fluent : User Experience

NEW

2021 R2 Release Highlights

Display Improvement: 25M cells, 1400+ zones

Display	2021 R1	2021 R2	Improvement
Mesh	145.548s	50.217s	65.5%
Vector	156.400s	63.580s	59.3%
Contour	183.148s	59.377s	67.5%



Performance Improvements for Large Cases

Performance improvement for case reading and visualization using Fast Interactive Display option, a new logic to adaptively reduce model detail if necessary to preserve interactive manipulation performance

Benefit: Case read improved up to 5x-10x for cases with 10k+ zones

Simulation Reports

Create a report of your simulation data and results using Fluent's simulation reports feature that can be viewed in Fluent or as a pdf

Benefit: Improved report generation speed in R2

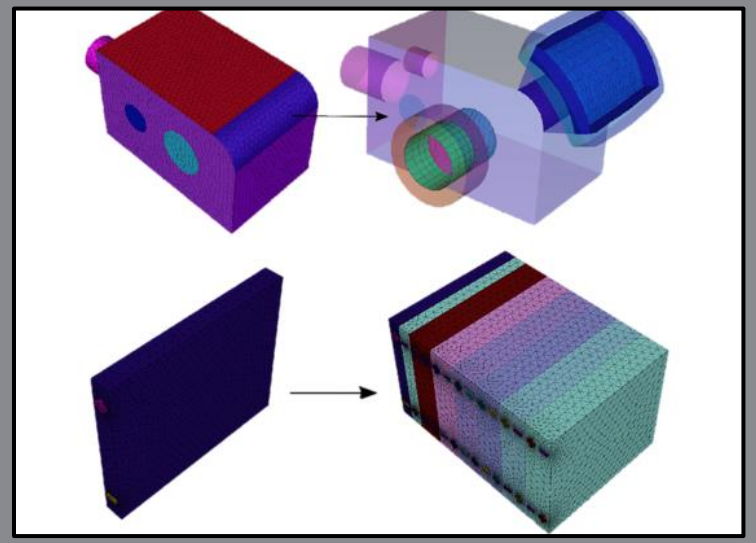
Meshing Workflows

Ansys

Ansys Fluent : Meshing Workflows

NEW

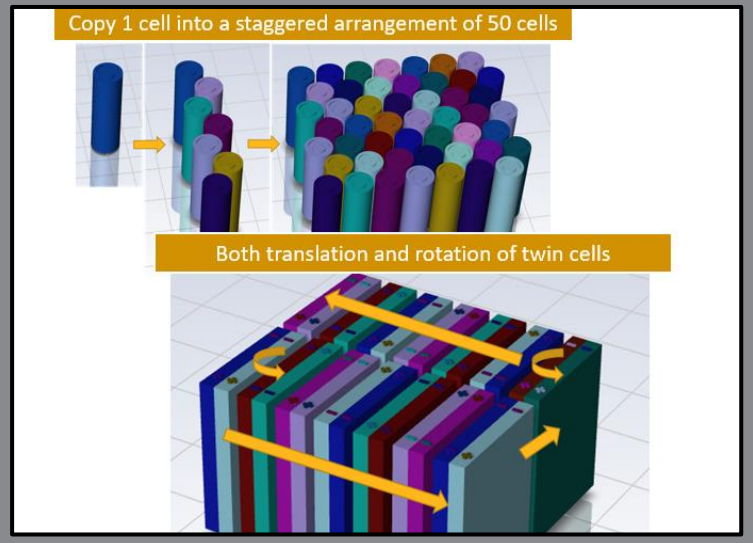
2021 R2 Release Highlights



Watertight Meshing : New Tasks

- Extrude Volume Mesh: Mesh extrusion from planar and non-planar surfaces
- Import Body of Influence Geometry: Use imported CAD or mesh files to define bodies of influence
- Set Up Periodic Boundaries task can now be inserted before the Generate Surface Mesh task

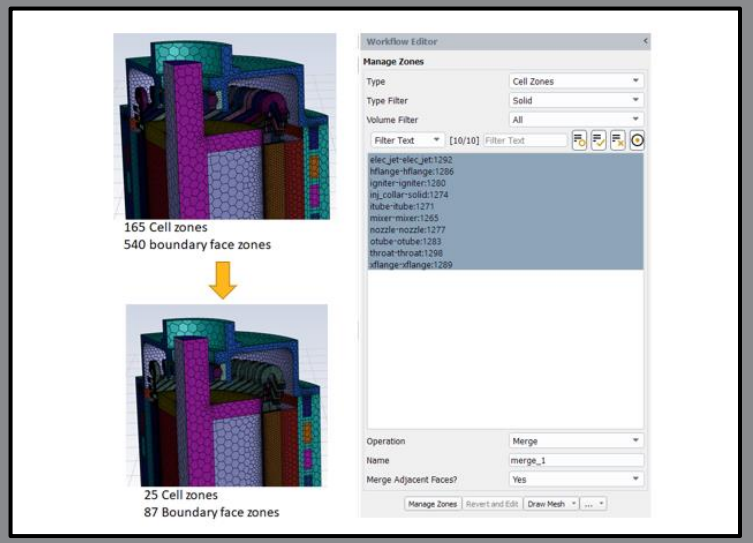
Benefit: Enable new meshing workflows



Watertight Meshing : Linear Mesh Patterns

Add Linear Mesh Pattern task now allows custom patterning (including re-orientation) / naming conventions

Benefit: Simplifies mesh generation for battery simulations where simple linear arrays of cells are insufficient



Watertight Meshing : Zone Merging

By default, the following merges are done automatically (additional controls possible with new Manage Zone task) :

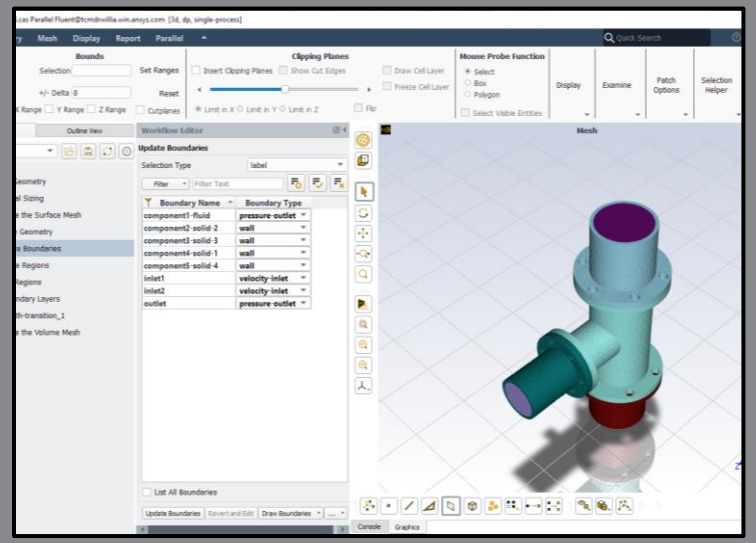
- Cell zones within body Named Selection
- Merging of adjacent face zones
- Re-naming of internal zones

Benefit: Automatically reduces mesh complexity

Ansys Fluent : Meshing Workflows

NEW

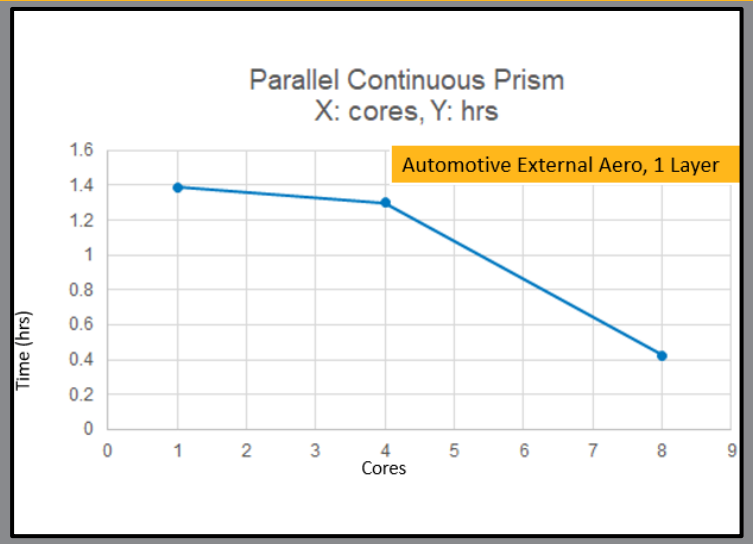
2021 R2 Release Highlights



Dockable Workflow Editor

The dockable workflow editor enables to optionally separate workflow task editor from task list

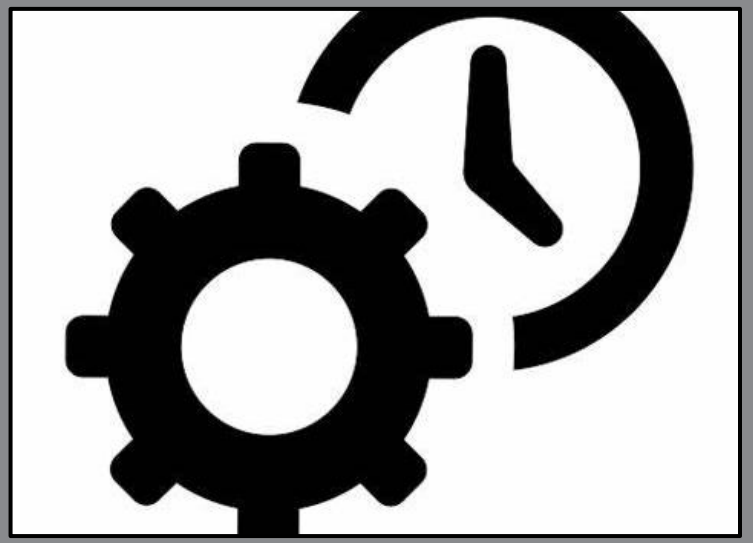
Benefit: More space to work with in task editing for cases with many regions, etc.



Fault Tolerant Meshing : Speed/Robustness

- Parallel continuous prism generation
- Compute one size field for wrap or target and use wrap/target size ratio for the other
- Import wrap/target size fields rather than computing
- Improved inner wrap robustness

Benefit: Improved mesh generation throughput (i.e., 25.1 hrs --> 13.2 hrs for reference automotive case)



Fault Tolerant Meshing : Improvements

- Parallel polyhedral volume mesh support
- Usability enhancements in Import CAD/Part Management task
- Transformation ops can be applied prior mesh objects creation
- Porous region creation through text file import
- Auto Assign Zone Types? In Generate Surface Mesh allows you to automatically assign zone types based on names.

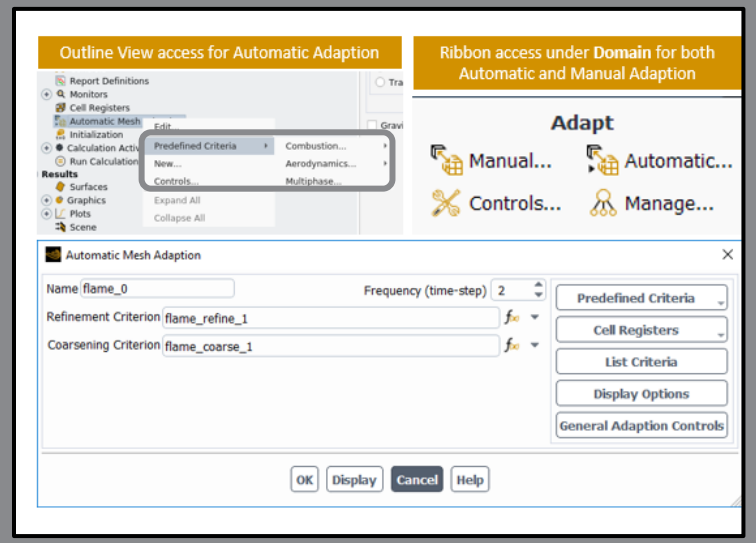
Benefit: Improved productivity when using Fault Tolerant Meshing

Ansys Fluent : Meshing Workflows

Join our experts in the **Mesh Adaption Update** on September 16 | 10 AM EDT

NEW

2021 R2 Release Highlights

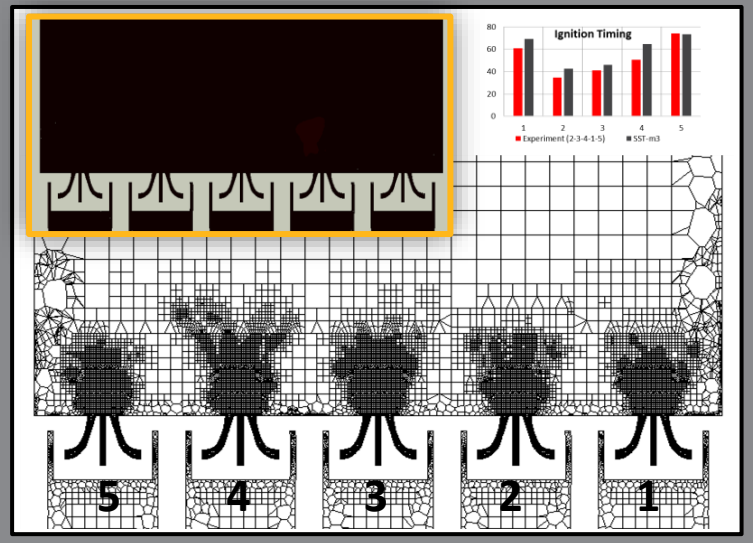


Mesh Adaption : Best Practices

Powerful automatic mesh adaption for all cell types using named object architecture with support for new adaption criteria for combustion and high-speed aerodynamics.

Benefit: Significantly reduce simulation time by using highly refined mesh only where needed

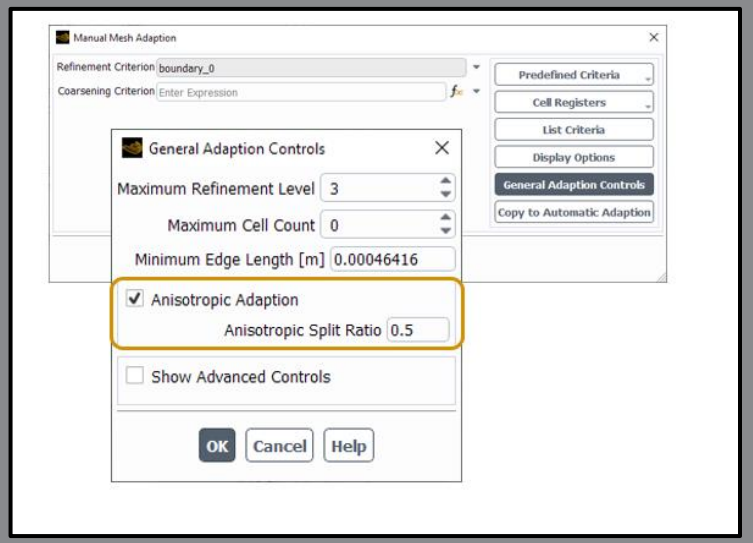
.



Mesh Adaption : Combustion Criteria

Refine the mesh based on combustion criteria for Finite-Rate and Flamelet Generated Manifold (FGM) combustion.

Benefit: Simulation reduced from 3 weeks (12M cells) to 3 days (5M cells) for ignition sequence simulation



Mesh Adaption : Anisotropic Boundary

PUMA-based anisotropic adaption for prismatic boundary layers now available in GUI. Once enabled, prismatic boundary cells matching any defined adaption criteria will be anisotropically adapted/coarsened.

Benefit: Improved legacy anisotropic adaption

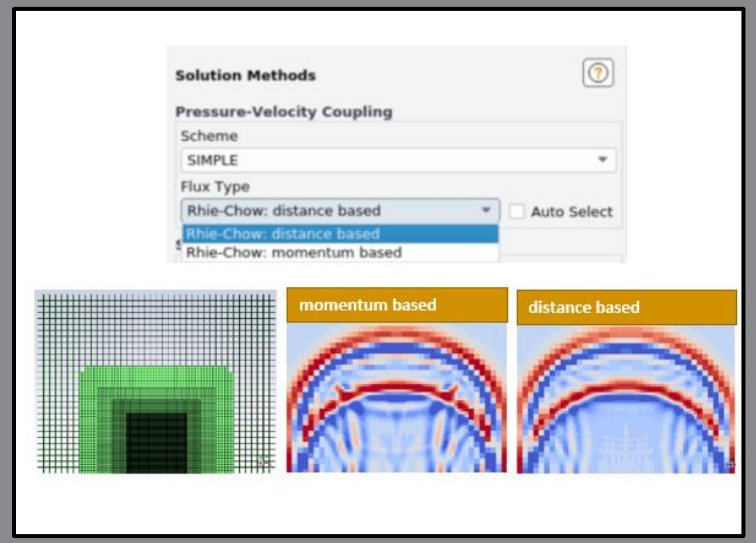
Solver/HPC

Ansys

Ansys Fluent : Solver/HPC

NEW

2021 R2 Release Highlights

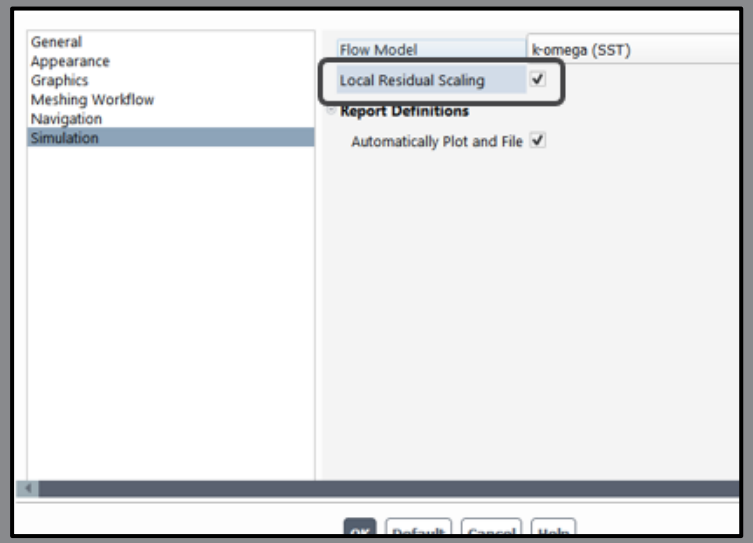


Pressure Based Solver : Rhie-Chow Flux

Enables manual (or automatic) selection of the optimal flux formulation for different applications :

- Rhie-Chow distance based : recommended for compressible flows/acoustics; tends to avoid spurious reflections at cell-size jumps
- Rhie-Chow momentum based : more robust for incompressible flows and combustion

Benefit: Improved accuracy based on application (i.e. no spurious reflections with distance based)



Pressure Based Solver : Enhancements

Enhanced adaptive time stepping with adaptive meshing and local residual scaling.

Benefit: Improved performance and accuracy with mesh adaption and residuals less mesh dependent with local residual scaling

Features	Contact Detection	Gap Model
Blocking Flow	✓	✓
Multiple gap Type		✓
Multiple gap definition		✓
Multilevel Solution Stabilization		✓
Consistent Interpolation in gap regions		✓
Search based Marking	✓	✓

Gap Model : Flow blockage in tiny gaps

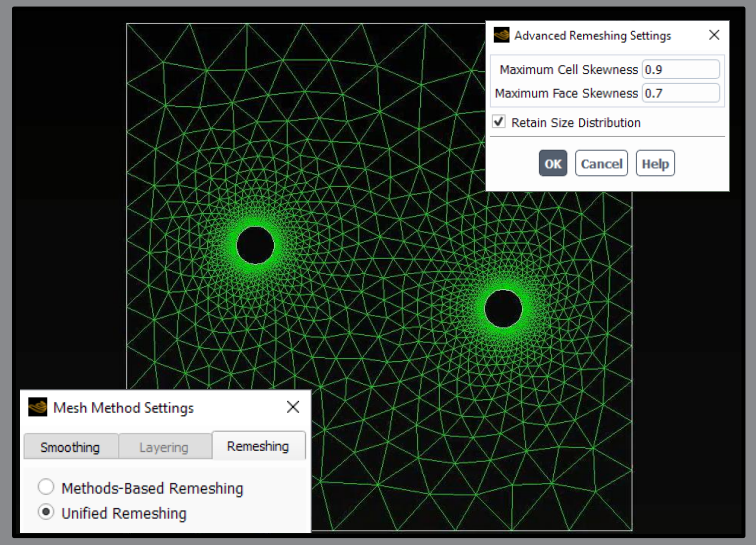
Enables to simulate the flow blockage in tiny gaps with all moving mesh simulation techniques.

Benefit: Improved accuracy with additional advantages compared to dynamic mesh contact detection

Ansys Fluent : Solver/HPC

NEW

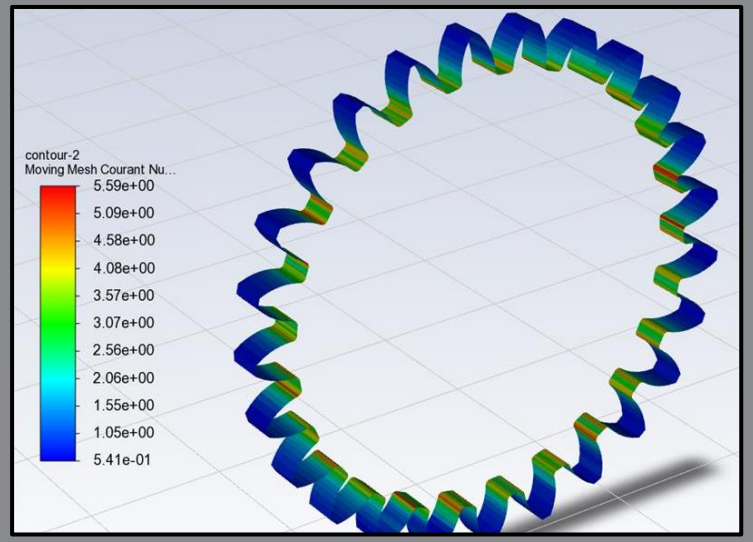
2021 R2 Release Highlights



Remeshing: Usability and Mesh Size Control

Unified remeshing with simple checkbox to enable it sufficient for most cases. Retain Size Distribution approach is used as default, so the initial mesh is used to control mesh size during remeshing.

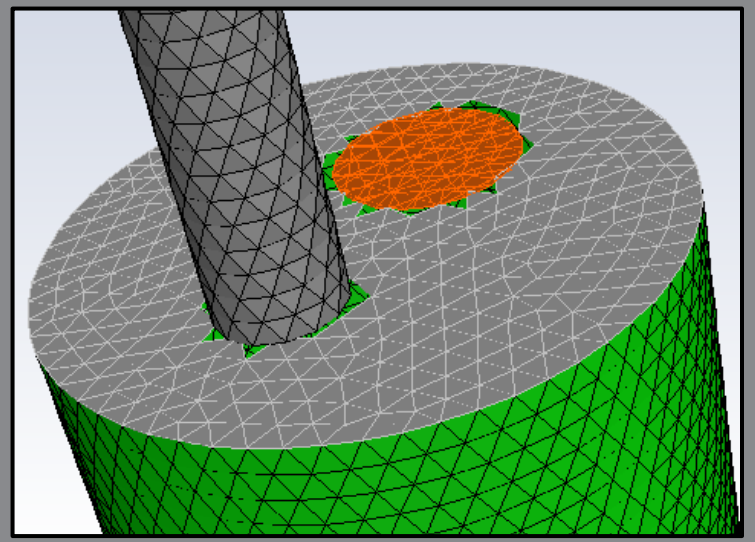
Benefit: Further simplified dynamic mesh setup



Moving Mesh Courant Number

Moving Mesh Courant Number field variable extended to all single-phase and multiphase simulations (not only VOF)

Benefit: Moving Mesh Courant Number field variable helps to assess appropriate timestep size for Sliding Mesh and MDM calculations.



Mesh Interfaces: Usability

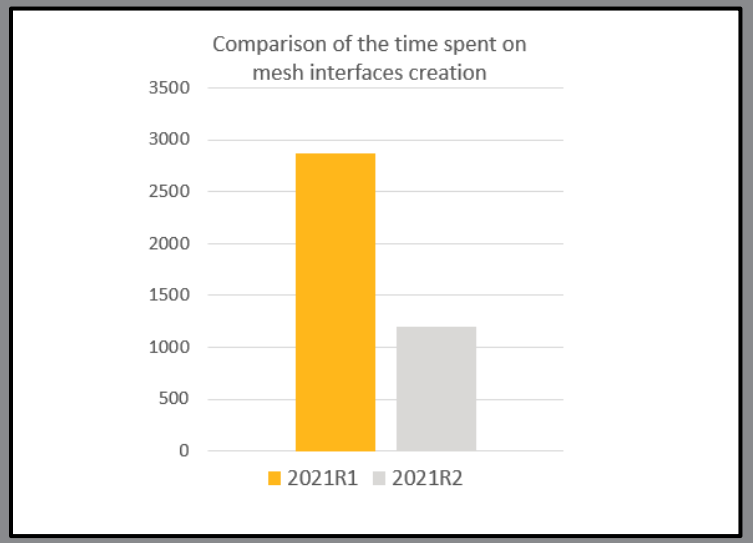
Visualization of non-overlapping zones (in addition to overlapping zones) for easy assessment of intersection quality

Benefit: Easier assessment of intersection quality

Ansys Fluent : Solver/HPC

NEW

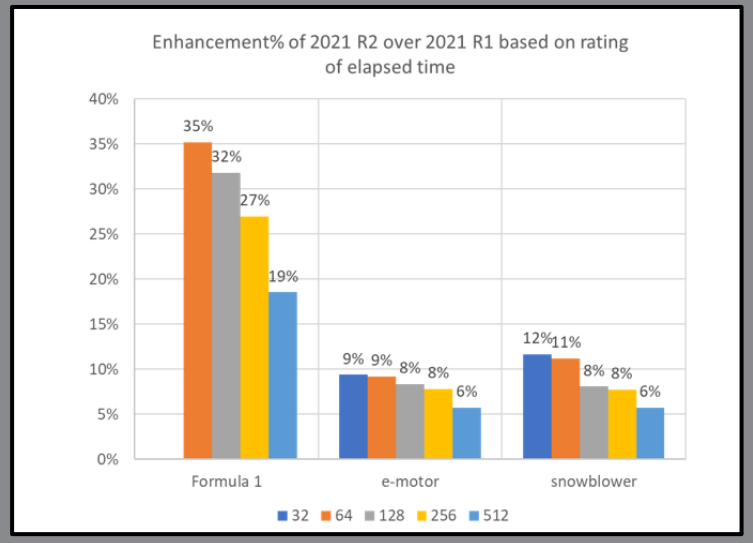
2021 R2 Release Highlights



Mesh Interfaces: Performance/Robustness

Automatic creation time for problems with many interfaces significantly reduced. Improved gradient method at fluid-fluid interfaces that provides more robustness and potentially accuracy for poorly matching interface

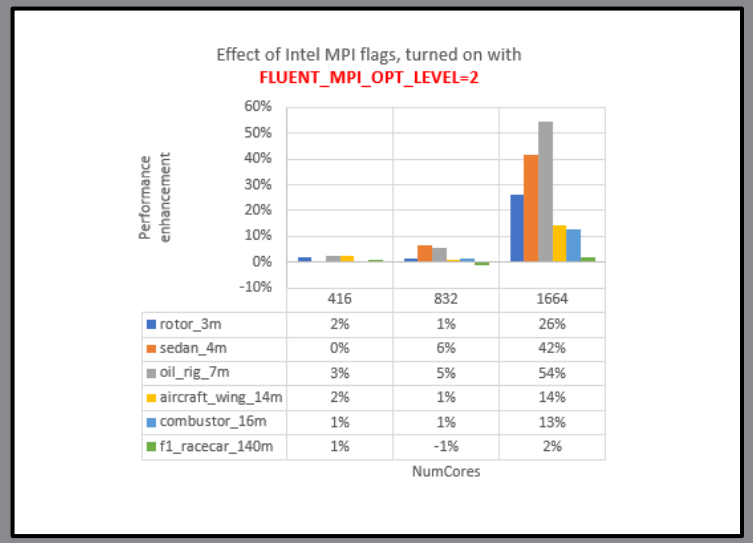
Benefit: More than 2x speed up on mesh interface creation



HPC: Sliding Mesh Performance

Sliding mesh parallel performance enhancement and enhancement in sliding mesh cases at each time step after mesh slide.

Benefit: Up to 35% performance speed up for sliding mesh



	416	832	1664
rotor_3m	2%	1%	26%
sedan_4m	0%	6%	42%
oil_rig_7m	3%	5%	54%
aircraft_wing_14m	2%	1%	14%
combustor_16m	1%	1%	13%
f1_racecar_140m	1%	-1%	2%

HPC: Job Scheduler / MPI Support

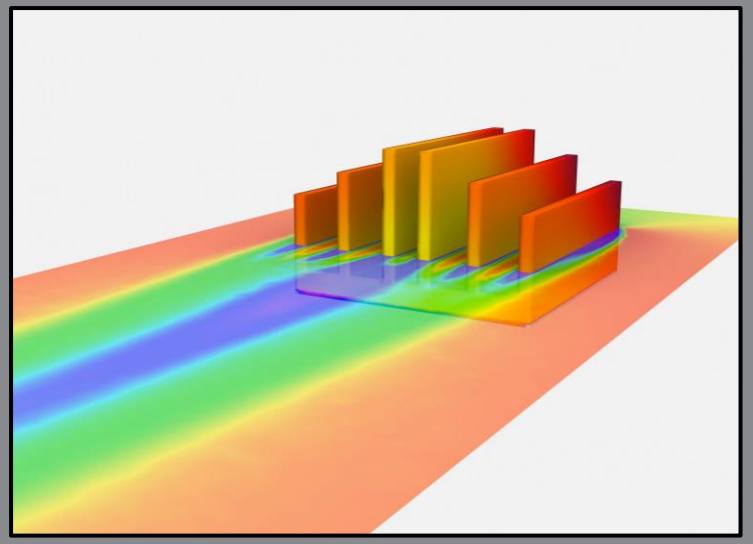
Added SLURM support and a flag (FLUENT_MPI_OPT_LEVEL 2) for parallel scalability with Intel MPI 2019 U8

Benefit: Up to 54% performance enhancements with Intel MPI flag turned on

Ansys Fluent : Solver/HPC

NEW

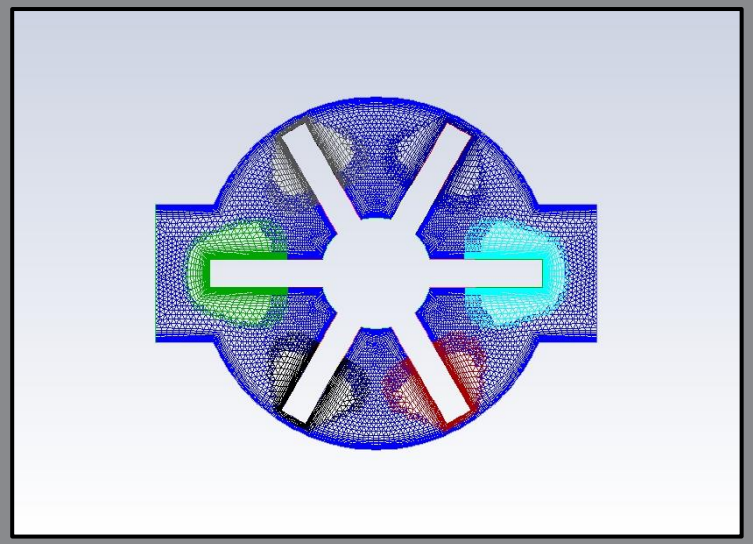
2021 R2 Release Highlights



Adjoint Speed and Memory Improvements

- Decouple the shape sensitivity calculation from the AMG allocation, which reduces the memory cost of postprocessing and design tool calculation considerably.
- Support partial coupling adjoint solver: the adjoint continuity and momentum equations are solved in the coupled manner, while other equations are solved in a segregated manner

Benefit: per iteration speed up to 2x faster and memory reduction up to 30%



Overset Mesh Enhancements

Faster Solutions with NITA compatibility

Benefit: Significant performance gains possible compared to Coupled (8x) and SIMPLE solvers (3x)

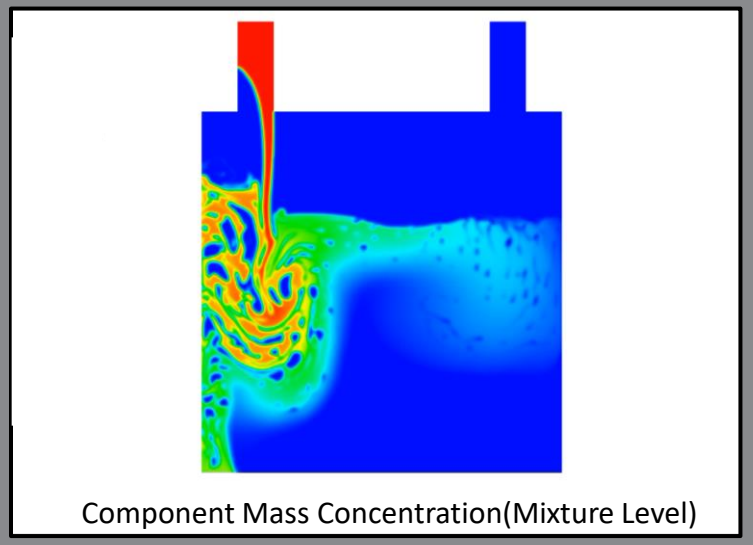
Combustion/Reacting Flows

Ansys

Ansys Fluent : Combustion/Reacting Flows

NEW

2021 R2 Release Highlights

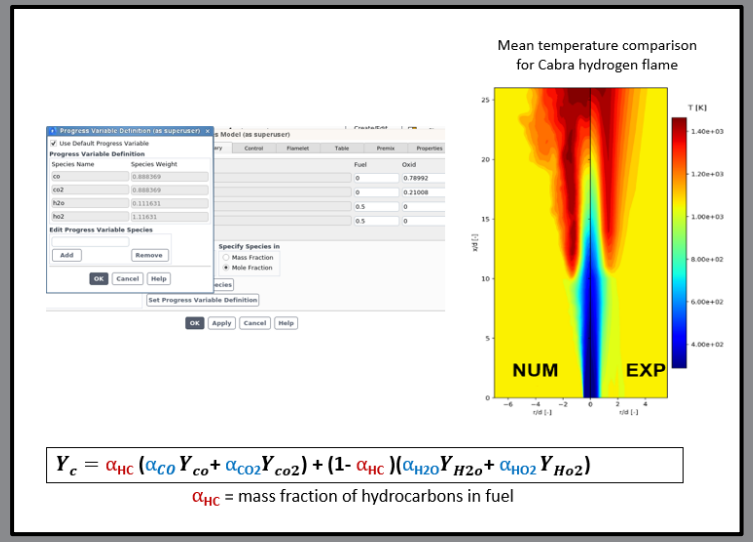


Component Mass Concentration(Mixture Level)

Species Post-Processing Improvements

Addition of Mixture Level Mass Concentration. Historially, Fluent has not had any native post-processing variable to calculate mass of a component.

Benefit: Better visualization of a component physics. Facilitates mass of a component by taking a volume integral



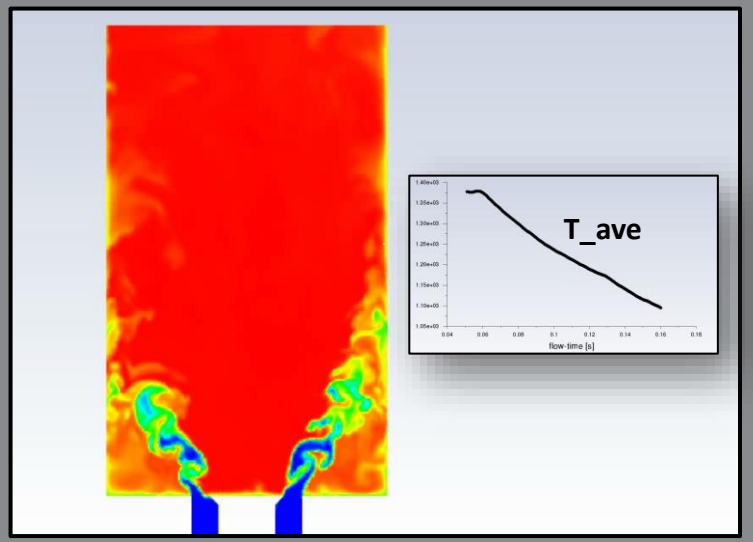
$$Y_c = \alpha_{HC}(\alpha_{CO}Y_{co} + \alpha_{CO2}Y_{co2}) + (1 - \alpha_{HC})(\alpha_{H2O}Y_{H2o} + \alpha_{HO2}Y_{Ho2})$$

α_{HC} = mass fraction of hydrocarbons in fuel

Hydrogen Combustion

Progress variable as weighted combination of hydrogen and hydrocarbon flames and FGM and SBES proven for H_2 and H_2-CH_4 blends

Benefit: Accurate prediction of Hydrogen concentration and blends



Strained FGM for Lean Blow-out modeling

Addition of a strained FMG model using CKCFD APIs :

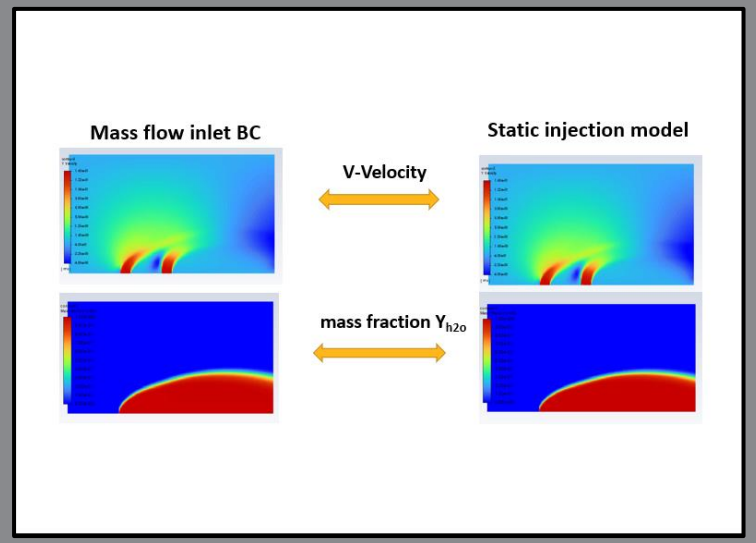
- Compute laminar flame speed table using Oppdif
- Parallel manifold generation (minutes *vs* days using Cantera)
- Use the table for turbulent flame speed in FGM-TFS model

Benefit: Better flame stability particularly at lean conditions like lean blow out

Ansys Fluent : Combustion/Reacting Flows

NEW

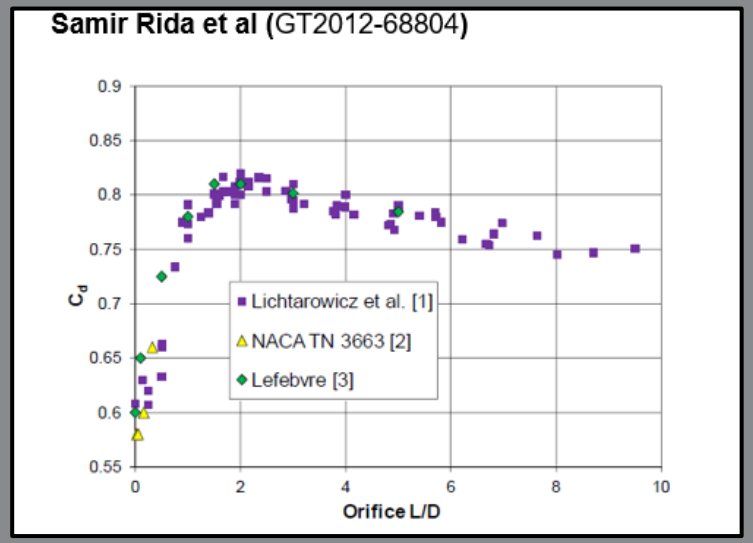
2021 R2 Release Highlights



Perforated Wall

User-specified injection conditions for mass, temperature, velocity, species/combustion scalars

Benefit: Accurate static Injection model for perforated wall and other applications (e.g., fluidized beds with DDPM)



Perforated Wall: User Defined "Dynamic" Cd

Discharge coefficient (Cd) is often a complex function of hole geometry. Using Dynamic Cd? option, you can use a UDF instead of a specified value for Cd.

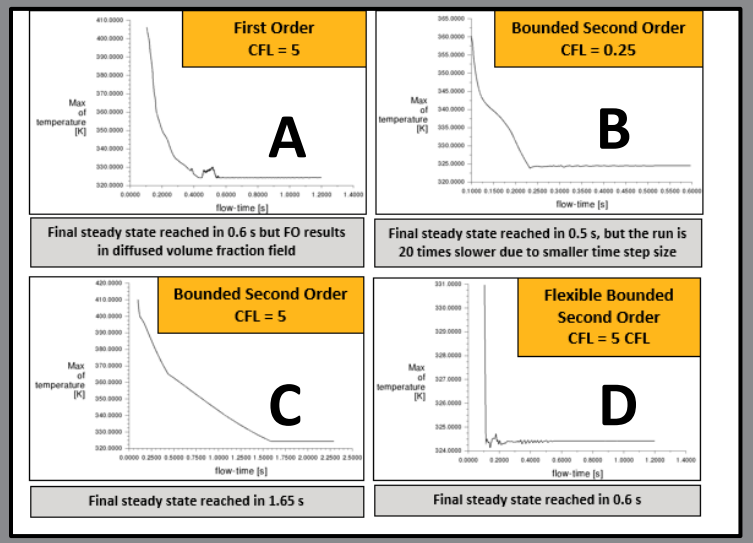
Benefit: Enables custom function for Discharge Coefficient

Multiphase

Ansys Fluent : Eulerian Multiphase

NEW

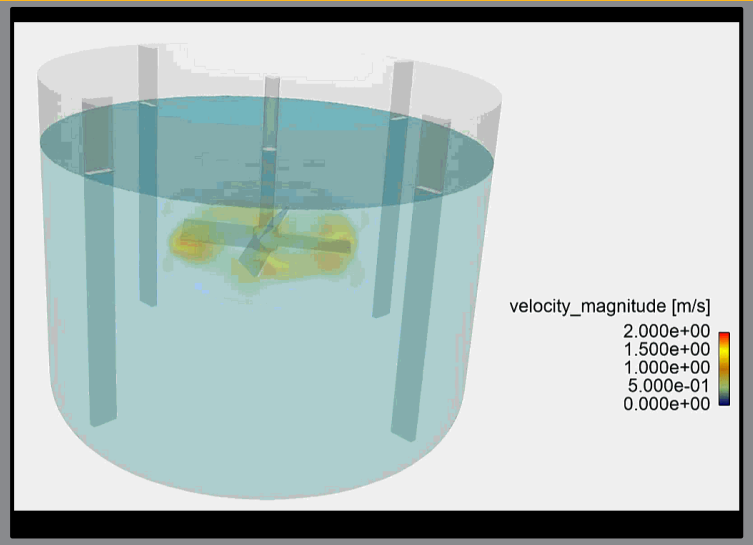
2021 R2 Release Highlights



Flexible Bounded Second Order

Uses BSO for volume fraction, flow, turbulence equation for a sharper interface resolution. Uses 1st order time for other equations like species, temperature, population balance etc. to ensure local boundedness.

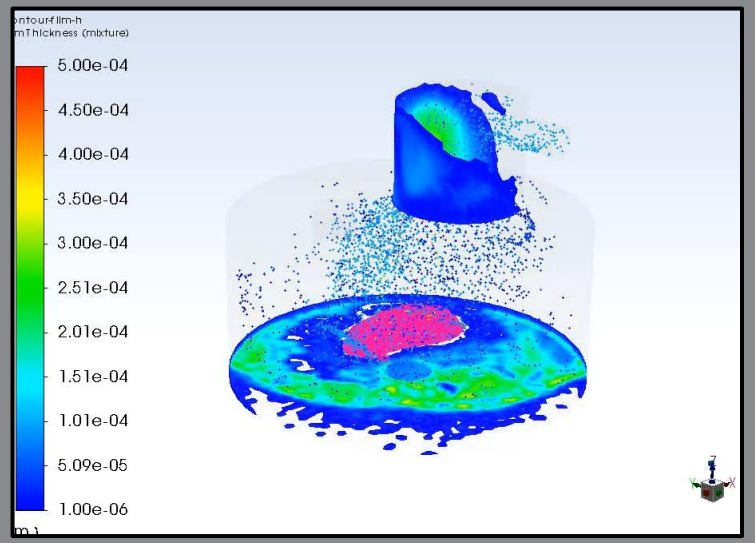
Benefit: Improves solution speed and robustness. Improves transient evolution of such cases. No negative impact on solution accuracy.



Instability Detector Improvements

New CFL type based on interfacial cells was introduced to synchronize Instability Detector with Global Courant Number. Old default of CFL cut-off for instability detector was too conservative and adversely affected the solution speed.

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



Adaptive Time Stepping for Model Transition

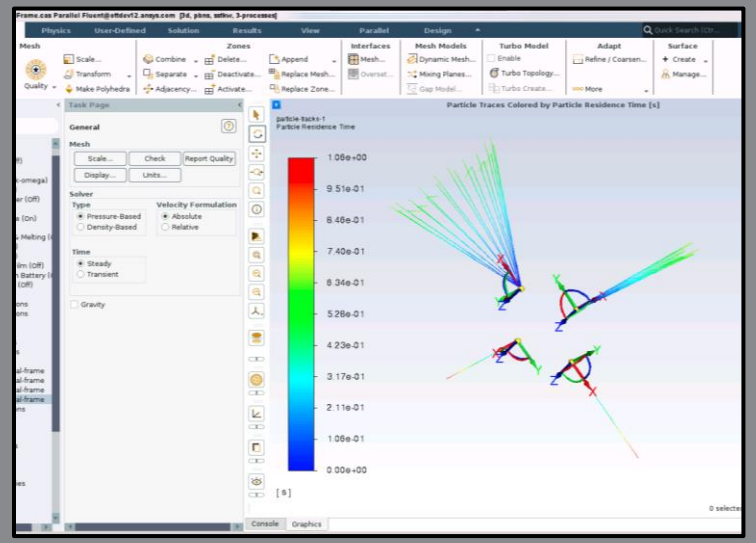
New Adaptive Time Stepping considers that VOF is formed due to transition → Time step size gets updated from all cells until the interface is detected. Old adaptive time stepping treatment picked the time-step size only from interfacial cells (not present at beginning of simulation).

Benefit: Better selection of the time step during model transition

Ansys Fluent : Discrete Phase

NEW

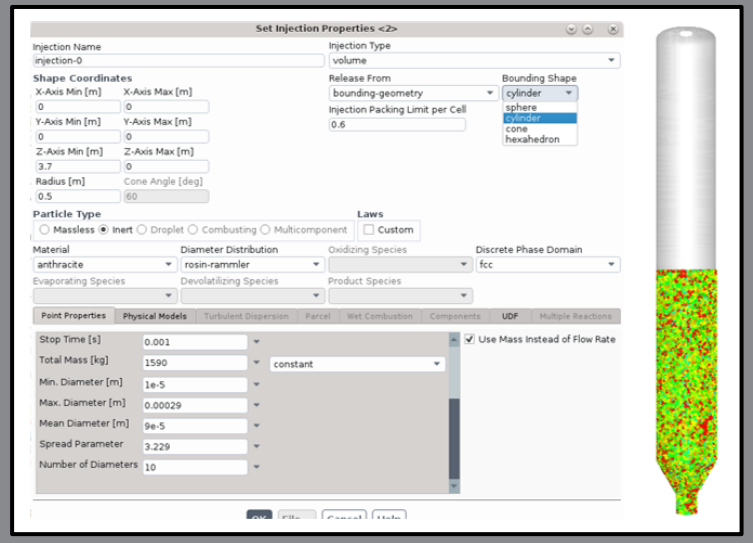
2021 R2 Release Highlights



Injection : Local Reference Frames

Support of local coordinate systems for most injection types. File injections can be applied to different reference frames without the need to manually transform the injection data

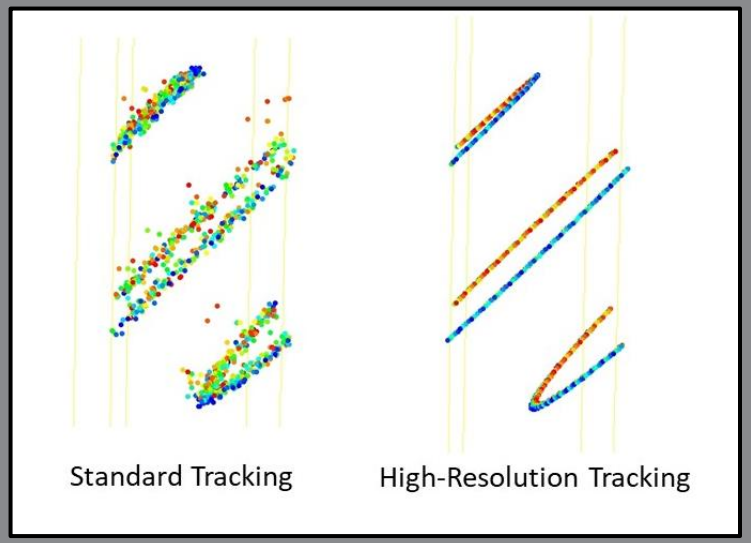
Benefit: Simplifies setup of multi-hole injectors



Injection : Random Surface Injection

Fully supported (not still beta) option for surface injections

Benefit: Enable particle injections from volumetric regions defined by cell zones and bounding geometries (sphere, cylinder, cone, hexahedron)



High Resolution Particle Tracking

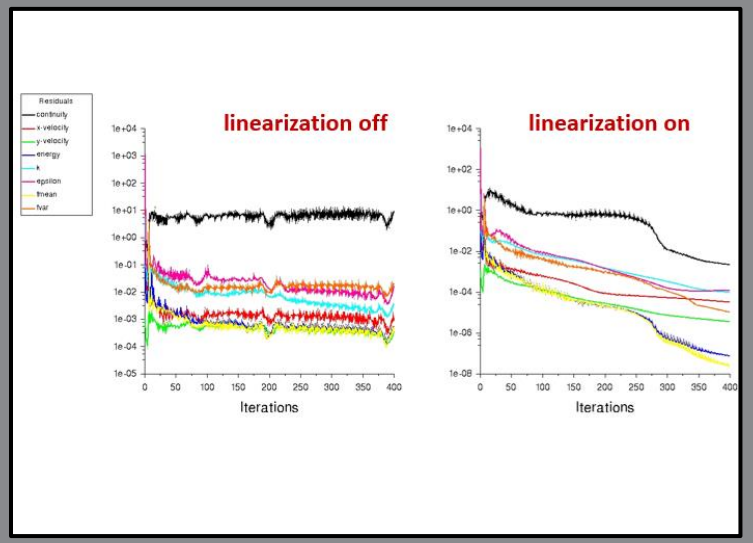
Improved accuracy and robustness of particle tracking, including compatibility with overset mesh.

Benefit: More accurate particle tracking

Ansys Fluent : Discrete Phase

NEW

2021 R2 Release Highlights

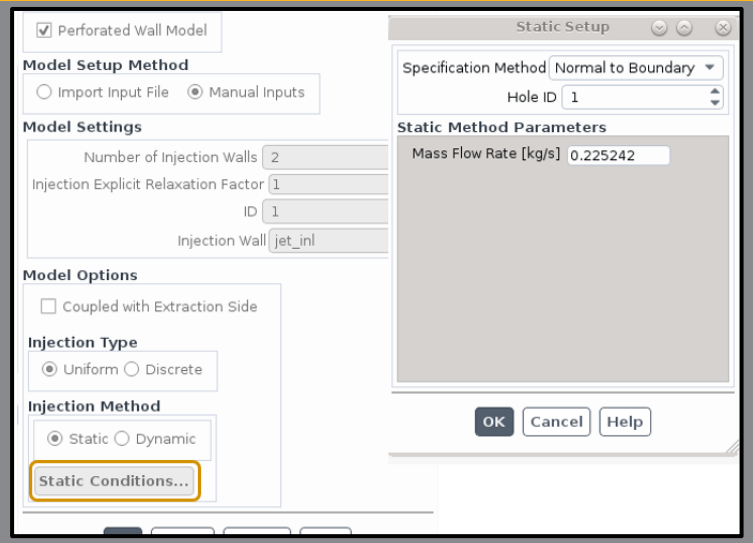


Linearization of DPM Mixture Fraction Source

Linearization is available for both primary and secondary mixture fractions, as well as the inert species.

Can be enabled via the TUI:
/define/models/dpm/interaction/linearized-dpm-mixture-fraction-source-terms? Yes/No

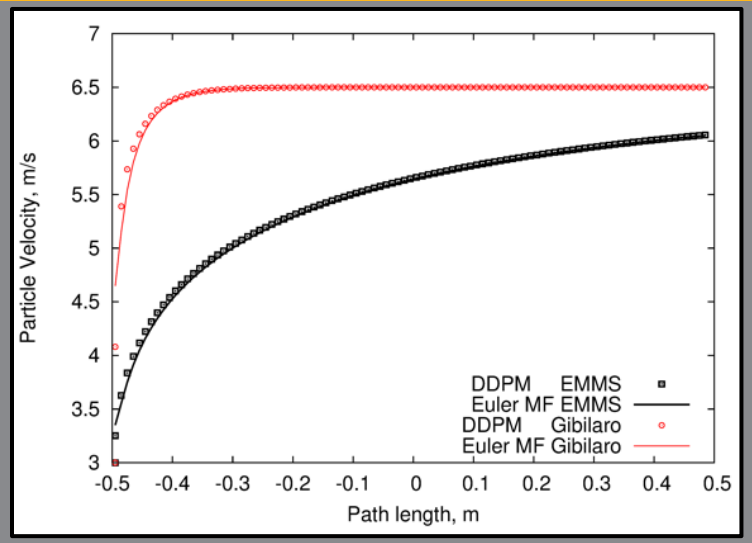
Benefit: improve stability and convergence for combustion simulations with liquid fuel sprays using the non/partially premixed combustion models



Perforated Wall Injection

Perforated wall boundary condition can now be used for multiphase flows to avoid meshing of tiny inlets

Benefit: Avoids small cells for tiny inlets which requires small DDPM parcels leading to impractically large mesh and parcel count



Advanced Drag Laws for Granular Materials

EMMS, Filtered, Gibilaro and Huilin-Gidaspow drag laws

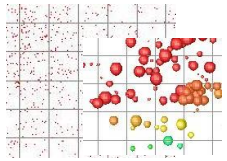
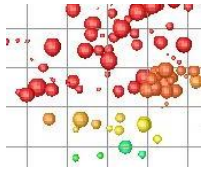
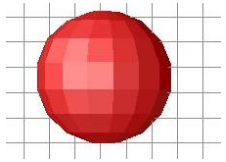
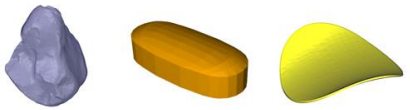
Benefit: Increased accuracy for granular materials

Ansys CFD Coupling (Fluent)

- Right solution for the right application

LAGRANGIAN MULTIPHASE MODELS

Particles treated as discrete

DPM & DDPM	DEM	MPM	ROCKY
Dilute to dense systems	Sub-grid Soft-sphere model	Super-grid Hard-sphere model	ROCKY
			

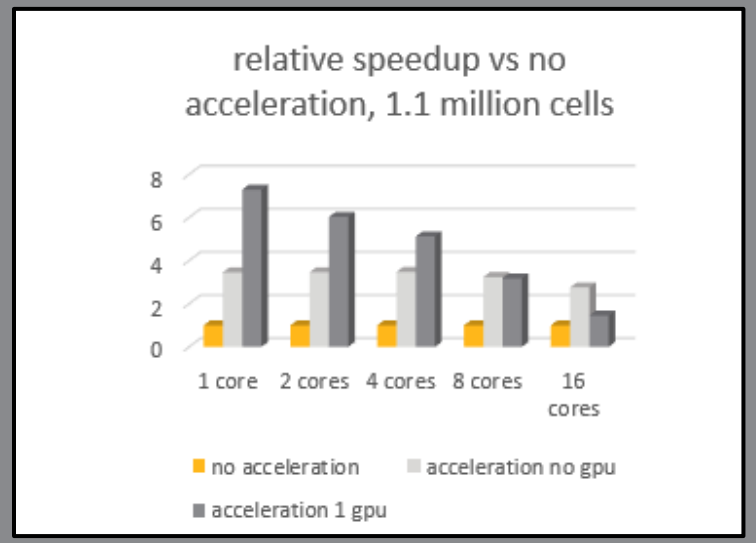
Heat Transfer / Turbulence

Ansys

Ansys Fluent : Heat Transfer

NEW

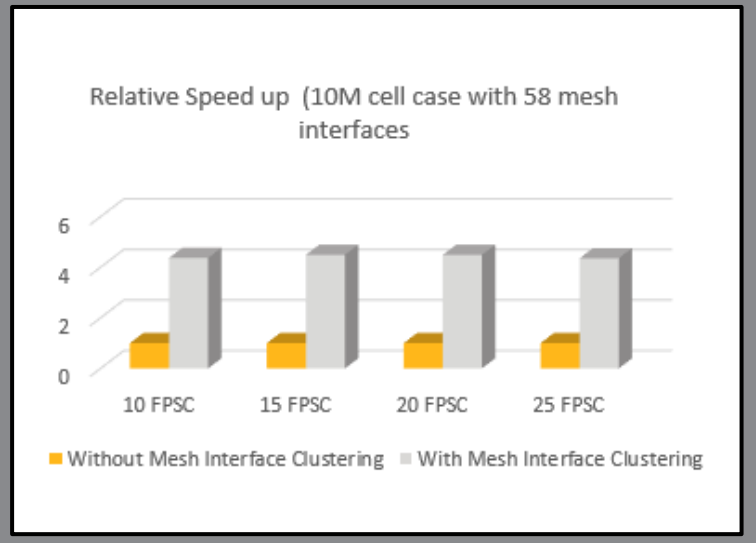
2021 R2 Release Highlights



DO acceleration – Periodic BC support

Support of periodic BC with DO acceleration (with and without GPU acceleration)

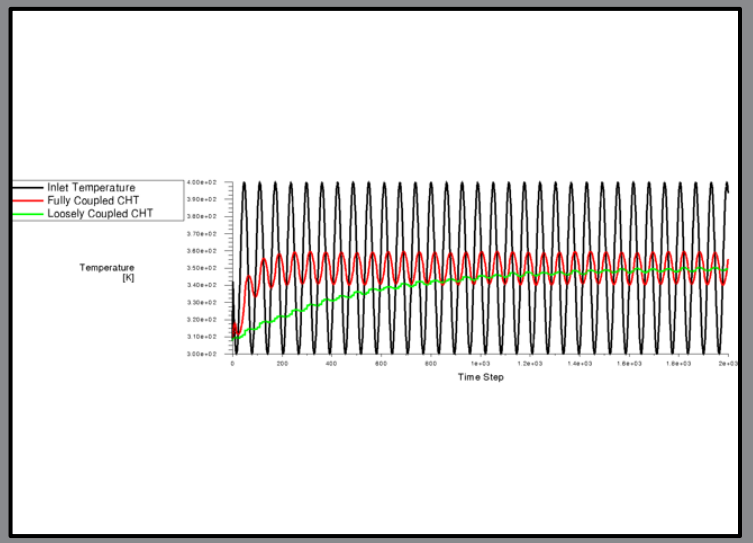
Benefit: Up to 35-40% time reduction for industrial case



Mesh Interface clustering

Mesh Interface clustering for optimal VF calculations time

Benefit: Up to 3x performance speedup



Loosely-Implicit Coupled CHT

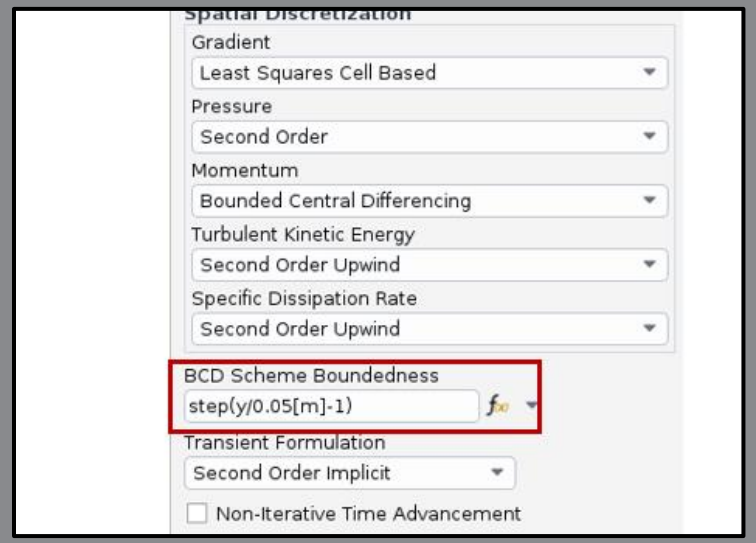
Fluid is solved explicitly with external thermal boundary conditions for most of the simulation time. Fluid is solved implicitly only when coupled with recovered solids in one linear system of equations (as in a single Ansys Fluent session with fully coupled CHT)

Benefit: 2X speedup in energy equation computation and 25% overall speedup in performance

Ansys Fluent : Turbulence

NEW

2021 R2 Release Highlights

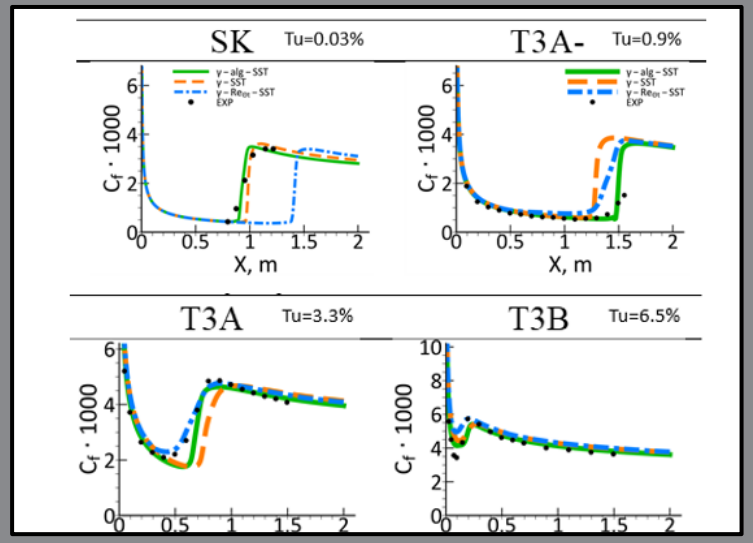


Tunable Bounded Central Differencing (BCD)

Unified implementation of tunable & standard BCD with a customizable boundedness (α) parameter:

0 (standard BCD) < α < 1 (pure CD)

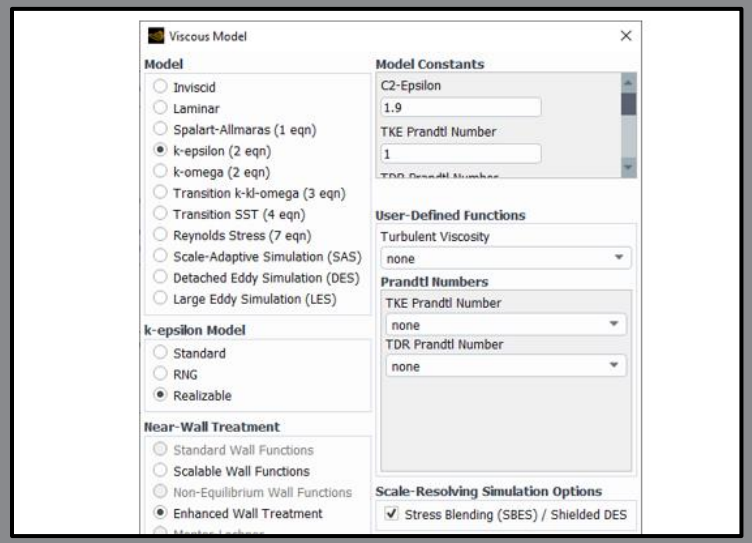
Benefit: Avoids standard BCD being too dissipative for LES



Algebraic Transition Model

The algebraic γ-Model solves zero transport equations (For reference : the γ-Re_θ model solves 2 additional transition equations, the γ-model solves one additional equation)

Benefit: Up to 7% CPU saving with similar accuracy



Enhanced Wall Treatment for SBES-RKε

Enhanced wall treatment for Stress-Blended Eddy Simulation & Realizable (k,ε)-model (SBES-RKε)

Benefit: Enhanced support for wall treatment (not longer beta)

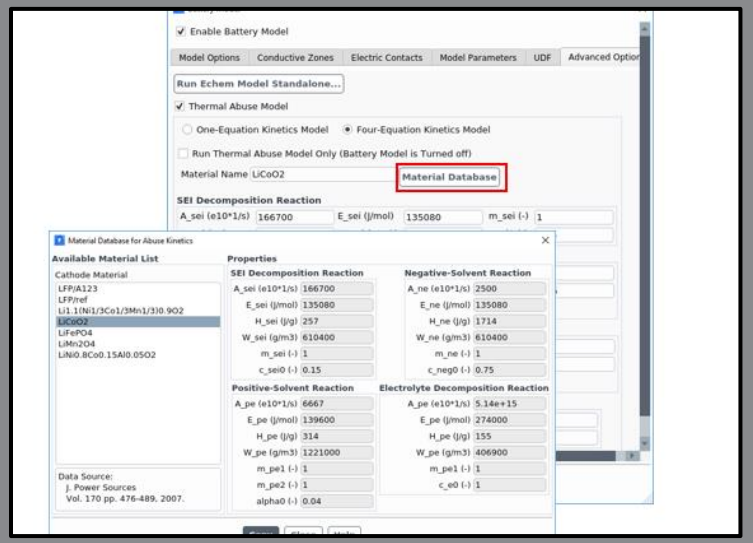
Batteries and Fuel Cells

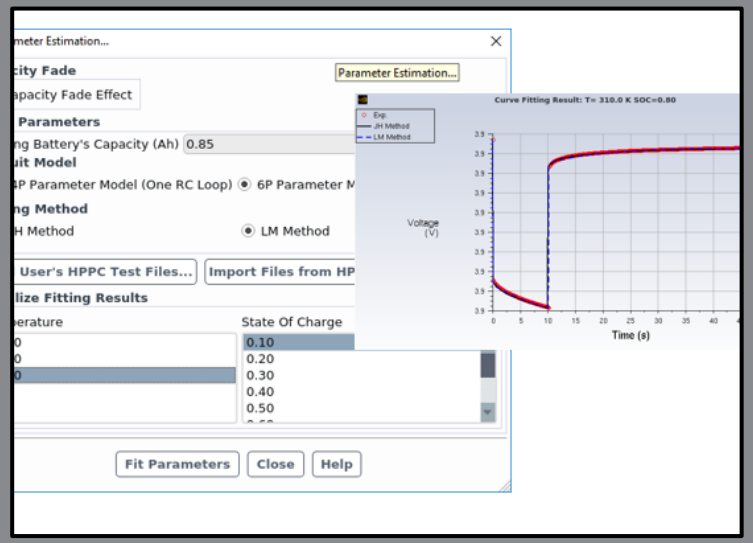
Ansys

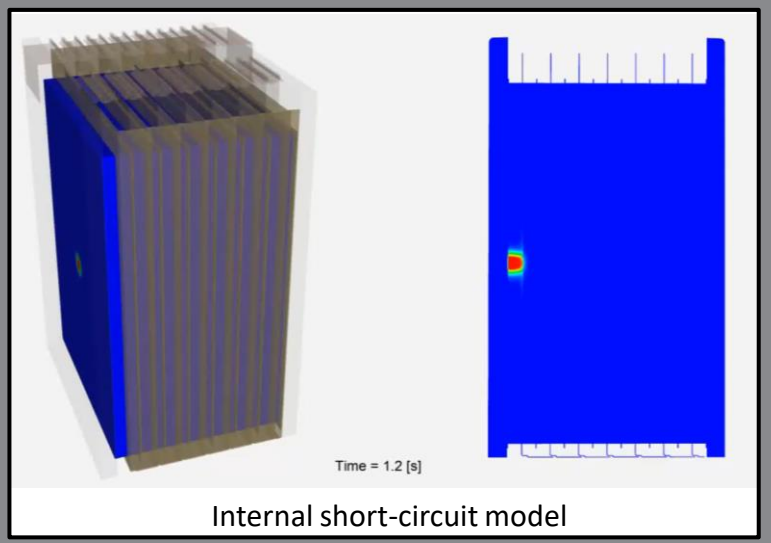
Ansys Fluent : Batteries and Fuel Cells

NEW

2021 R2 Release Highlights







Internal short-circuit model

Battery Material Property Library

Common material properties for Newman model (cathode, anode, and electrolyte materials). The library can be expanded by users

Benefit: Removes the burden of finding material properties in literature

Visualization and Postprocessing

Enables visualization of results from parameter estimation tool and related to Newman sub-model and thermal abuse model

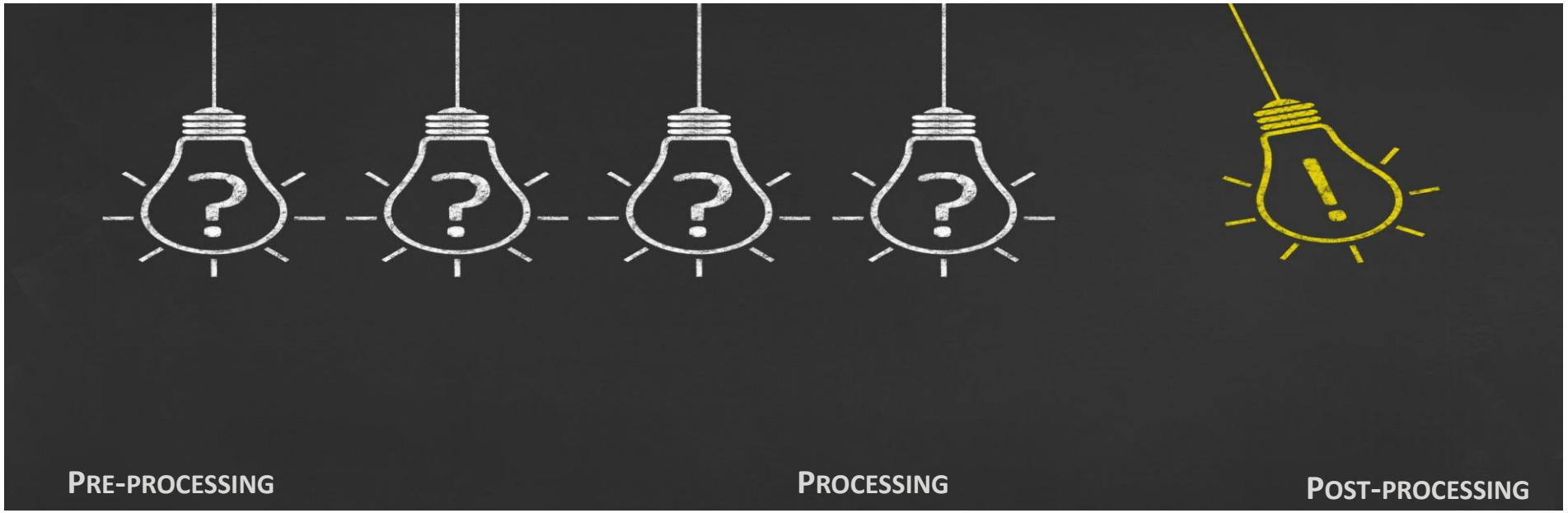
Benefit: Improved post-processing and visualization of additional results

Other Enhancements

- Dynamic cell clustering (*vs bounding box*)
- Internal short-circuit model
- Newman's P2D solver is more robust
- SOFC Fuel Cell Model Usability Improvements
- PEMFC Fuel Model Usability Improvements

Benefit: Improved productivity

FLUENT 2021R1/R2:
Questions?



CFD workstream

GEOMETRY PREPARATION → MESHING → RUN SETUP → RUN SIMULATION → POST PROCESSING

https://www.ansys.com/resource-center/webinar/improving-fuel-cell-designs-for-fcevs-using-simulation

Follow Ansys Fluids on LinkedIn

Like and comment.

Search: Ansys Fluids

Find Ansys Fluids:
linkedin.com/showcase/computational-fluid-dynamics