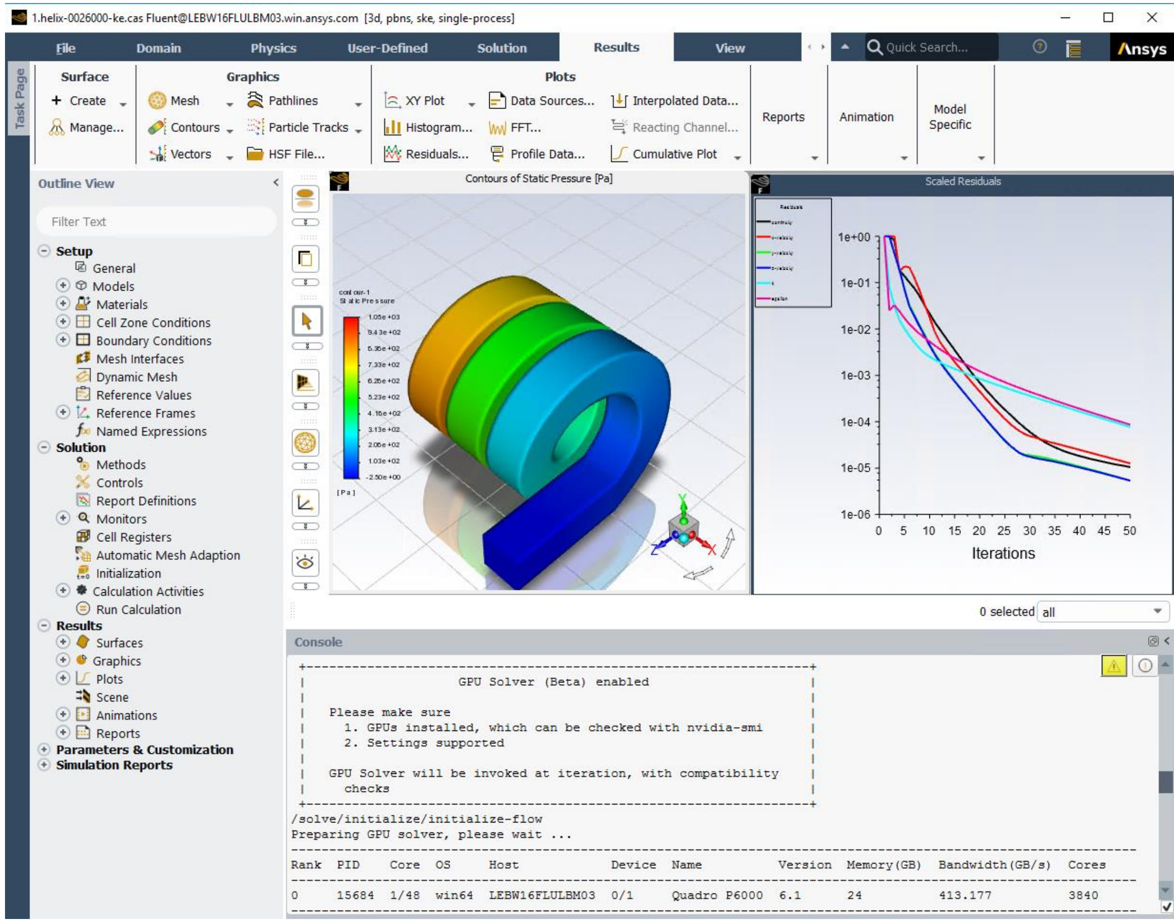
Release 2022 R1 Highlights
Ansys Fluent

Ansys

Fluent Multi-GPU Solver (Beta)

Multi-GPU Solver (beta) accelerates steady-state CFD simulations



Supported Capabilities:

- Single/multi-GPU (shared / distributed memory)
- Subsonic compressible flows
- Ideal gas
- Material with constant properties
- Turbulence: standard k-epsilon, GEKO k-omega
- CHT
- Porous media

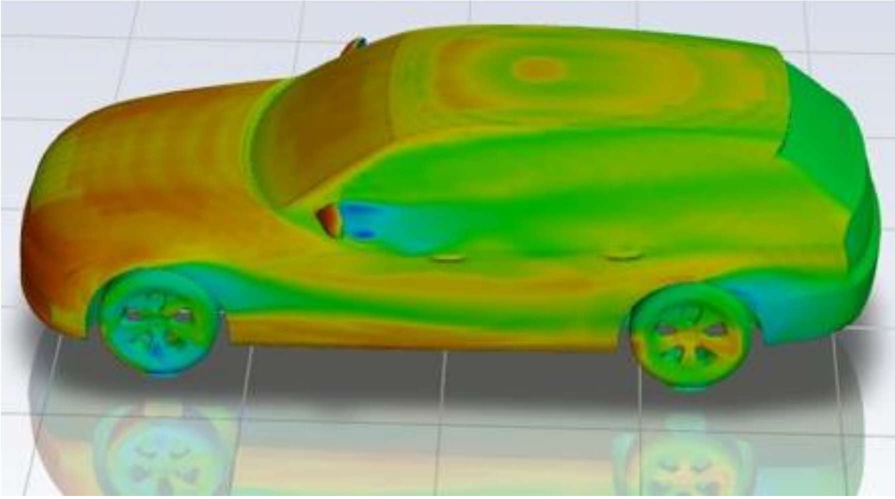
*7x cheaper hardware purchase cost and 4x lower power consumption**

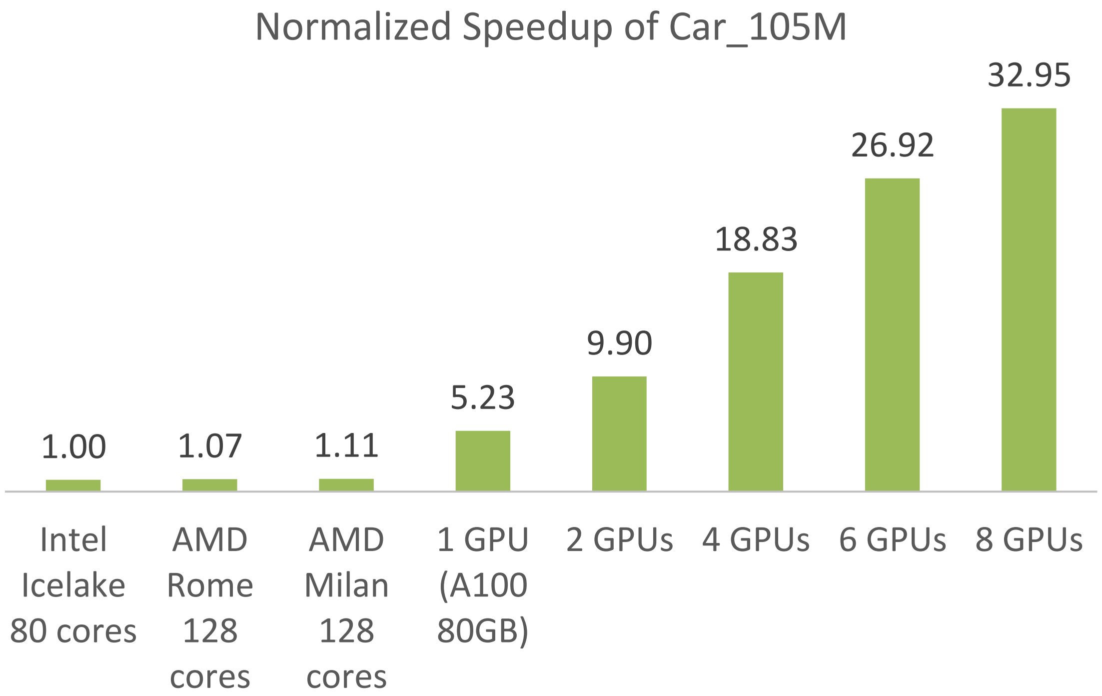
** 1024 core CPU cluster using 9600 W versus 6*V100 server using 2400 W*

Fluent Multi-GPU Solver (Beta) cont.

Strong scaling with parallel efficiency of 80% on automotive external aerodynamics case

- 105 million cell case, single precision with GEKO, runs on 1 A100 80GB card
  - ~1 iteration per 2 seconds, 20 minutes to converge such a case from initialization
- 1 A100 GPU ≈ 640 AMD Milan cores on 5 nodes
- 8 A100 GPUs ≈ 3840 AMD Milan cores on 32 nodes
- Parallel efficiency is 80% from 1 to 8 GPUs





Performance for Large Cases With Many Zones

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

- Example: Battery case - 24M cells, 32k face zones
  - Graphics Display ***up to 3X Faster***
  - Various command executions ***up to 20X Faster***
  - Bottlenecks removed in display of many dialog boxes
- Example: CHT – 64M cells, 22k face zones
  - Case read 2X faster compared to 2021 R2
- Combined, some scripted case-setup workflows are ***up to 6X Faster***

Operation	Percentage Improvement (Average of serial and parallel on 8 cores)
Mesh Display	73%
Contour Display	83%
Vector Display	80%

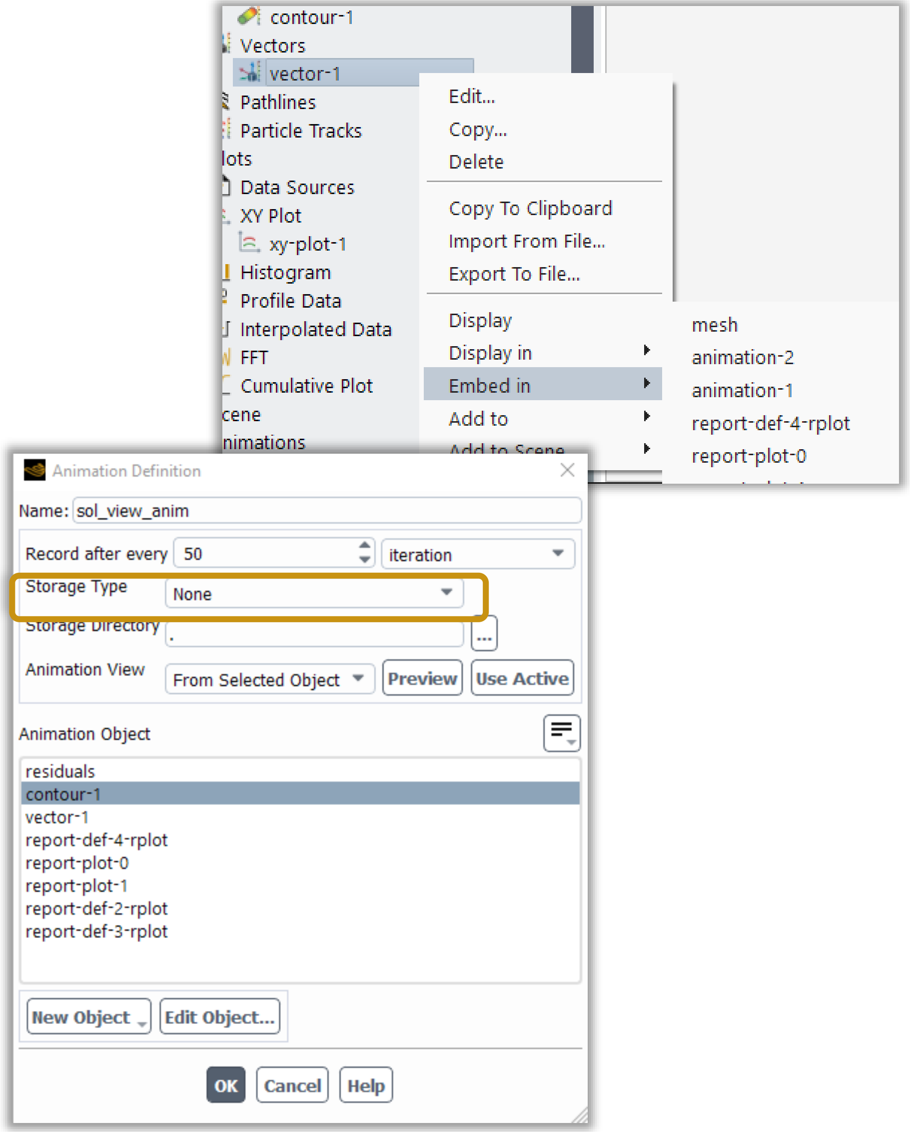
Battery test case: 24M cells, 32k face zones

	Time (s)		Improvement	
	21.2	22.1	(seconds)	(%)
Time taken to turn on energy	12.72	5.89	6.83	53.67
Time taken to open Turbulence panel and apply change	26.41	16.61	9.8	37.13
Time taken to open Battery Model and OK	224.76	201.04	23.72	10.55
Time taken to open Fluids material panel and change/create	0.77	0.7	0.07	8.7
Time taken to open Solids material panel and change/create	0.26	0.22	0.04	14.89
Time taken to open fluid cell zone panel and apply	74.16	64.83	9.33	12.58
Time taken to open solid cell zone panel and apply	20.56	12.32	8.24	40.1
Time taken to open wall BC panel and apply	20.48	14.67	5.81	28.4
Time taken to create solid zone surfaces	18.37	17.33	1.04	5.63
Time taken to open mesh object panel	12.86	6.28	6.58	51.14
Time taken to open contour object panel	3.98	0.26	3.72	93.58
Time taken to open vector object panel	7.65	0.21	7.44	97.31
Time taken to open pathline object panel	7.72	4.71	3.01	38.99
Time taken to open particle object panel	0.31	0.28	0.03	7.27
Time taken to open surface area report definition panel	19.69	0.73	18.96	96.3
Time taken to open surface custom vector based flux report definition panel	8.09	0.32	7.77	96.02
Time taken to open surface custome vector weighted avergae report definiti	10.97	0.29	10.68	97.32
Time taken to open surface facet average report definition panel	8.15	0.32	7.83	96.11
Time taken to open surface facet maximum report definition panel	11.92	0.81	11.11	93.19
Time taken to open surface flow rate report definition panel	8.26	0.34	7.92	95.83
Time taken to open surface integral report definition panel	11.34	0.25	11.09	97.8

Embedded Window and Animation Enhancements

Workflow improvements

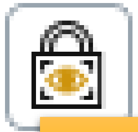
- Embedded Windows
  - Exposure in Outline View context menus
  - Not limited to Reserved windows
  - Placeholder frames can be embedded before start of simulation
  - Journal support
  - Animation frames not stored by default when using **Automatically embed residuals during calculation**
- Animation options
  - **None** option for **Storage Type:** render periodic visualization updates during solution without any memory / file use
    - Precludes later playback / recording



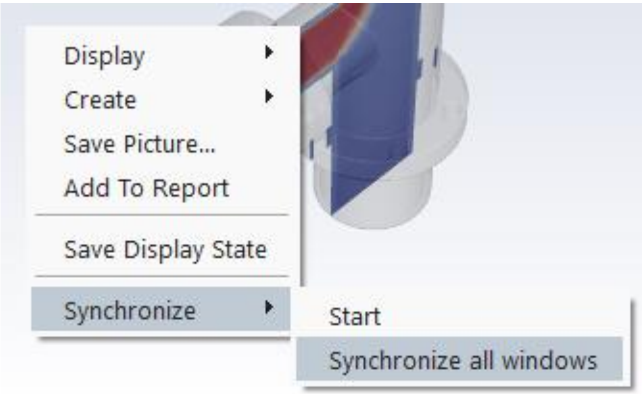
View Synchronization

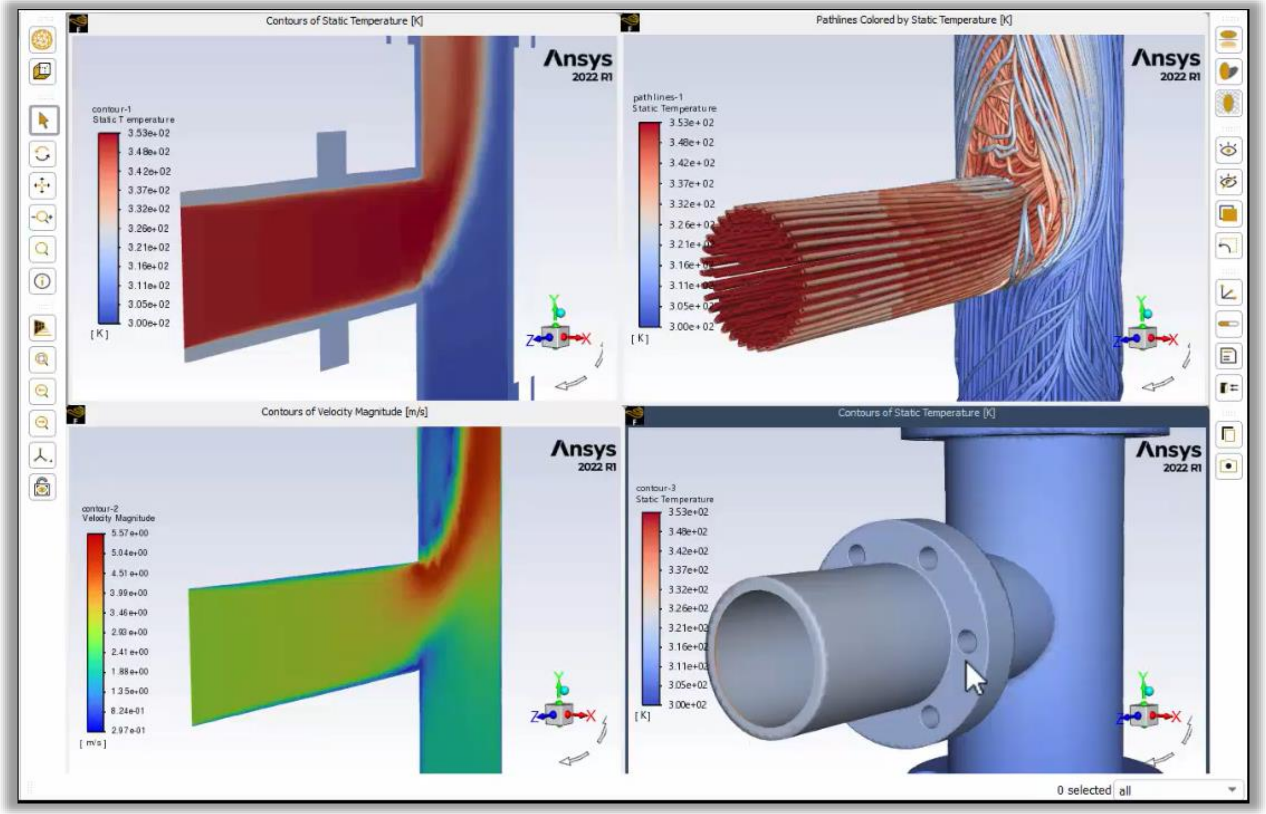
Lock view orientation among multiple windows for visual comparison

- Simultaneously view multiple visualizations from a consistent viewpoint
- Synchronize all sub-windows, or only selected sub-windows
- Accessible from toolbar icon or graphics window context menu:



Click to synchronize all windows





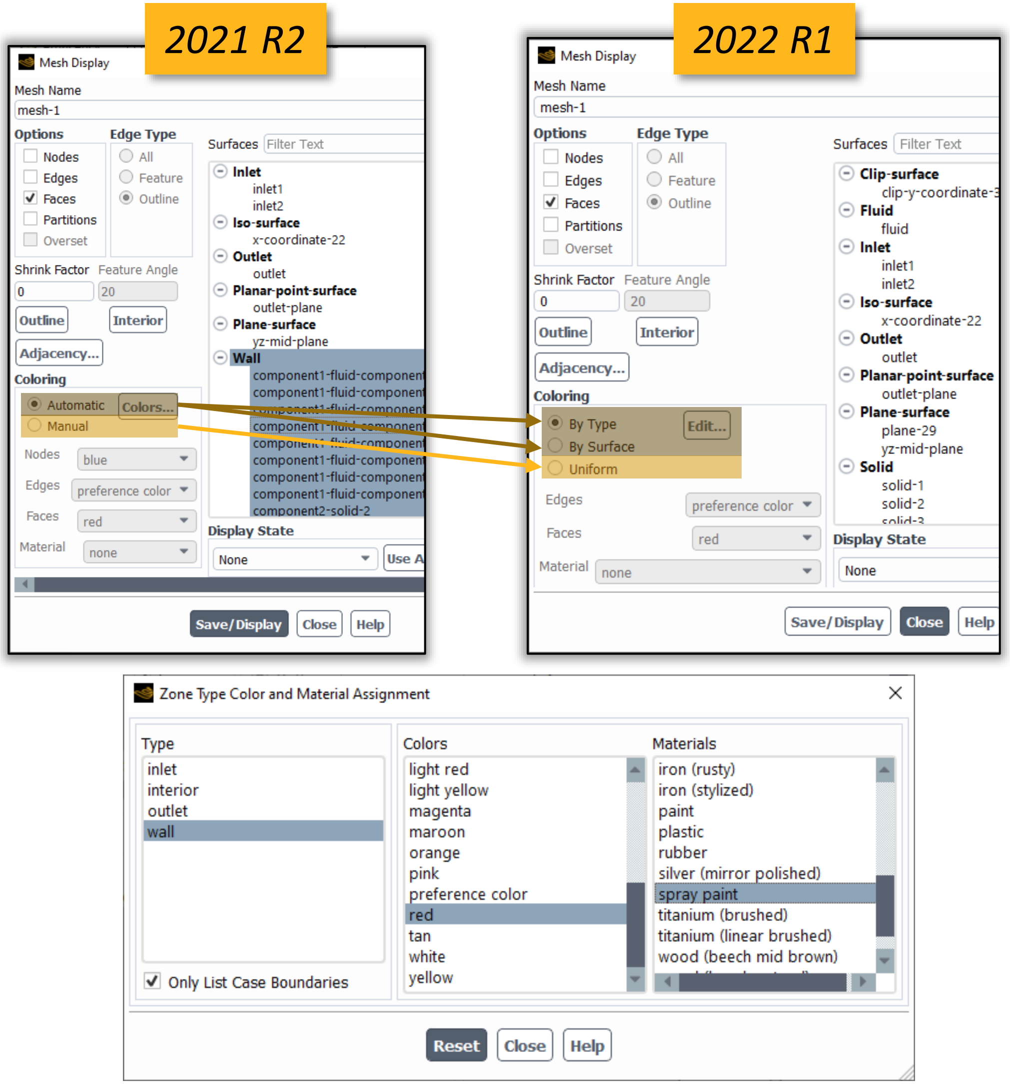
Video showing viewport synchronization

Mesh Rendering Enhancements

New materials for rendering and added flexibility for Mesh Display objects

- Many new solid materials available for rendering (brick, concrete, etc.)
- Choice of color **by ID** vs **by Type** are now stored for each Mesh Object
  - **Automatic** option split into **By Type** and **By Surface** (analogous to by ID)
  - **By Type** can be used to configure both color and material choices

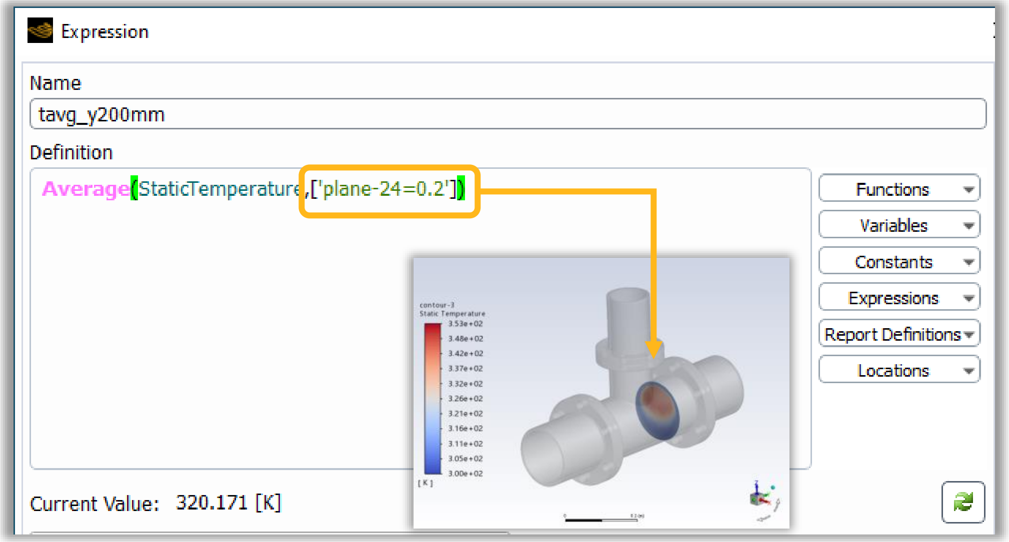
    ⇒ **Removes requirement that same material must be used for all surfaces in a mesh display object**
  - **Manual** renamed to **Uniform**

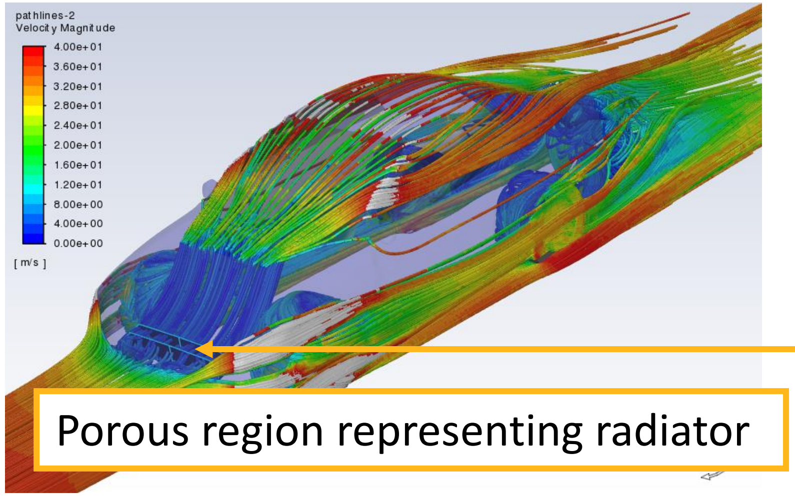


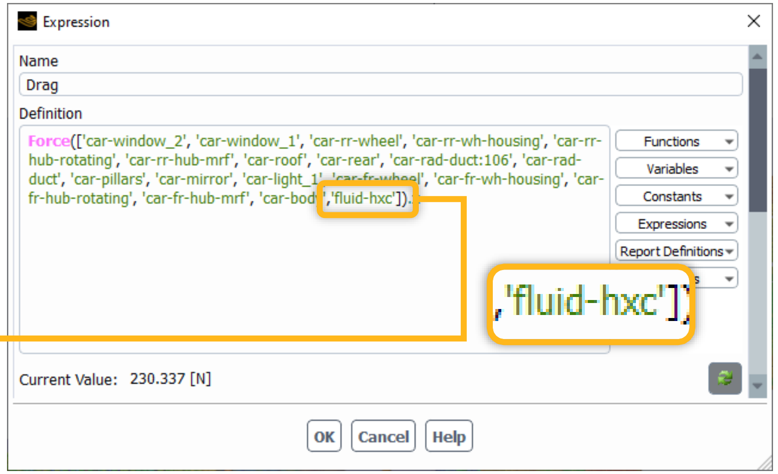
Expressions Enhancements

Support for User-Defined Surfaces and Porous Region Contribution to Forces

- User-defined surface(s) can be included in the **Location** for reduction functions (Average, Minimum, etc)
- Porous region contributions to forces
  - E.g., aero cases with radiators





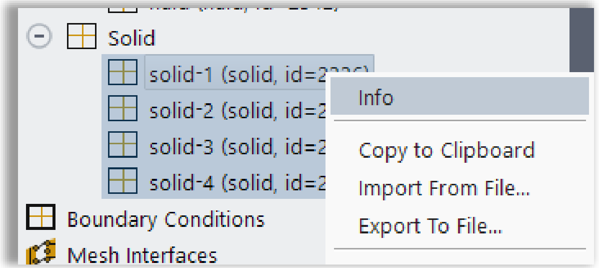


Ansys

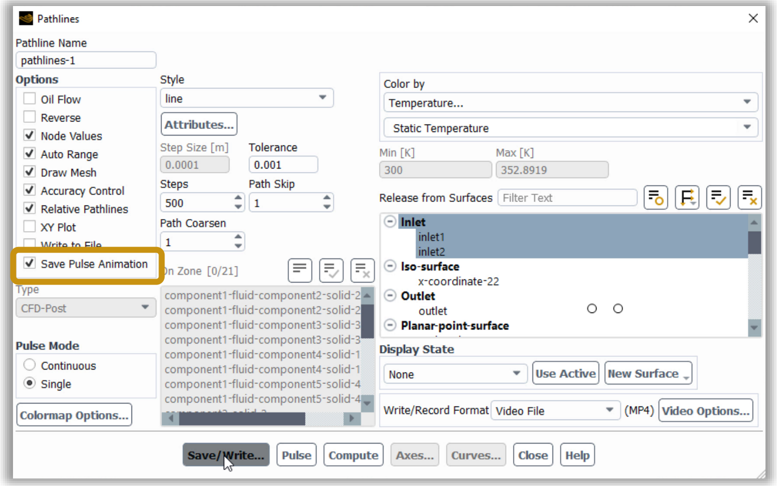
Miscellaneous Usability Enhancements

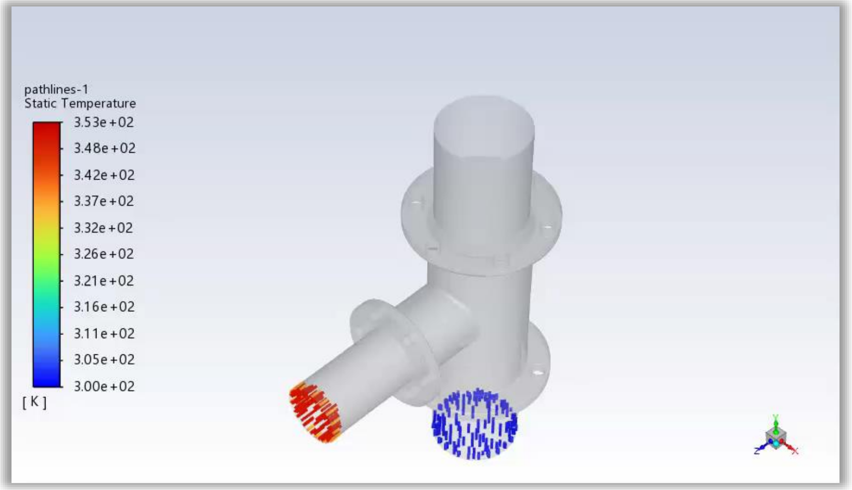
- *Easier access to cell-zone details including quality statistics*
- *Recordable pathline animations*

- Right-click → Info on cell zone(s) in **Outline View** to print cell quality statistics in console
- Pulsed pathline animations can be saved in common video formats (MP4, AVI, FLV, MOV, MPEG)



Cell Zone Name	Cell Count	Minimum Orthogonal Quality	Orthogonal Quality Below 0.1
solid-4	8083	0.20356136	0
solid-3	6293	0.32538144	0
solid-2	20388	0.20182542	0
solid-1	6322	0.27628726	0



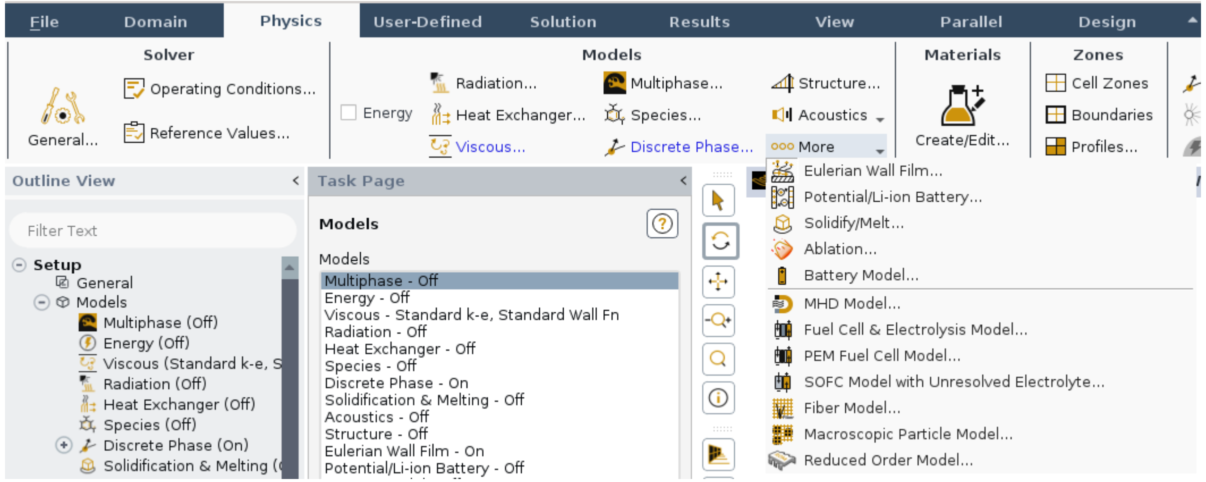


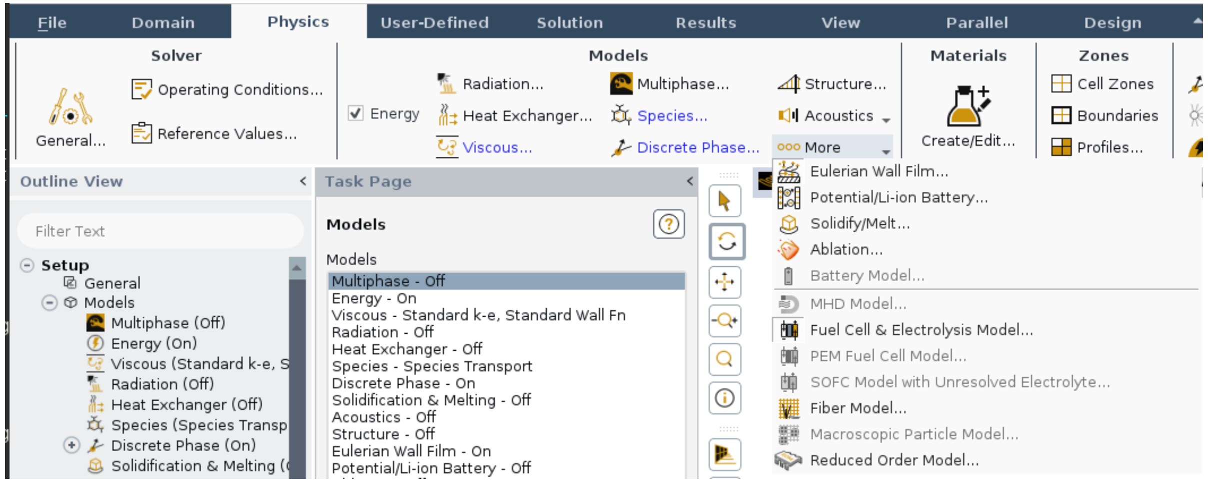
Video showing pulsed pathlines

Exposure of Add-On Modules in Ribbon

Easier access to add-on modules from GUI

- Provide access to supported add-on modules in the ribbon
  - Simplifies access
  - Includes new defined icons
  - TUI is maintained for scripting
- Added compatibility checks of add-on modules where possible
  - Incompatible options are grayed out





New Share Topology Method in Watertight Workflow

Offers potentially faster and more robust method than Join-Intersect

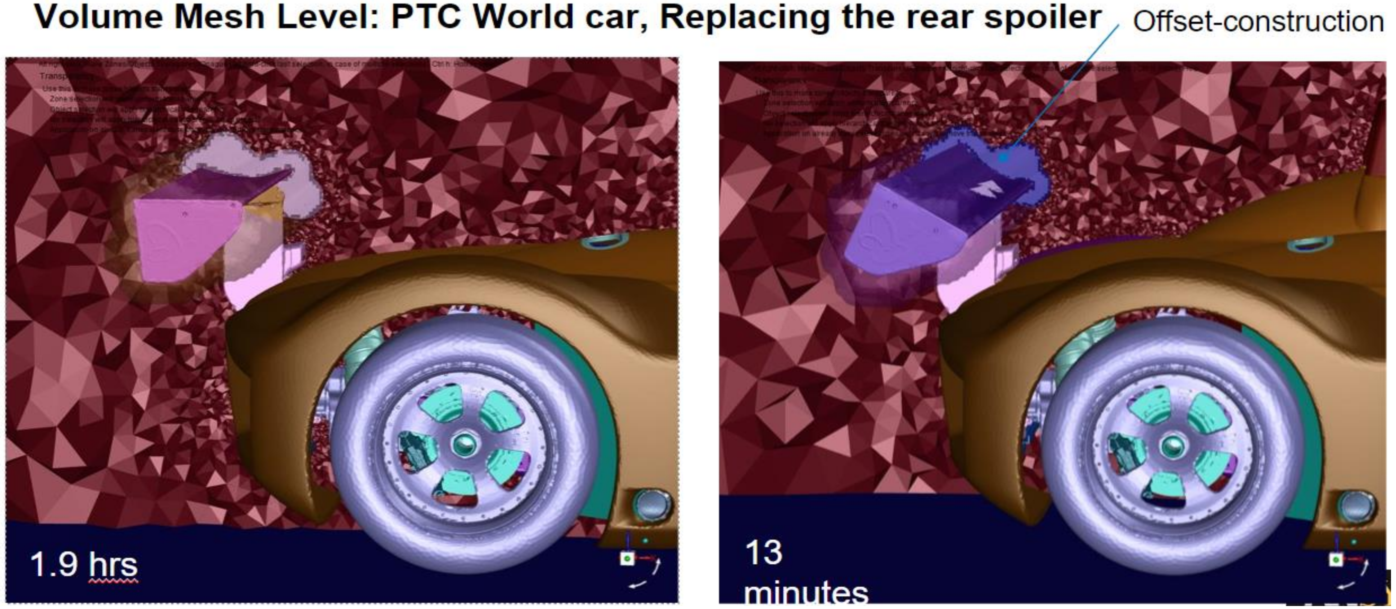
- New **Interface Connect** method for **Apply Share Topology** task
  - Connects edges of overlapping face pairs (rather than intersecting faces like **Join-Intersect**) which can be faster / more robust
  - Three modes:
    - **Automatic - Using Connect Topology:** use the labels created by SpaceClaim if the Force Share feature was used.
    - **Manual:** user manually selects the interface labels from the list of available labels.
    - **Automatic:** automatically separate face zones, identify overlapping faces, and assign the interface connect labels. Useful when connect topology has not been utilized in SpaceClaim or if the mesh was obtained from another source.

Fault-Tolerant Meshing Part Replacement

Efficiently add, remove or replace geometry objects of your CAD model without having to re-mesh the complete model

- Users can replace/add/remove parts of CAD model
- Two approaches are implemented which reduces the time to study the design variation by many times
  - Volume-mesh level
  - Surface-mesh level
- Supports all mesh types

Volume Mesh Level: PTC World car, Replacing the rear spoiler

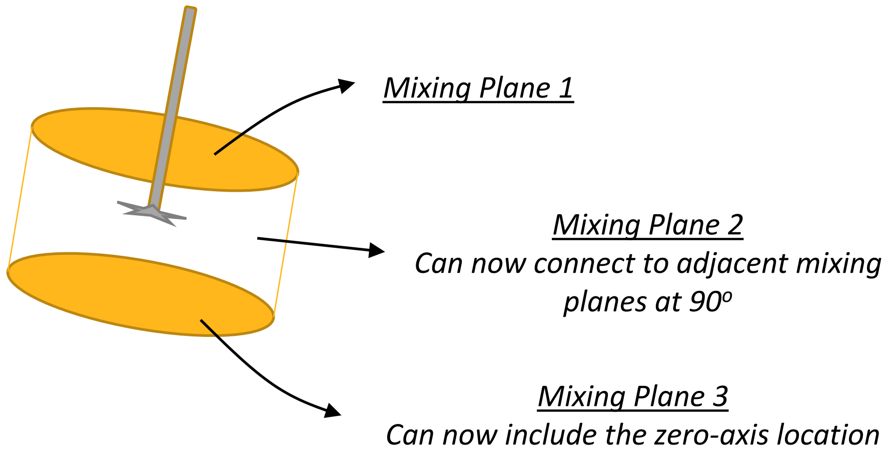


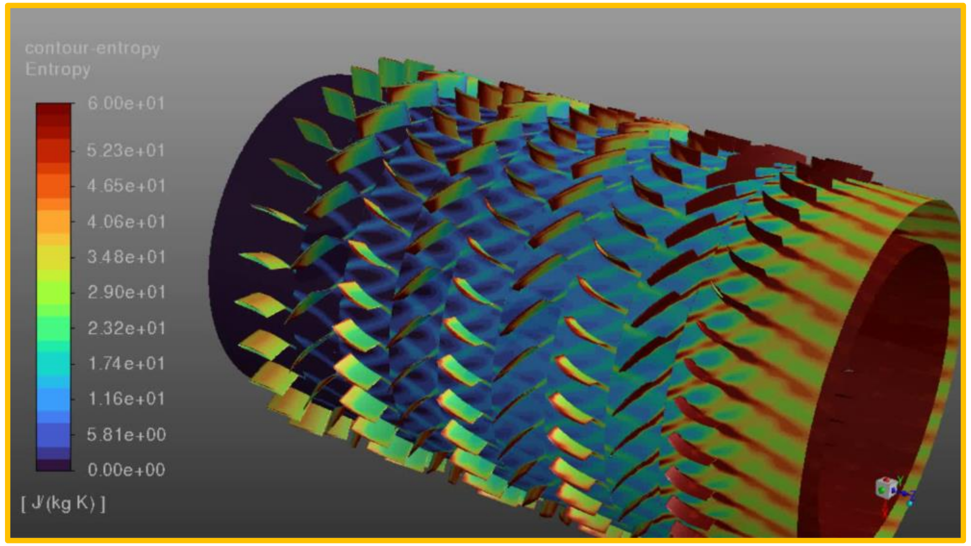
Blade Row Interface Improvements

Improved Results and Extended Applications for Generalized Turbo Interface (GTI)

- Mixing plane formulation improvements
  - Consolidation of the two previous methods (*intersection-based* and *side-based*)
  - Handles more turbo/rotating machinery geometry
  - Improved conservation
  - Improved performance on radial machines

- Tip gap interface matching
  - Tip gap interface created from within **Turbo Create** applies a matching condition
    - Avoids extra empty non-overlapping walls



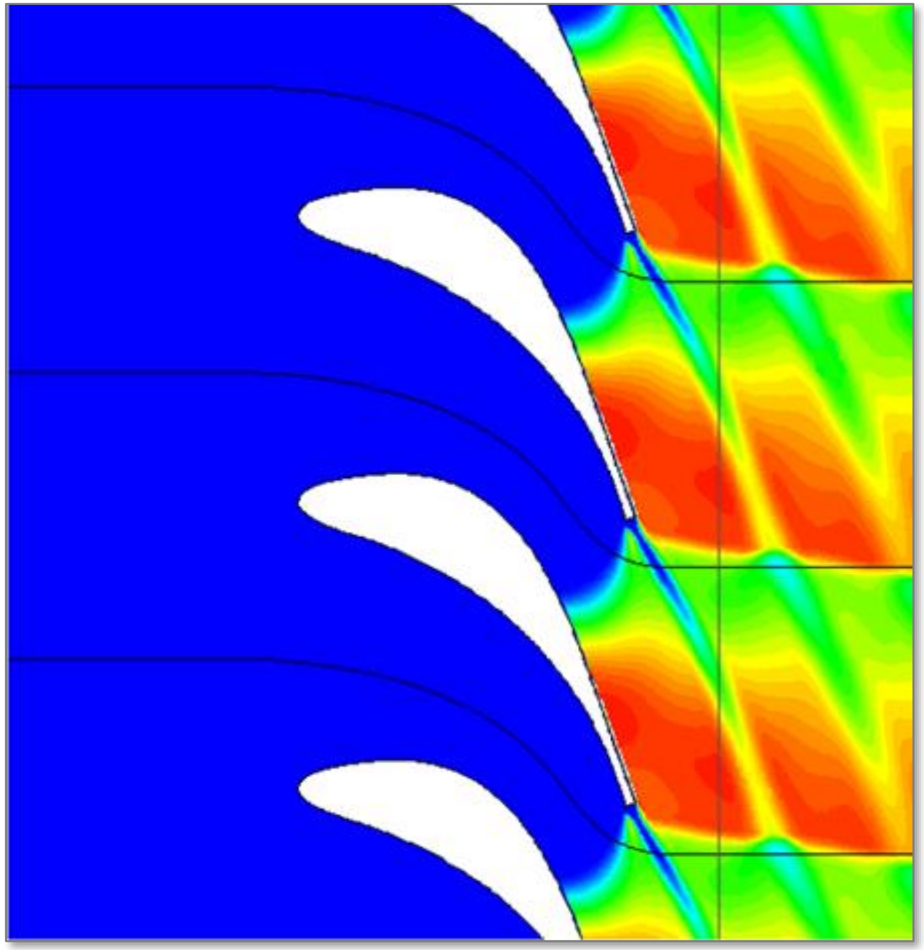


Hannover 4.5 Stage Compressor (animation)
Transient simulation using Pitch Scaled GTI single-passage per row & periodic-instancing

Non-Equilibrium Wet Steam Model

Access to IAPWS97 provides industry standard steam properties

- Real Gas Property (RGP) tables for wet steam model
  - Alternative to the built-in thermodynamic wet steam properties, using the industry standard IAPWS97
    - Same format as used in the CFX real gas model
  - Built-in RGP file for steam available:
    - Turn on Turbo mode, turn on wet steam model
    - Read RGP file: `file/table-manager/read-rgp-file`
    - Link the RGP to wet steam model: `define/models/multiphase/wet-steam/set/rgp-tables`
- Enhancements to Non-Equilibrium Wet Steam
  - Convergence improvements for 2nd-order wet steam model
  - Alternate stagnation condition computation available:
    - The default method is based on mixture of vapor and liquid droplets
    - The alternative is based on vapor gas phase (similar to CFX)
    - Accessed via TUI: `define/models/multiphase/wet-steam/set/stagnation-conditions`

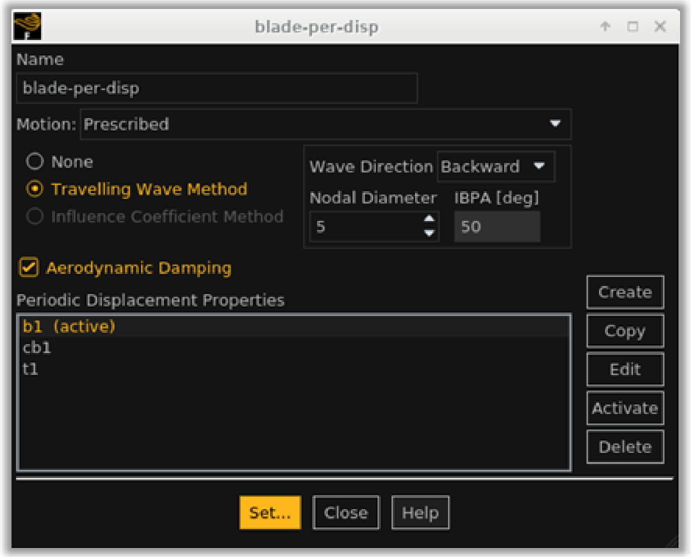


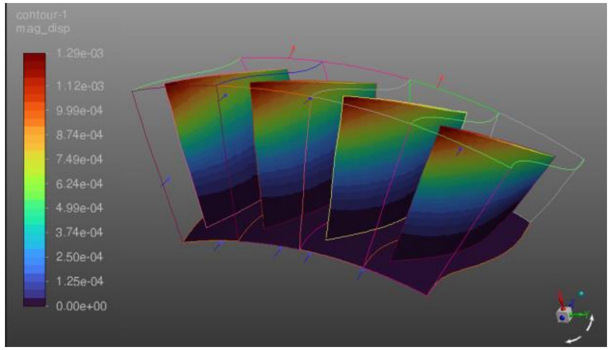
Liquid mass fraction in a stationary cascade of steam turbine blades

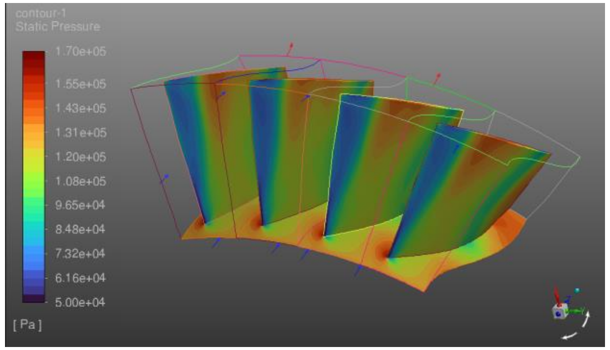
Aeromechanics: Aerodynamic Damping/Periodic Displacement

Usability Improved With New Periodic Displacement GUI

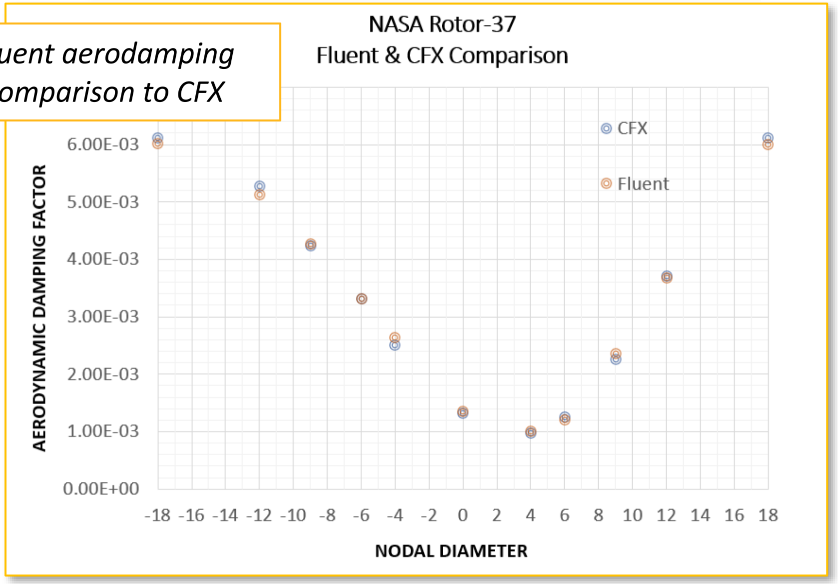
- Read and set up multiple mode-shape profiles which can be activated selectively to test aero-damping of different modes
- Simulate a range of 1-way FSI problems
  - **Traveling Wave Method (TWM)** for blade-row aerodamping
  - **None** option for turbo or non-turbo applications
    - E.g. transient simulations using periodic displacements







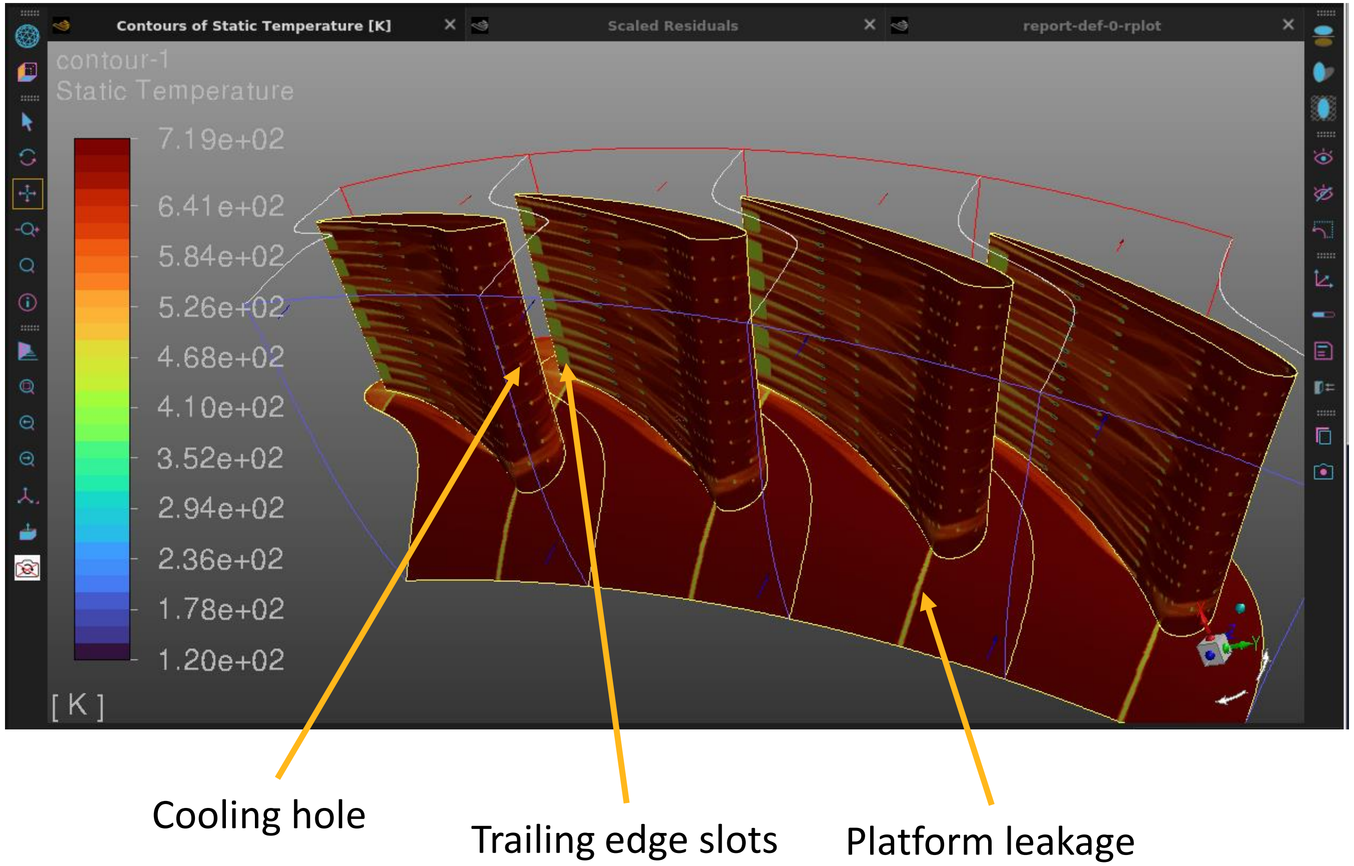
Fluent aerodamping comparison to CFX



Blade Film Cooling Model for Gas Turbines

Use Virtual Geometry & Boundary Conditions to Simulate Turbomachinery Blade Cooling

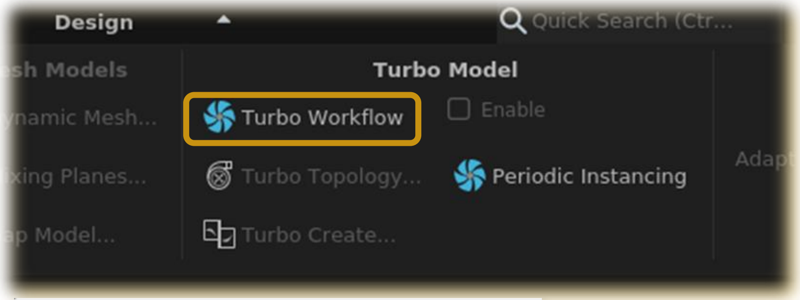
- Hole coordinates and parameters are imported from profile files to create virtual geometry
  - Rectangular or circular holes oriented normal-to-surface or along specified direction
- New **Boundary Interface** intersects virtual geometry with boundary surface to form virtual boundary conditions
  - Similar to creating conventional mesh interfaces
  - Overlapped intersection becomes mass-flow-inlet or mass-flow-outlet
  - Non-overlapped intersection retains underlying surface settings
- Requires Turbo Models to be turned on

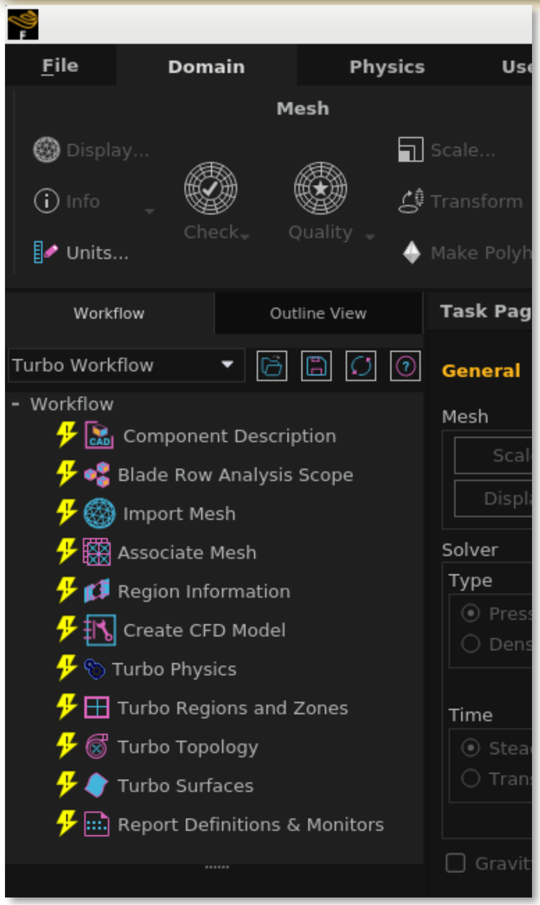


Turbo Workflow

Quickly set up turbomachinery flow problems and instrument primary performance parameters and post processing

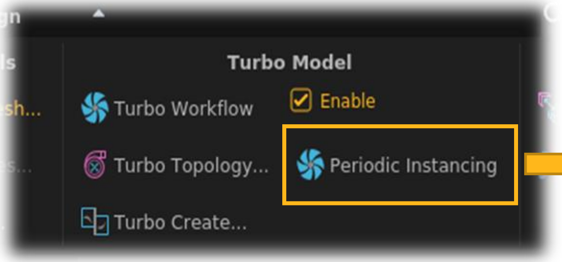
- Guided task-based approach
  - Eliminates repeated user inputs and minimize user error during setup
  - Supports reverting and editing the setup
- Features:
  - Setup for Axial/Radial Compressors & Turbines
  - Can read multiple mesh files (.msh, .def, .gtm)
  - Copy, rotate, and stitch fluid zones
  - Turbo coordinates for turbo post processing
  - Performance monitors

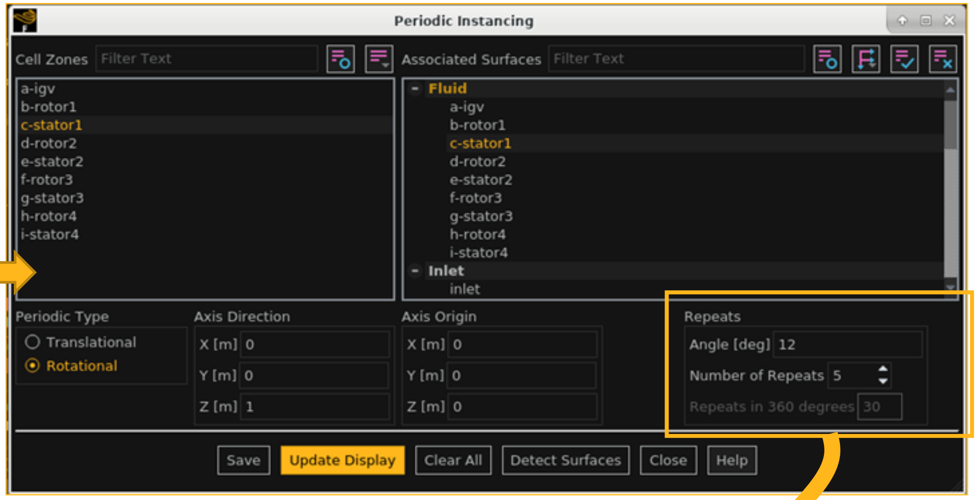


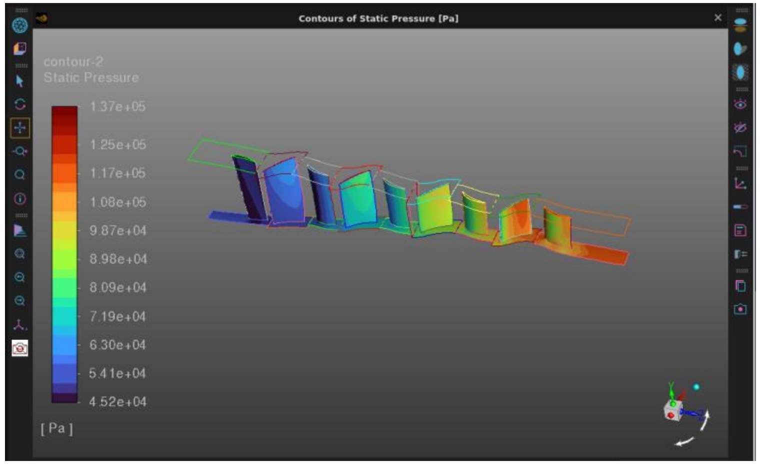


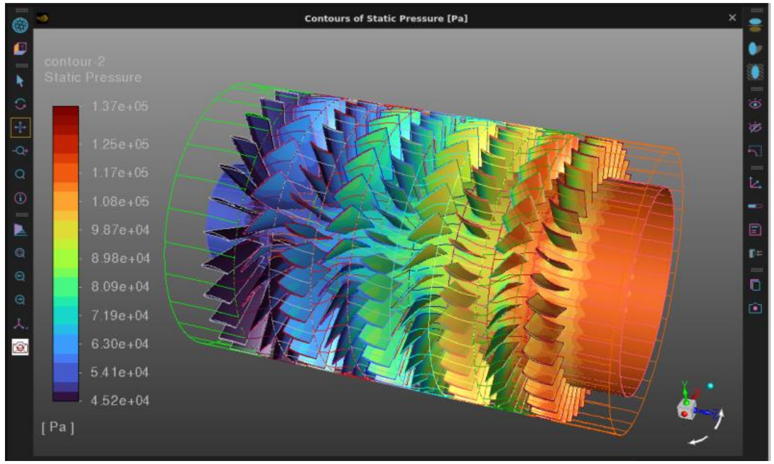
Periodic Instancing

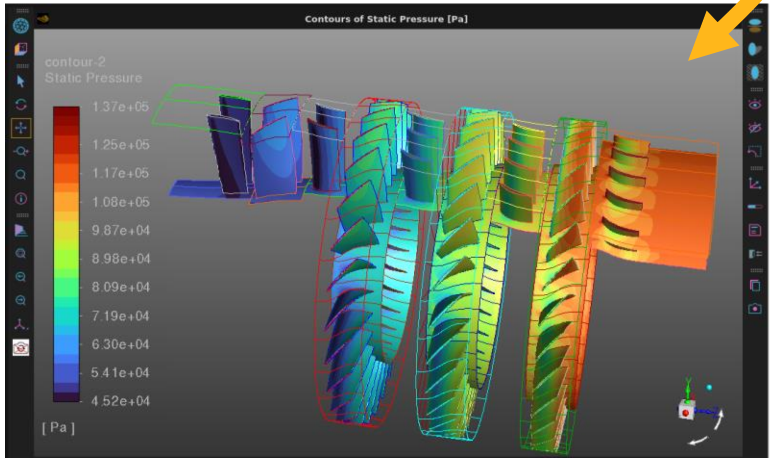
- Enhanced (and renamed) Periodic Repeats feature
- Improve post-processing for most turbomachinery applications











Clear All → Single Repeat

- Detect surfaces associated with each zone*
- Sets the view model to ***single repeat***

Detect Surfaces → Full Wheel

- Detect surfaces associated with each zone*
- Sets the view model to ***full wheel***

Or use the Repeats settings

- Select cell zone and enter the number of repeats
  - Positive value: forwarded instancing
  - Negative value: backward instancing

**If turbo surface cut spans more than one zone it will be ignored*

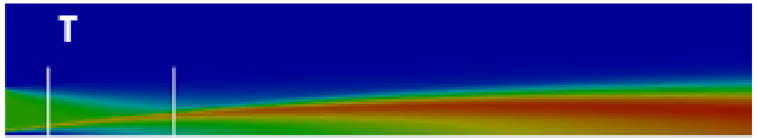
Improved Finite-Rate Chemistry (FRC) Combustion

Improved accuracy for mild combustion and supersonic combustion

- Reacting flows using the standard Eddy Dissipation Concept (EDC) for turbulence-chemistry interaction are known to over-predict reaction rates for mild combustion and under-predict the rates for supersonic combustion
- The new Partially-Stirred Reactor (PaSR) model addresses these weaknesses
  - Note that this model uses a partially stirred reactor concept – it is not just a model for partially stirred reactor applications

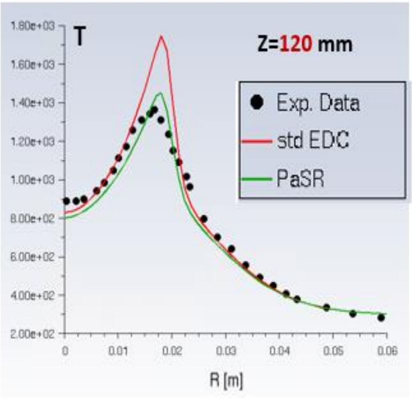
Mild Combustion Validation Case:
Adelaide Jet in Hot Cross Flow with O2 dilution

fuel jet: CH_4/H_2 305K
hot co-flow: 1300 K
cold air: 294 K

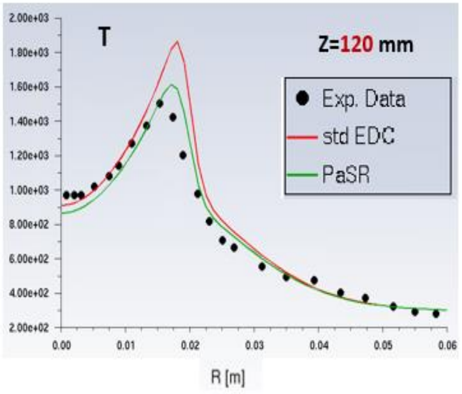


Z=30 mm Z=120 mm

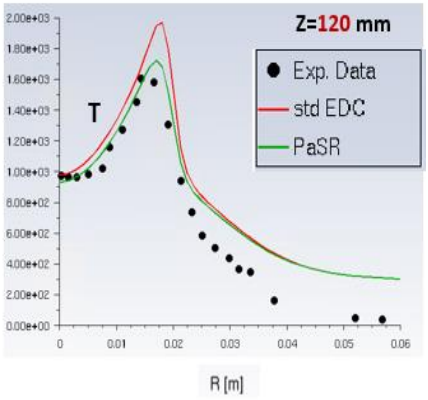
3% Y_{O2}



6% Y_{O2}



9% Y_{O2}

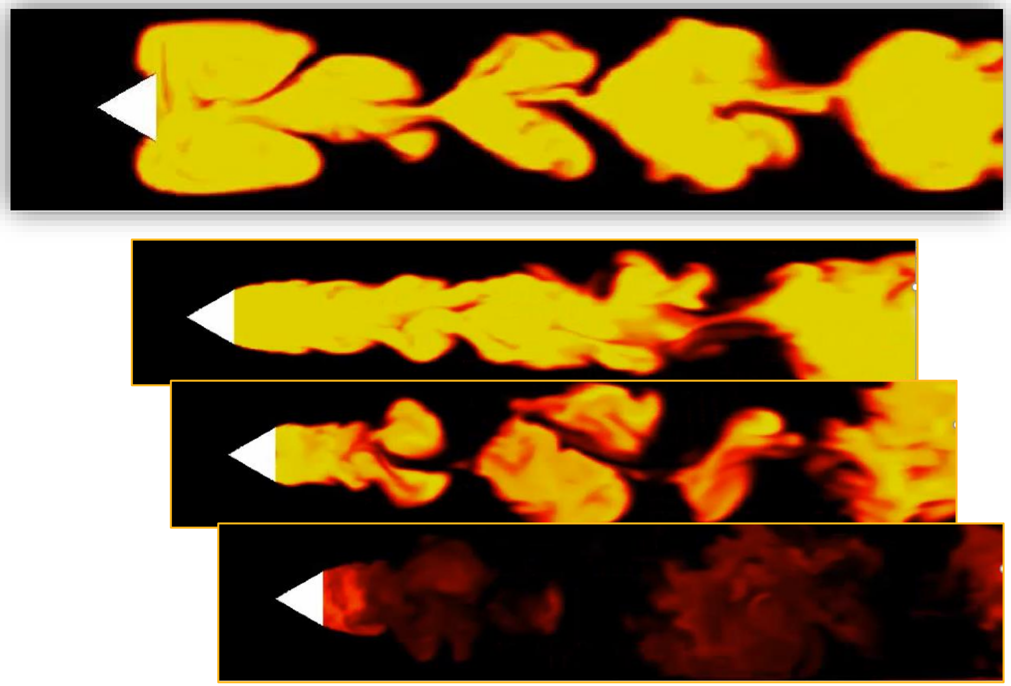


"Structure of Turbulent Non-Premixed Jet Flames in a Diluted Hot Crossflow", Proceedings of the Combustion Institute, Vol. 29, 2002, pp 1147-1154, B. B. Dally, A. N. Karpetis, R. S. Barlow.

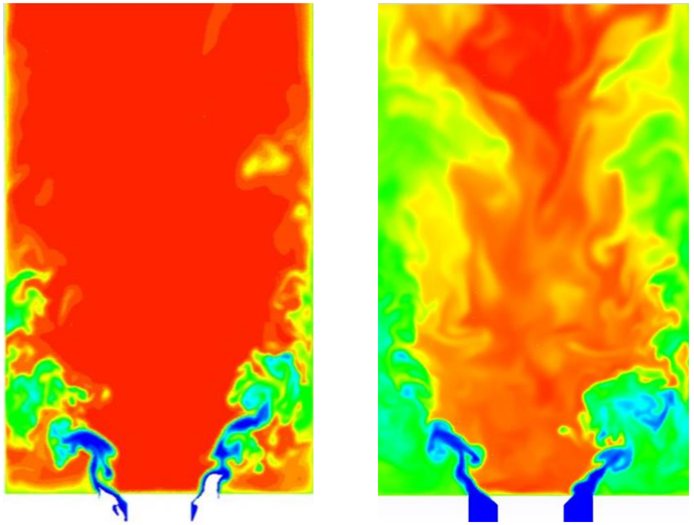
Non-adiabatic Strained FGM Combustion Model

Account for the effect of heat loss and strain on flame speed with the Non-adiabatic strained FGM model

Lean blow-off test case with propane using the non-adiabatic strained FGM model. Accounting for high strain is important to accurately predict the blow-off equivalence ratio



Cambridge Lean Premixed Burner: Lean Blow Off (LBO) Prediction



Unstrained FGM

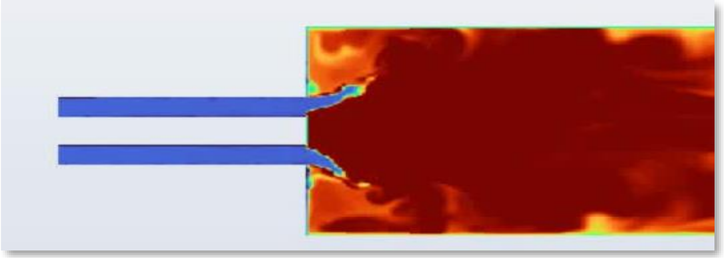
Strained FGM

Starting from a burning solution (above LBO), the mixture fraction is reduced below the LBO limit. Unstrained FGM shows a sustained flame whereas strained FGM correctly shows quenching of the flame. Lean conditions are very sensitive to reduced flame speed at increasing strain rate.

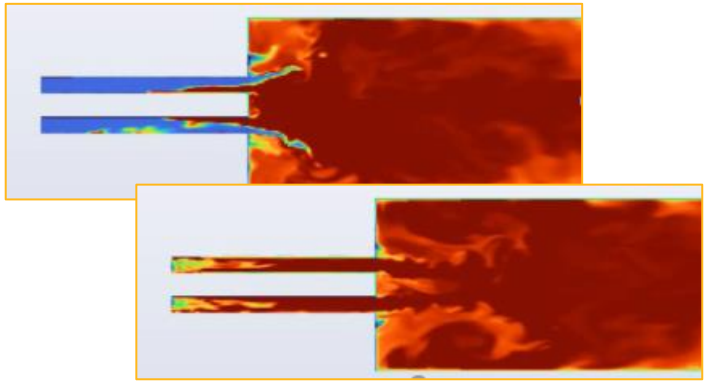
Hydrogen Combustion

Validating Fluent for hydrogen combustion

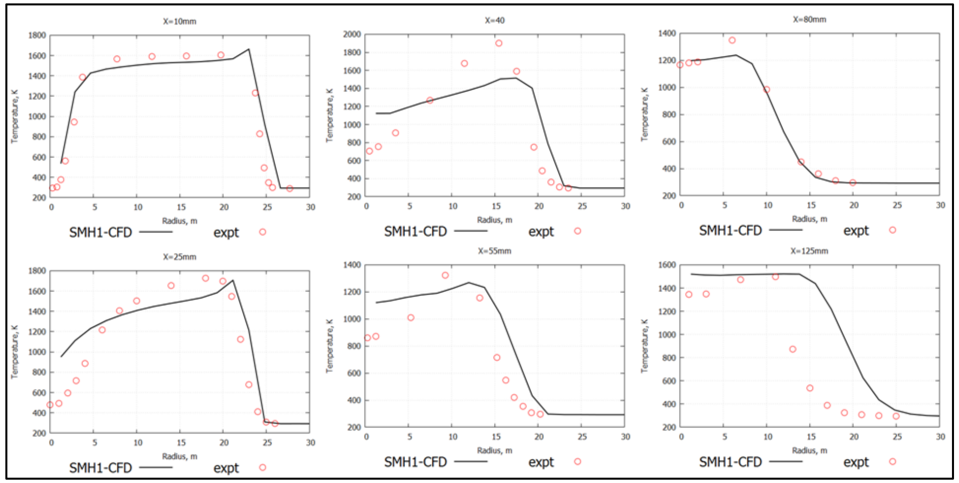
Flame flash back with H_2 . This also uses the non-adiabatic strained FGM model, which accounts for flame cooling in the nozzle and its impact on flame speed.



Flame flash back animation (100% H_2)



Swirl stabilized flame validation ($50\%\ H_2$, $50\%\ CH_4$), confirming the accuracy of FGM for this application

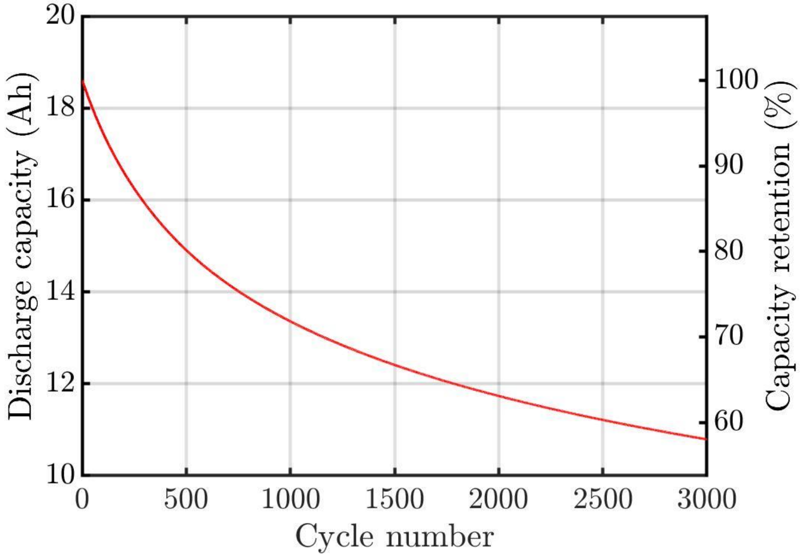


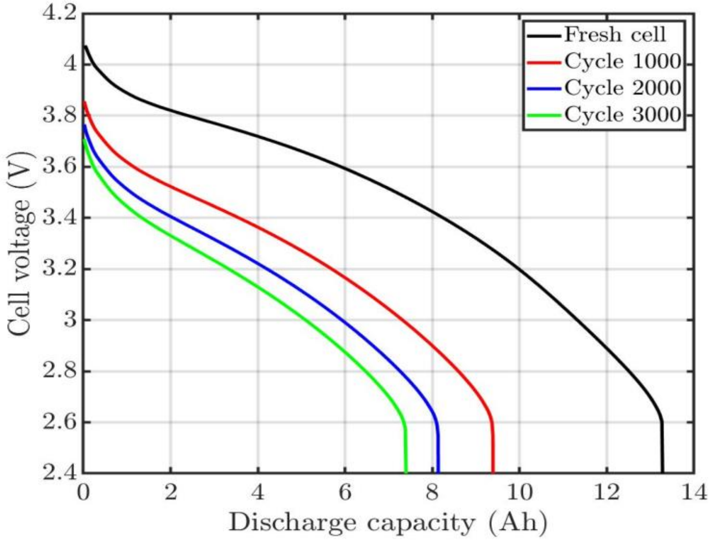
Temperature profiles vs experiment

Battery Life Modeling

Physics-based ageing model predicts battery's capacity fade from first principles

- A new physics-based battery life model provides deeper insights into capacity fade
  - Accounts for ageing effects using fundamental electrochemistry instead of an empirical model
  - Captures Solid-Electrolyte Interface (SEI) layer growth and lithium plating
  - Compliments the existing empirical battery life model





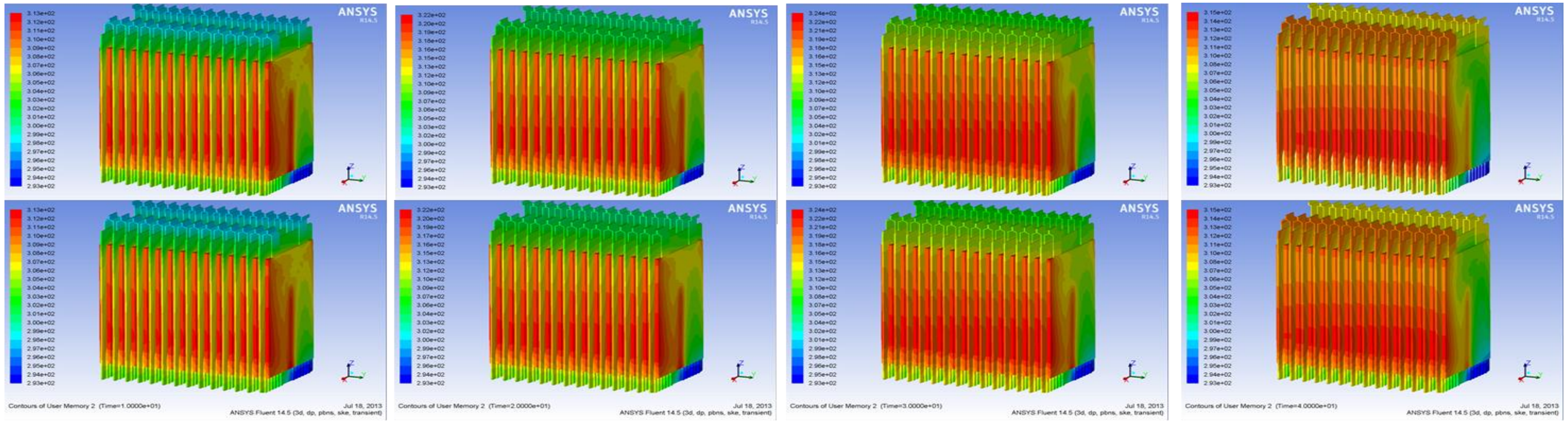
Battery Reduced Order Models (ROMs)

Streamline training data creation for Ansys TwinBuilder ROMs

- Ansys TwinBuilder includes LTI and SVD ROMs that provide 3D transient thermal results in seconds
  - E.g. battery drive cycle simulations
- Fluent simulations provide the training data to teach the ROMs
- Fluent now includes LTI and SVD ROM tool kits that automate training data creation, greatly reducing hands-on time to create training data

Transient CFD

SVD ROM

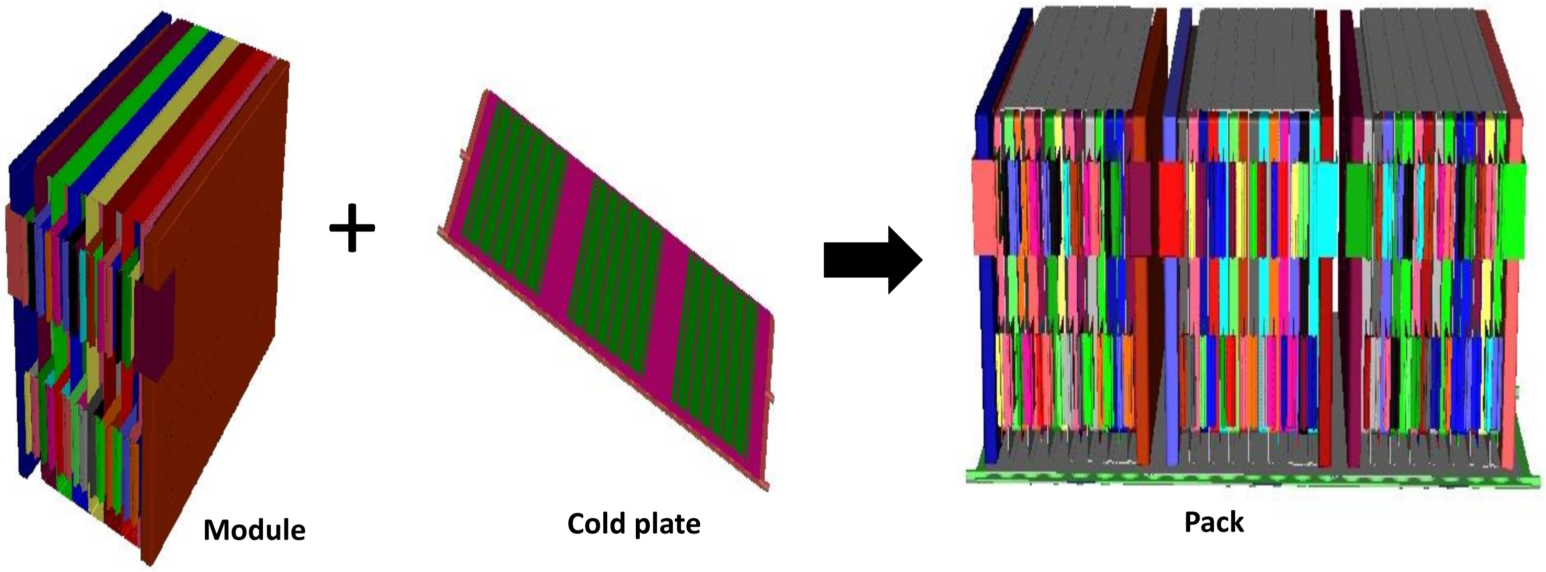


10 sec *20 sec* *30 sec* *40 sec*

Battery Pack Builder Tool

Battery Pack Builder allows you to capture module-to-module variation

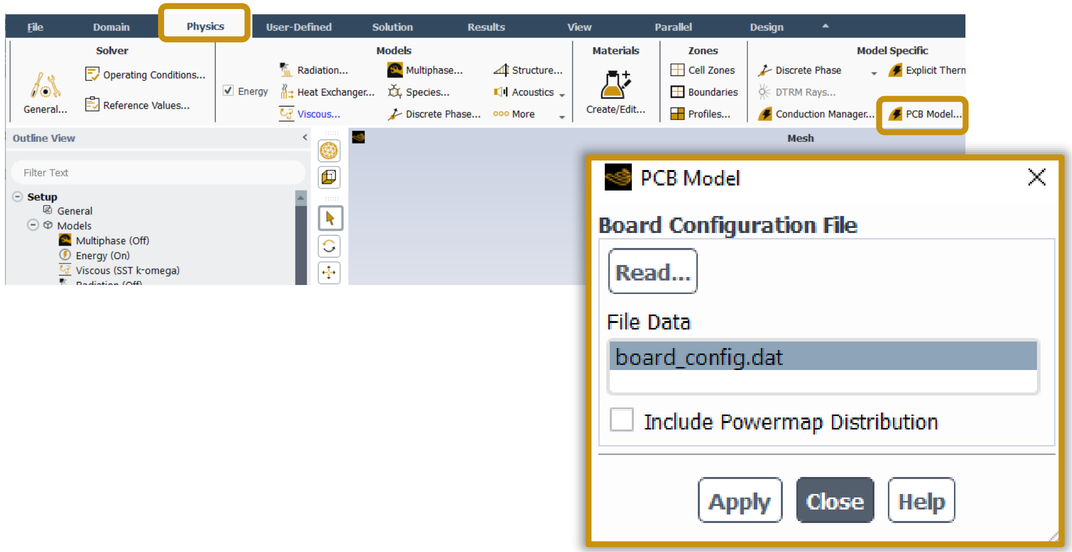
- Fluent already has an efficient workflow to simulate individual modules
- The Battery Pack Builder Tool streamlines the workflow to assemble a battery pack by using a cold plate and copy/paste an existing battery module setup
- Allows analysis of module-to-module variation



IcePak → Fluent Workflow for Printed Circuit Boards

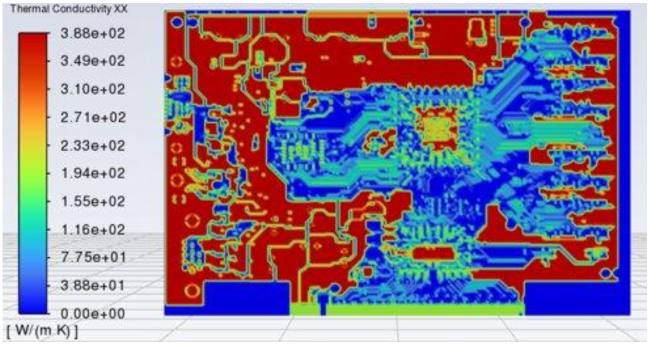
Include an IcePak PCB analysis in a Fluent case, for example LED headlamps and automotive sensor assemblies, where the complex geometry and/or physics handling of Fluent is needed

- Append a PCB-only *.cas file from IcePak into a Fluent case with other geometry / physics
- Use Fluent's Mesh interfaces to connect PCB to rest of the domain
- Use the IcePak *board_config* file* to apply an accurate PCB thermal conductivity profile in Fluent

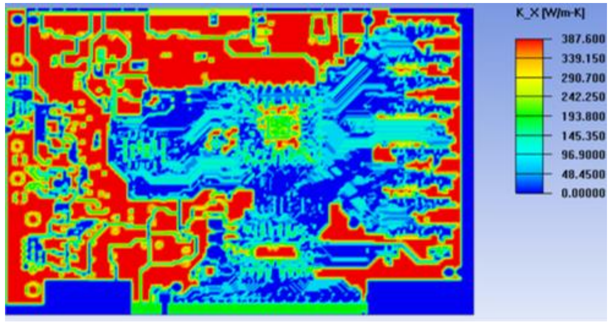


Thermal Conductivity

Fluent



IcePak

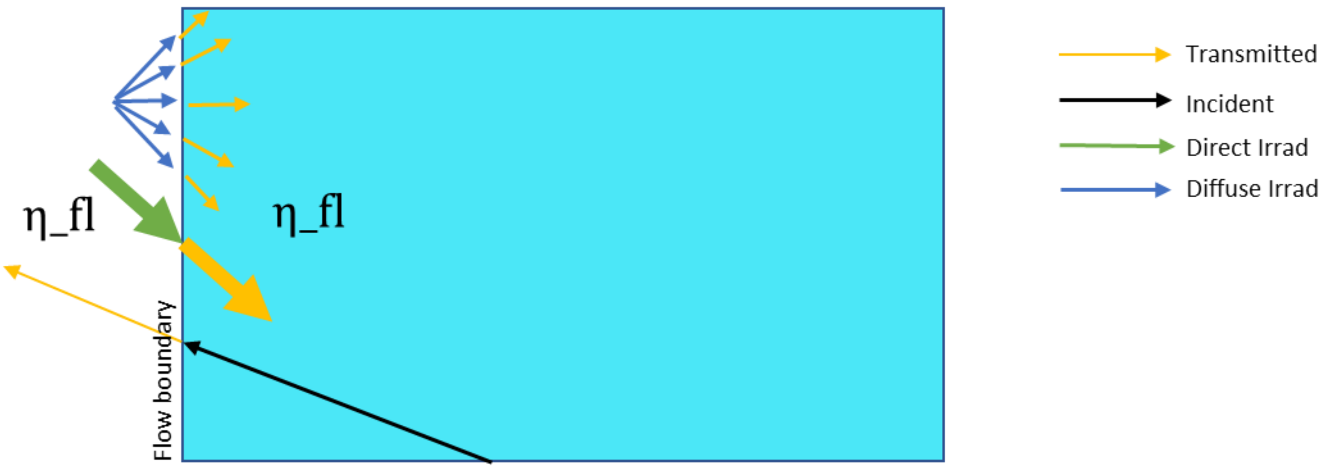


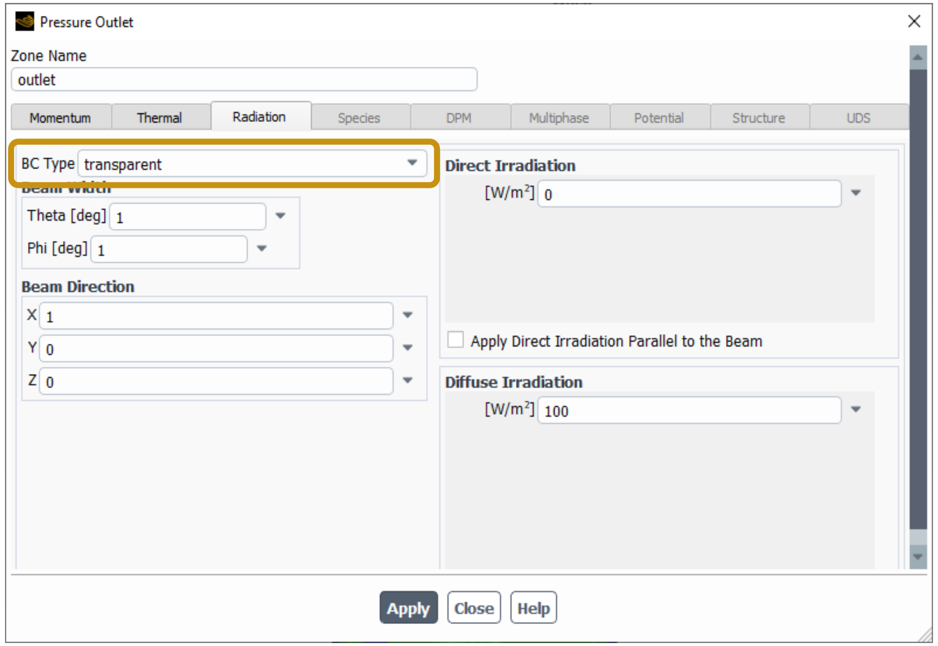
*It may be necessary to edit the thread_id in the board_config file to match the correct zone in the full Fluent simulation

Transparent Inlet/Outlet Boundaries for Radiation

For cases where an external flow boundary exists through which energy enters/exits via radiation

- Support for transparent inlet/outlet boundaries with Monte Carlo and Discrete Ordinates radiation models
  - Previously all flow boundaries were treated as opaque

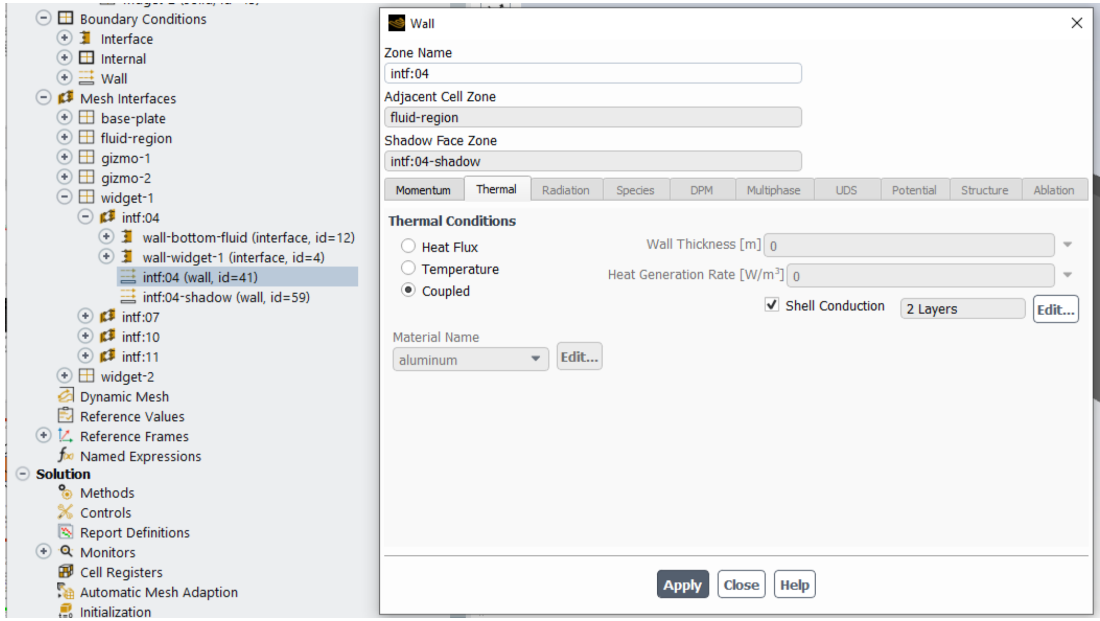




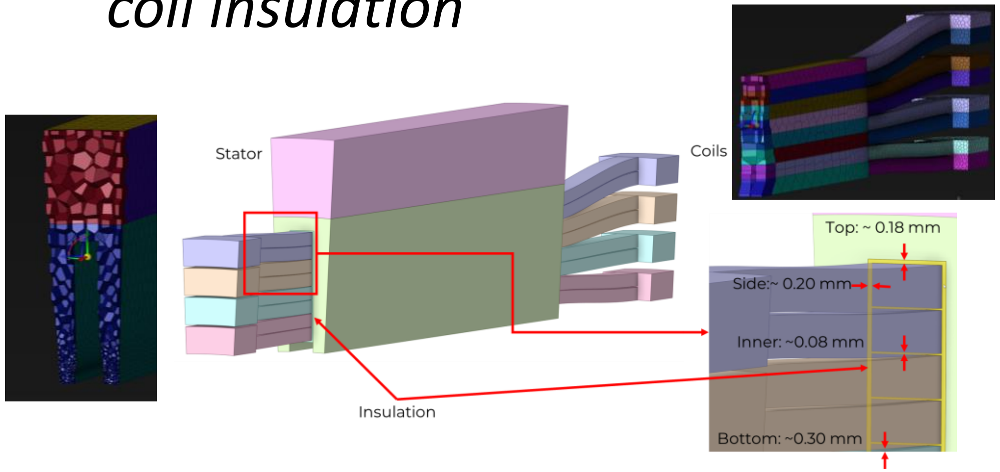
Shell Conduction With Non-Conformal Interfaces

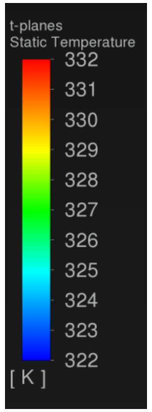
Greater meshing flexibility for cases that can benefit from shell conduction for insulation layers, heat-shields, etc

- Multi-layer shell conduction can be enabled on mesh interface children
- Otherwise same workflow as conformal meshes

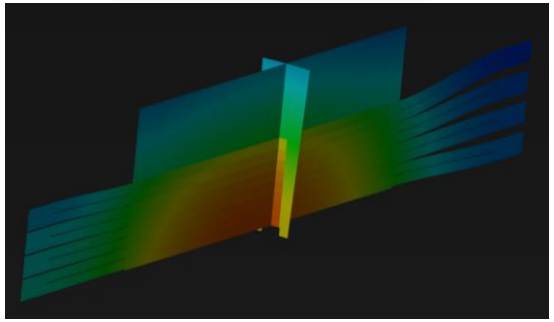


Example: 5° sector of electric motor stator with coil insulation

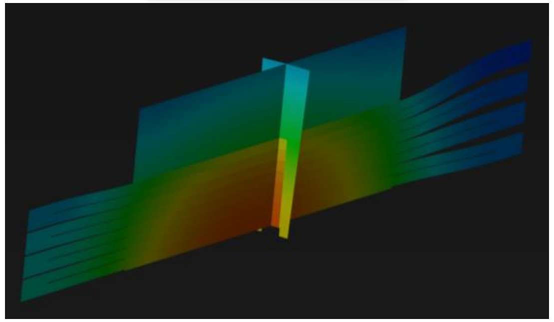




Shell Non-Conformal



Shell Conformal

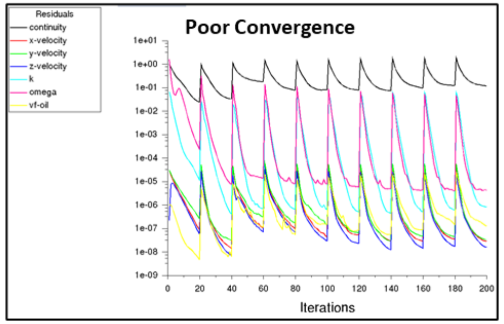


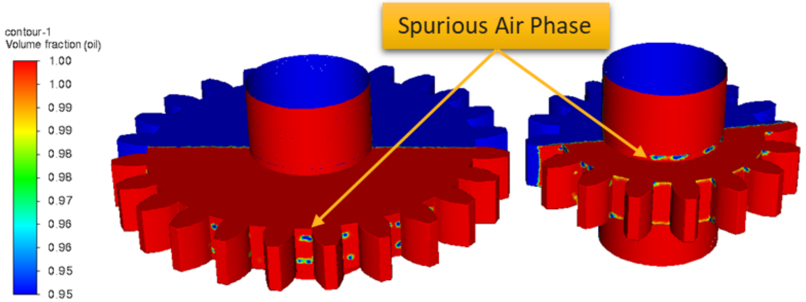
Volume of Fluid (Implicit Formulation)

Elimination of Spurious 2nd Phase Generation with Implicit Formulation

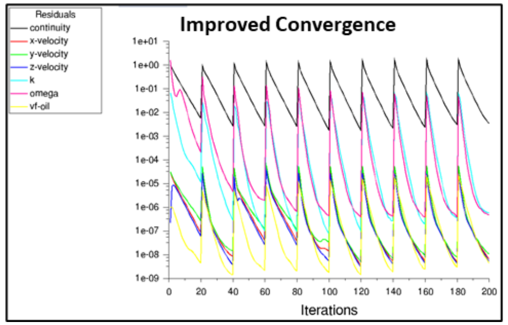
- Addresses unphysical results and associated poor convergence
- Improvements observed for water pumps, high-speed gearboxes, hydraulic components, sharp-edged-orifices

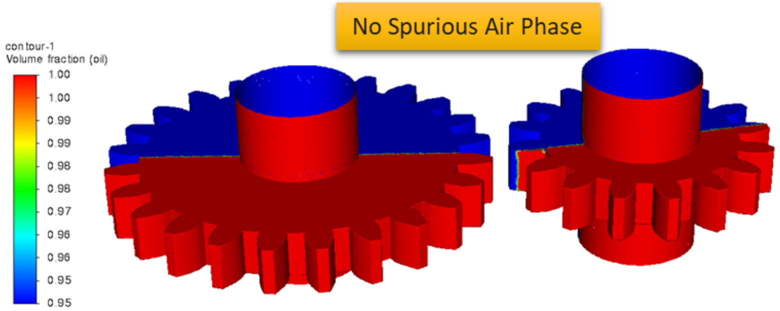
Old formulation





New formulation



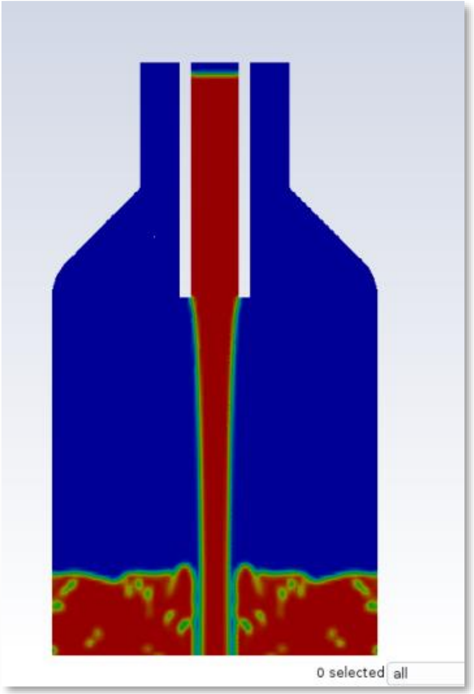


Volume of Fluid (Explicit Formulation)

Better convergence by solving volume fraction at end of time-step

- Existing treatment of Explicit VOF solves volume fraction at the beginning of time-step
- This treatment has certain limitations which do not have any workaround
  - CFL restrictions during first time-step
  - Restart issues especially for compressible flow
  - Sensitivity on change in flow boundary conditions during run-time
  - Sensitivity on mass transfer cases
- New option to solve VOF at the end of time-step addresses many of the above limitations

Vial Fill: inlet boundary condition change from velocity to pressure during run time



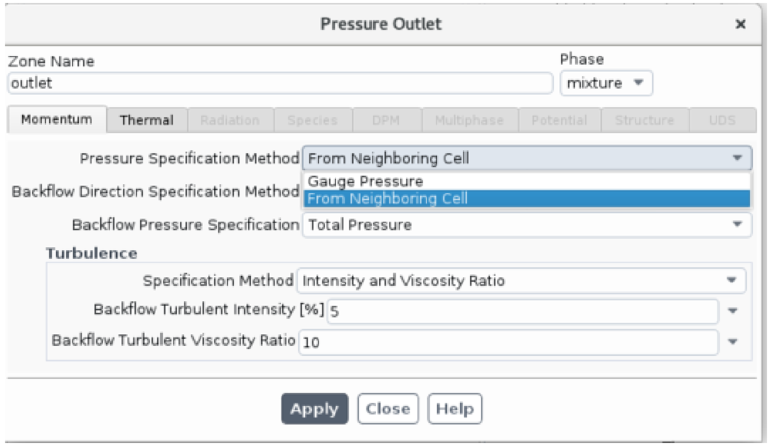
Diverges when VOF is solved at the start of time-step. Converges when solving at end of time-step

```
/solve/set/multiphase-numerics/advanced-stability-controls/equation-order> solve-exp-vof-at-end?
Solve Explicit VOF at the end of time-step? [no] yes
```

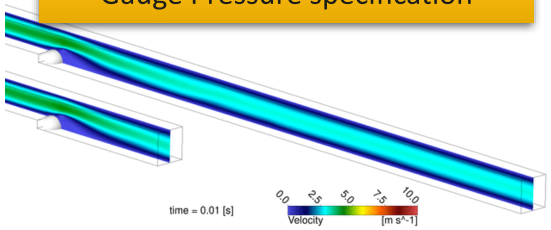
Pressure Outlet Boundary Improvements for VOF

Addresses unphysical results that can arise in certain surface-tension-dominated and rotational flows

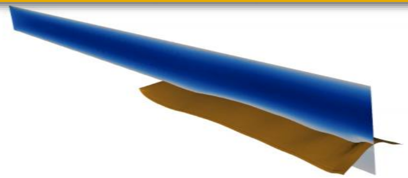
- New **From Neighboring Cell** method for Pressure Specification
  - Important for surface tension driven flow, when drop passes through outlet.
  - Can be used when boundary condition is unknown
- Backflow Direction default method changed from **Normal to Boundary** to **From Neighboring cell**
  - Critical for obtaining correct results in rotating flows

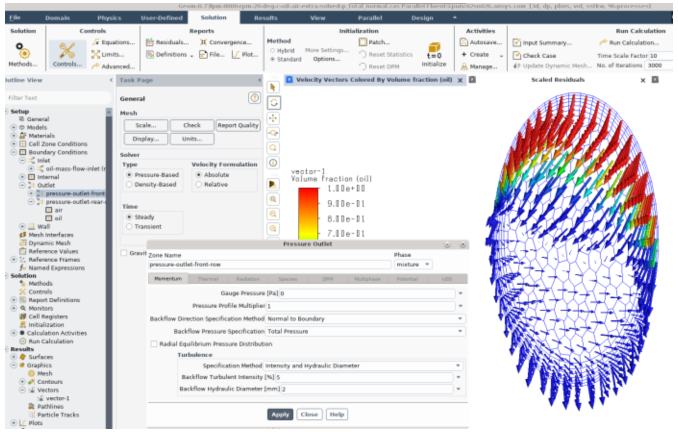


Unphysical fluctuations at exit with Gauge Pressure specification

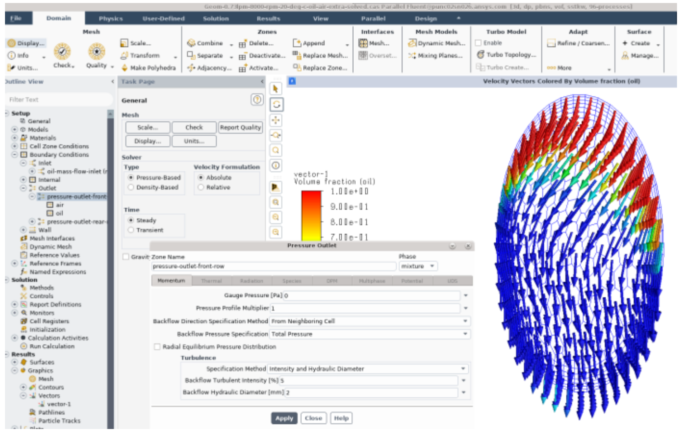


Smooth exit with From Neighboring Cell





Incorrect flow field using **Normal to Boundary** for rotating flow

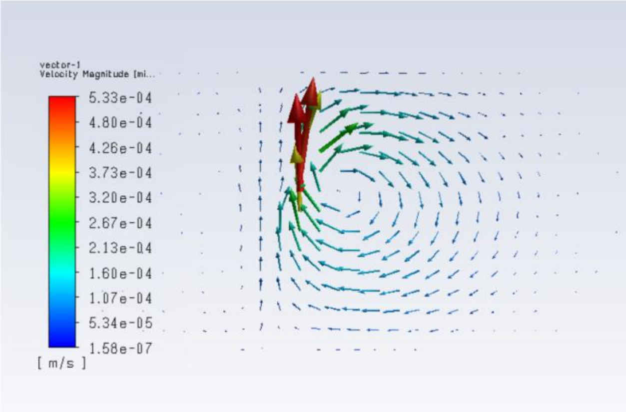


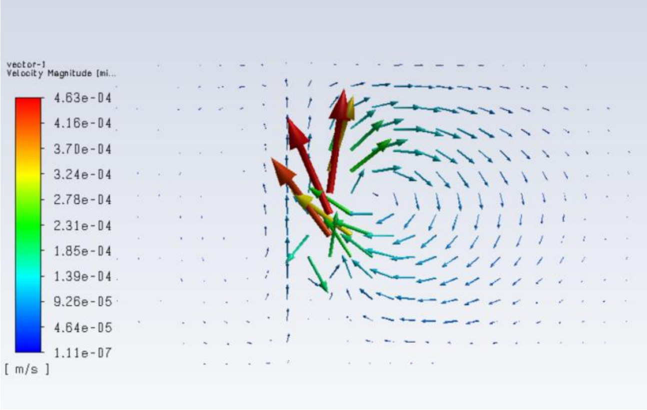
Correct flow field using **From Neighboring Cell** for rotating flow

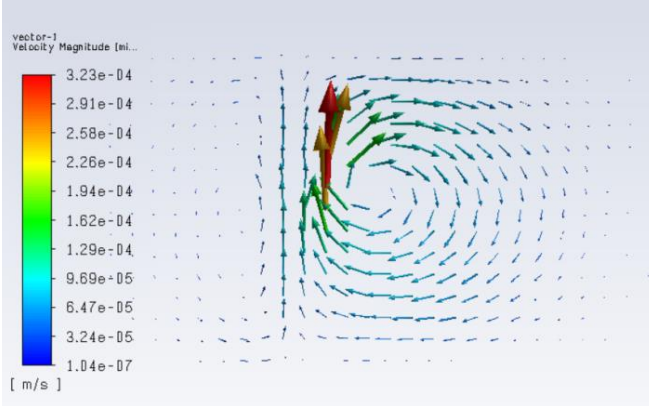
Phase-Level UDS and Species Transport Improvements

Eliminated parallel-dependency of zero diffusion boundary condition at the multiphase interface for UDS and species transport models

Before
Different results with different partitioning

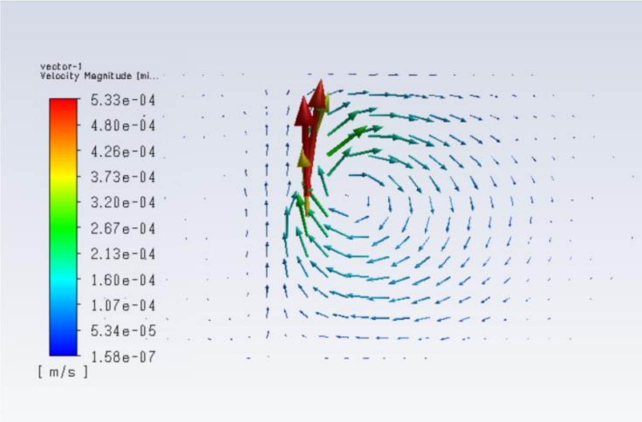


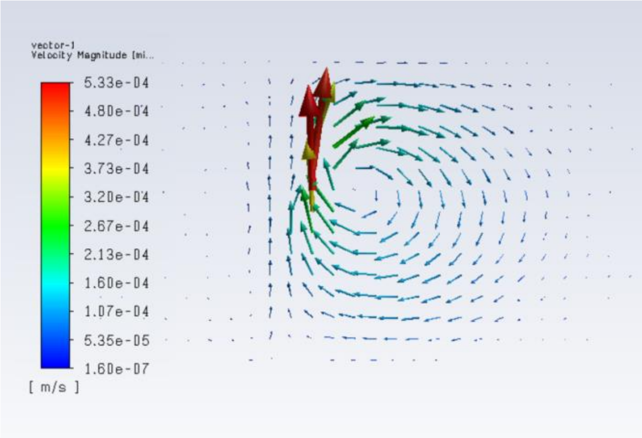


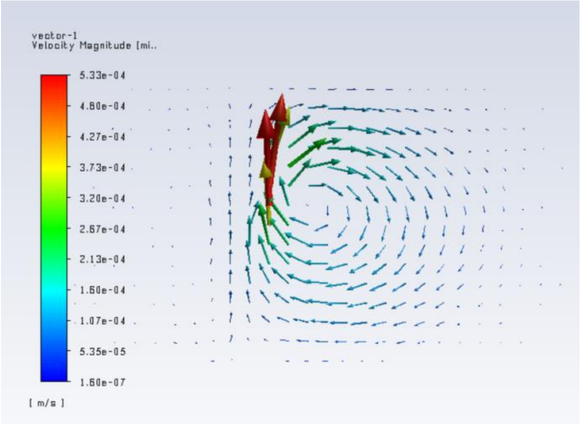


t1 **t2** **t7**

2022 R1
Results independent of partitioning





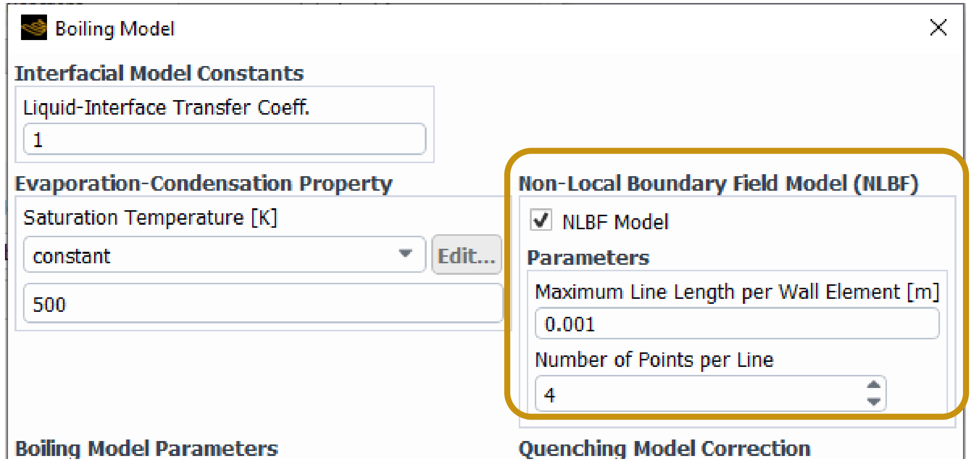


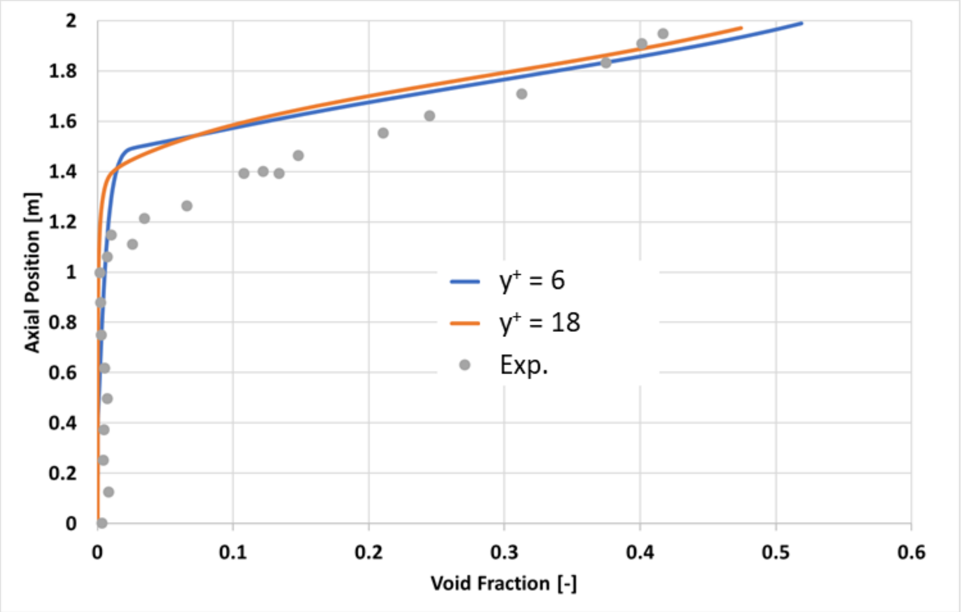
Ansys

Mesh-Insensitive RPI Boiling

*New **Non-Local Boundary Field** method removes apparent conflict of low-y^+ for most accurate heat transfer, and high-y^+ for RPI boiling*

- Reduces RPI model mesh-sensitivity below $y^+ < 30$
  - Calculating sub-cool from wall-adjacent temperature with low y^+ meshes can lead to unphysical results or divergence
  - **Non-Local Boundary Field** approach treats wall boiling as a non-local phenomena. Sub-cool temperature and departure diameter are computed along a series of lines extending into the bulk region with averaging of temperatures at sample points
- User activates model in Boiling Model panel and chooses maximum line length and number of points per line



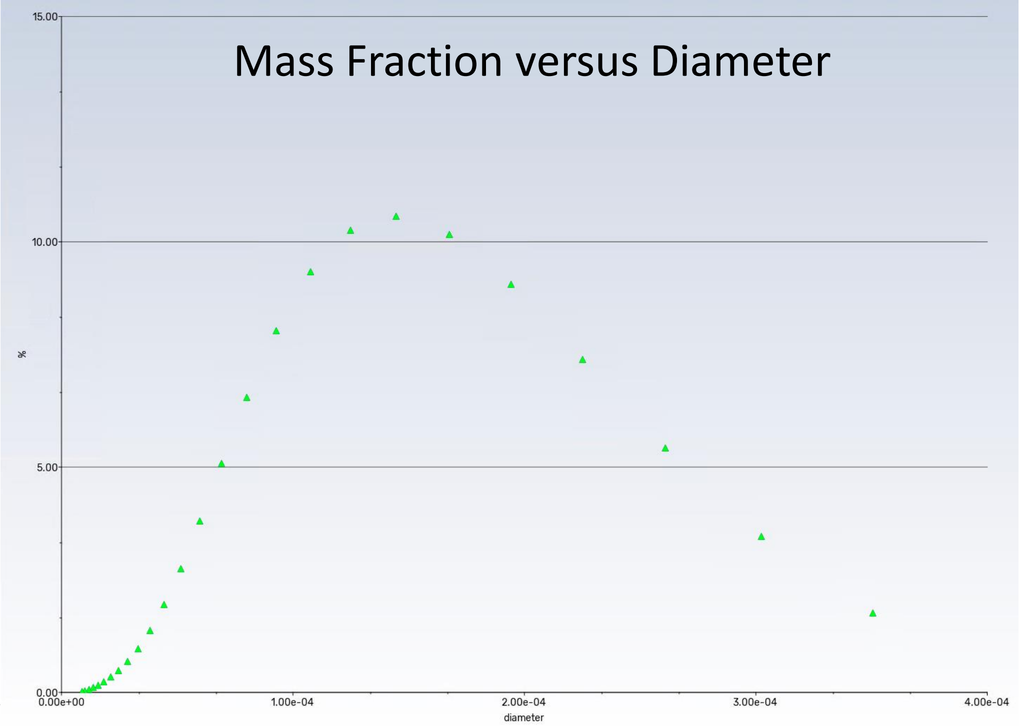


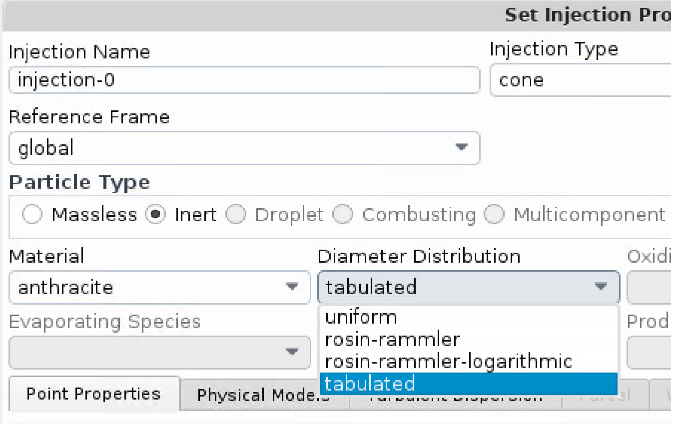
2D Axisymmetric Pipe with Heated Wall, Bartolomej et al

Tabulated Particle Size Distribution

Import tabulated data for injection size distributions

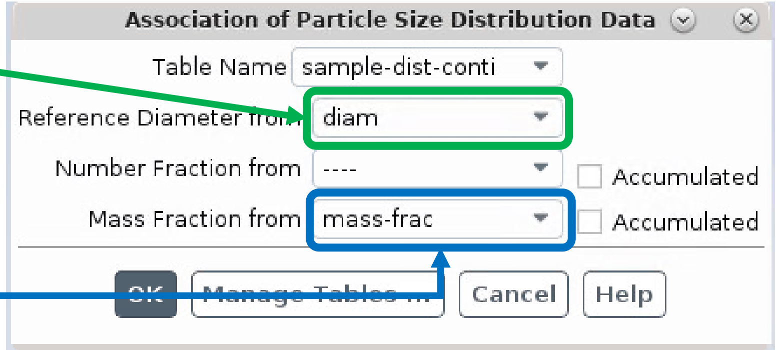
- For surface and cone injections
- Specify mass and / or number frequency distribution of measured particle sizes
  - Either as frequency or cumulative distributions





sample-dist-conti

num-frac	mass-frac	Diam
0.029871404	8.39949E-05	8.12819E-06
0.074395037	0.000331542	9.47675E-06
0.09261217	0.000654137	1.10491E-05
0.096133363	0.001076148	1.28823E-05
0.092064168	0.001633392	1.50197E-05
0.084628946	0.002379651	1.75116E-05
0.076196363	0.003395719	2.04171E-05
0.06802667	0.0048048	2.38046E-05



DPM Usability Improvements

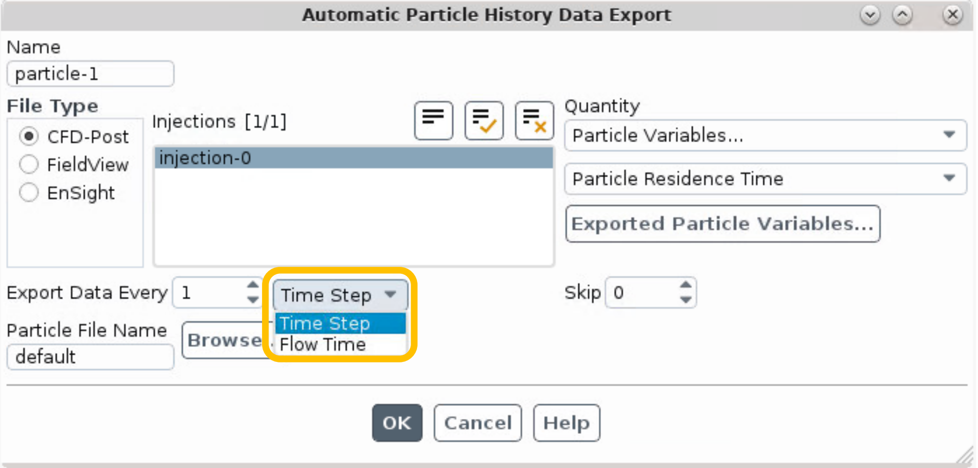
Faster preview of injections
Improvements to particle data export

- Improved injector display performance for all DPM injection types
  - Display of vectors representing DPM injections was improved by more than one order of magnitude
  - Example: display of 100 injections (Linux, no graphics acceleration):

2021 R2	2022 R1	Improvement
44 s	3 s	15x

- For transient simulations allow export of particle tracks based on flow time interval in addition to iteration frequency
  - File → Export → During Calculation → Particle History Data …

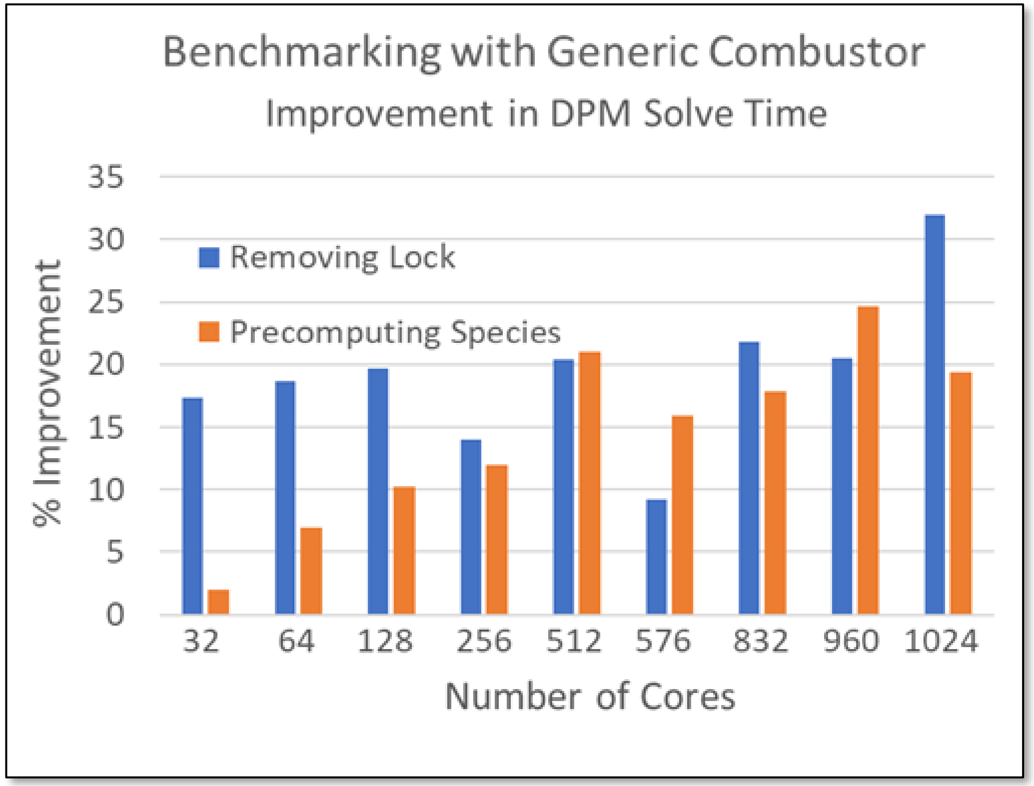
- Improved export of 2d particle data to Ensight
  - Transformation of axisymmetric particle data into xy-plane (as is done for CFD-Post)



Improved Particle Tracking

Significant reduction in DPM solve time
Improved robustness and accuracy in various scenarios

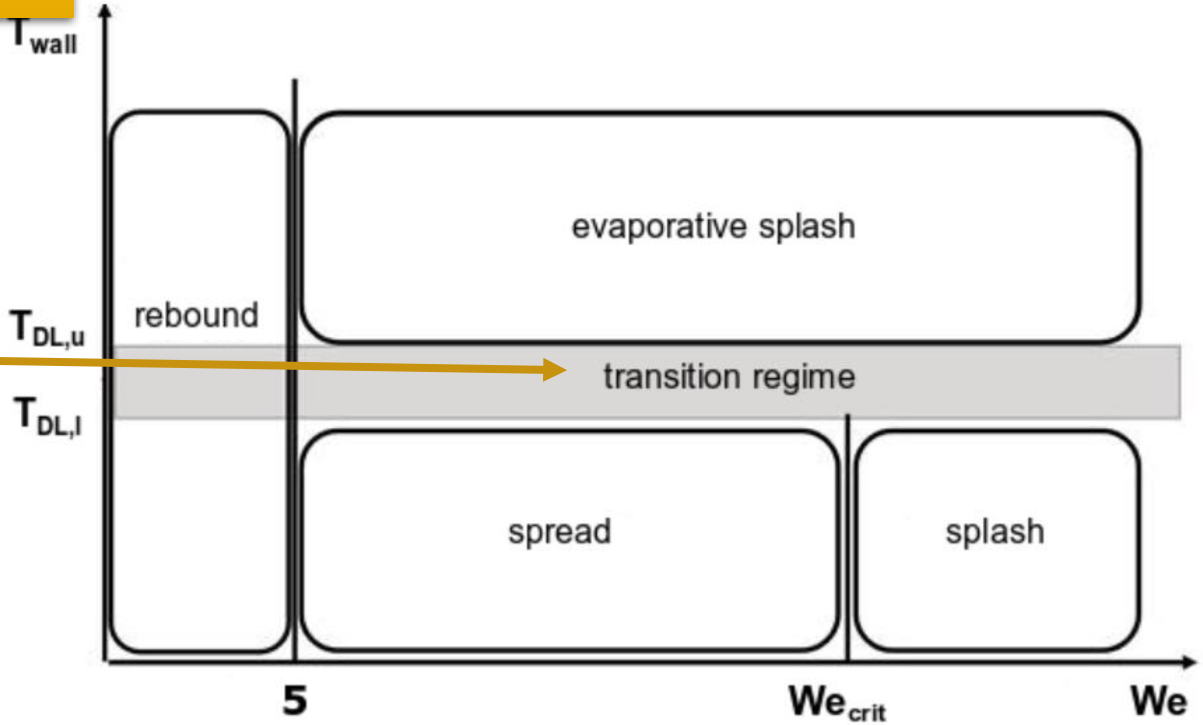
- Performance improvements
  - Thread lock removal improves core hybrid tracking performance
  - Precompute species option
    - Improves performance for combustion cases with all available tracking approaches and parallel modes
  - Further performance options for GTC simulations
    - Barycentric intersections
    - Timestep adaption of Darmofal and Haimes
- Accuracy and Robustness improvements
  - Improved variable interpolation for
    - Multiphase models
    - Steep gradients within a cell
    - Lagrangian wall film particles



Stochastic Kuhnke Model

Variant of Kuhnke model that considers critical temperature transition in Selective Catalytic Reduction (SCR) systems

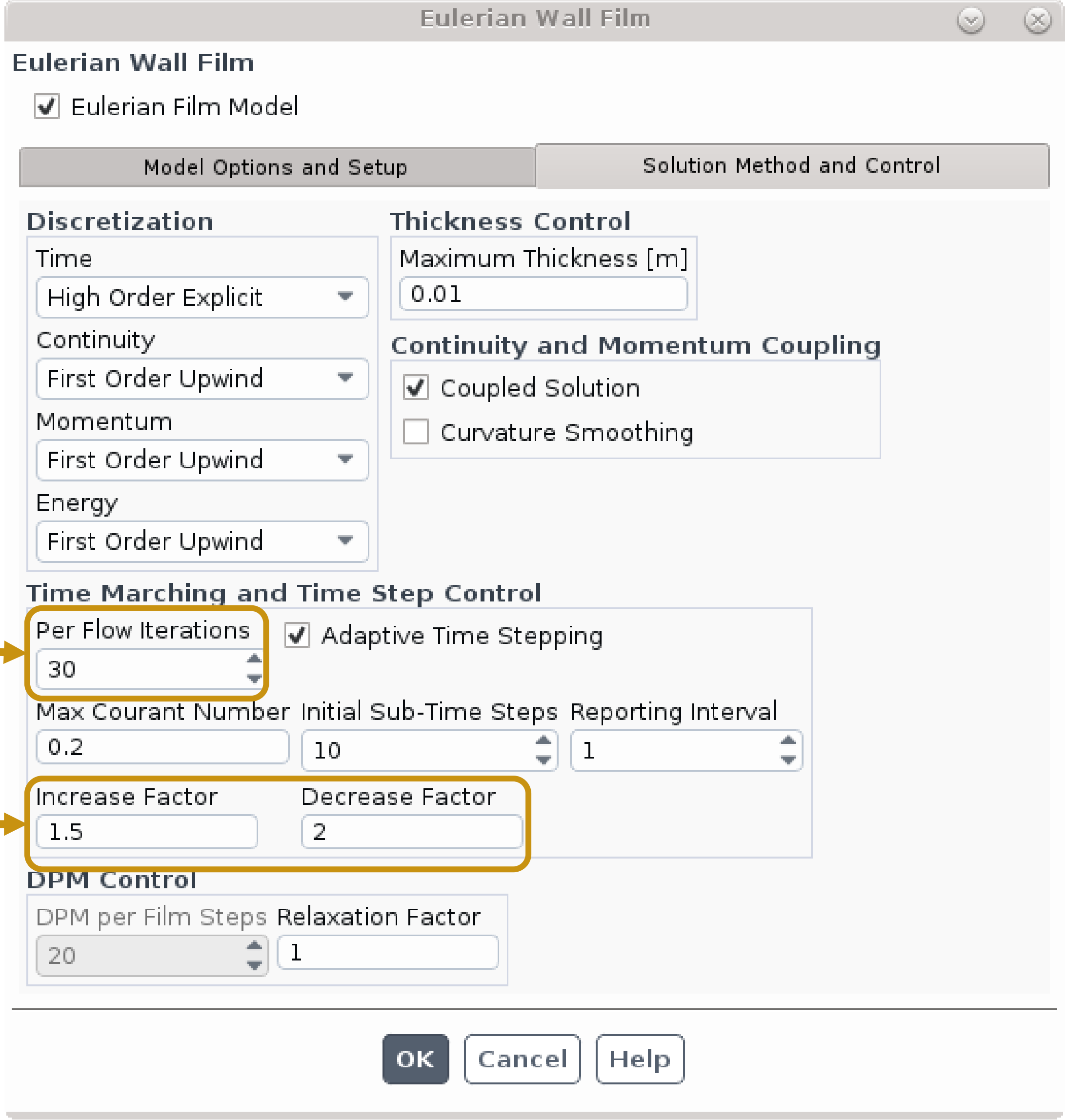
- Derived from Kuhnke model with the following extensions
  - Different regime transition criteria
  - Includes stochastic effects into the critical temperature transition process
  - Introduces evaporative splash regime with the “partial evaporation” concept
    - Fraction of the impinging droplets is assumed to completely vaporize under certain conditions
  - Model is **compatible** with both **Lagrangian** and **Eulerian Wall Film** models



Eulerian Wall Film Solver Advancement Options

Additional EWF advancement options for transient cases to improve robustness/performance

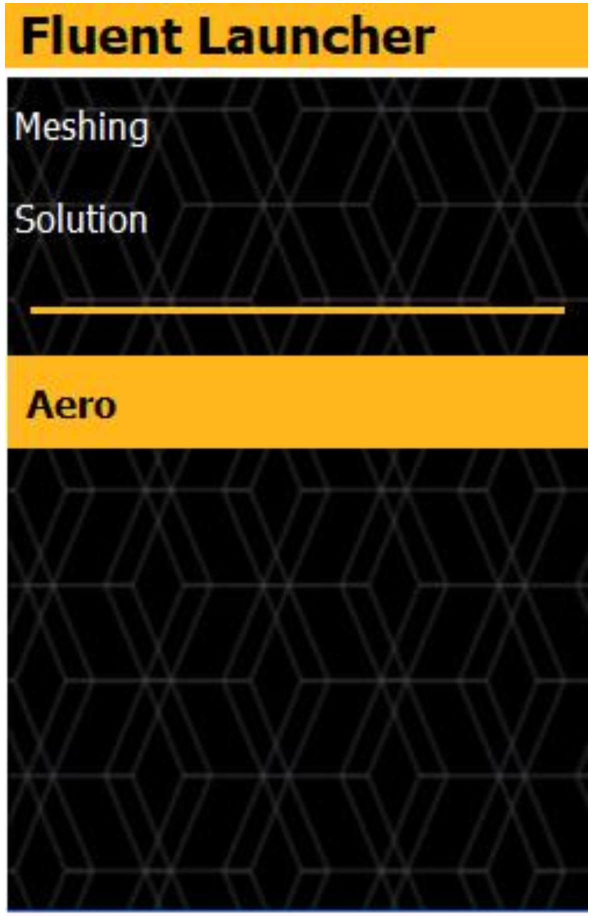
- Multiple film loops per fluid flow time step
  - Update film equations at an iteration-interval within the fluid time step to accommodate flow variable changes
  - Control via "Per Flow Iterations"
- Adaptive time stepping
  - Now available for transient flow simulations
  - User-specified **Increase/Decrease Factors** provide better control over solution process (steady and transient)

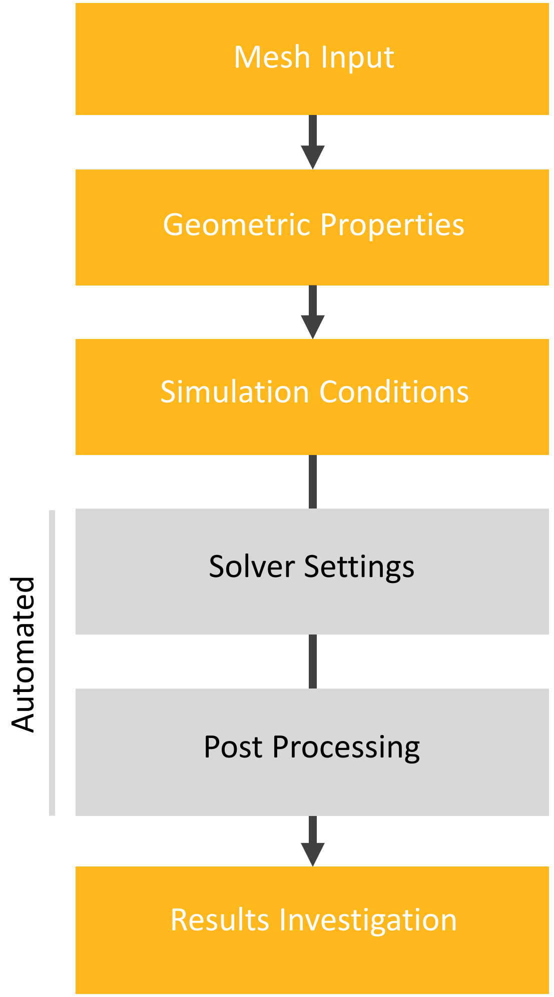


Fluent Aero Workspace

Dedicated Fluent Workspace for Aerospace External Aerodynamics Simulations

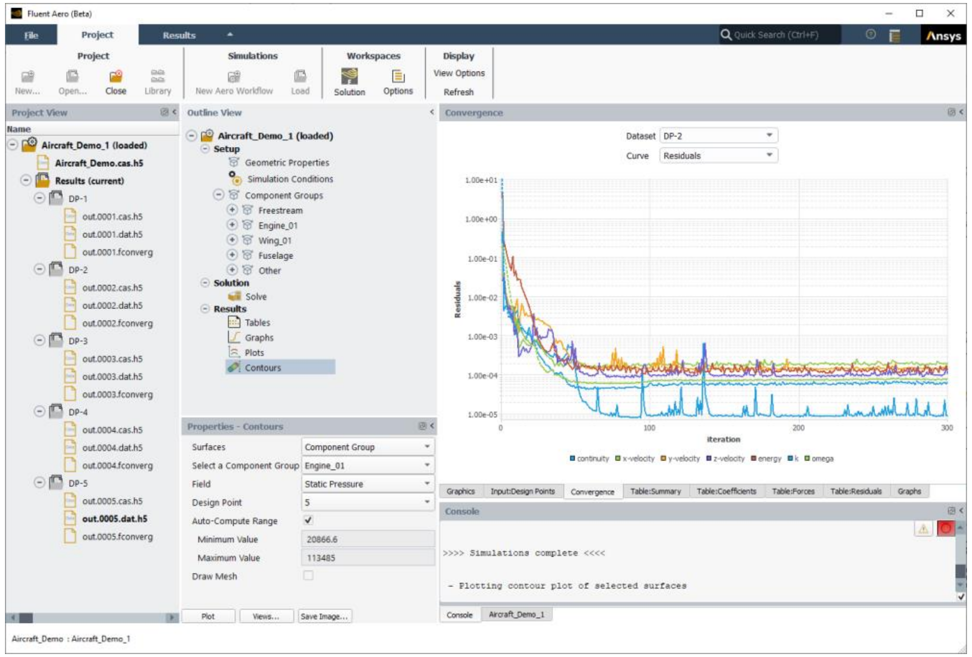
- Streamlined setup and analysis tool for aerospace CFD
- Workspace has best practices embedded to minimize need for user to adjust models and settings
  - Full access to Fluent features/capabilities when needed
- Automates the setup of solver settings, post processing output parameters, tables and charts

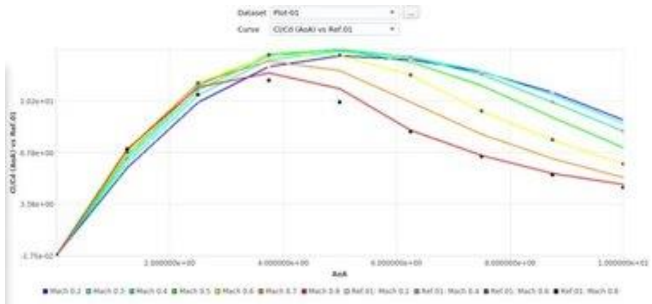




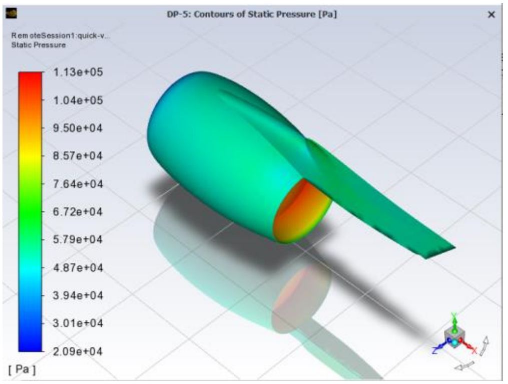
Fluent Aero Workspace cont.

- Aircraft Component Groups: input conditions and results
  - Aid simulation setup and automate specific component post processing
  - Focus user on aircraft components rather than CFD boundary conditions
  - Freestream or Wind Tunnel type domains
- Python scripting/journal recording
- Specialized post-processing

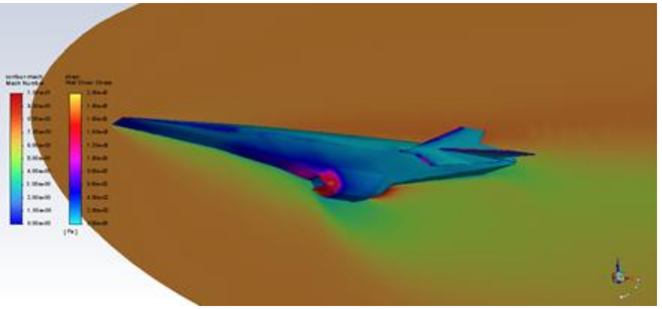




Comparison to Experimental Data: Parametric Plots



Component specific setup and post processing



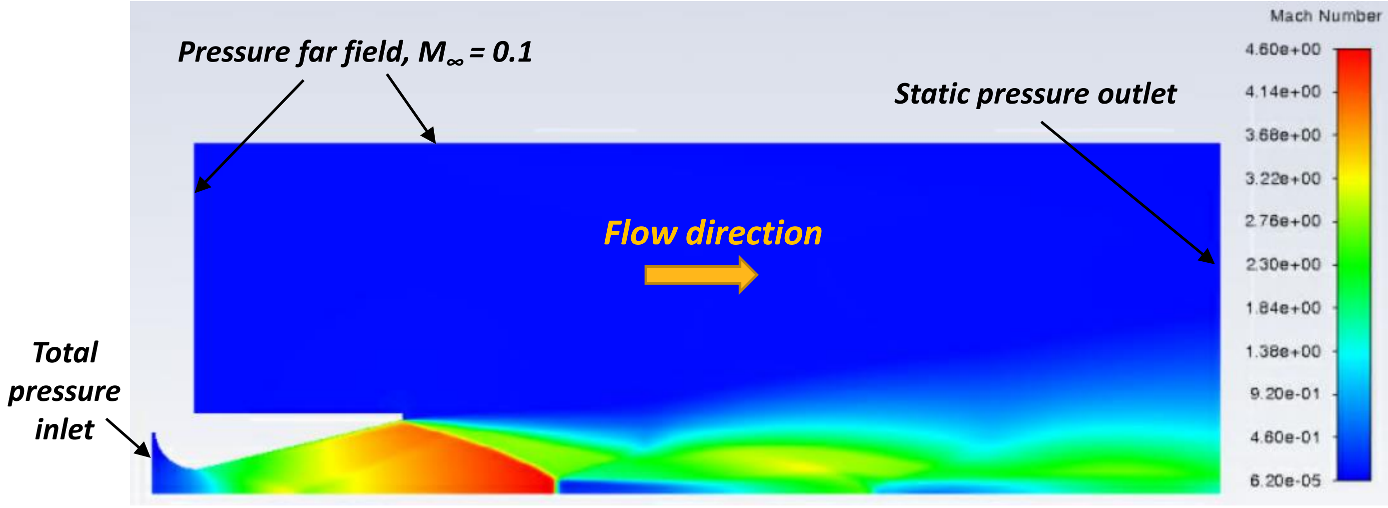
DBNS and Hypersonics support

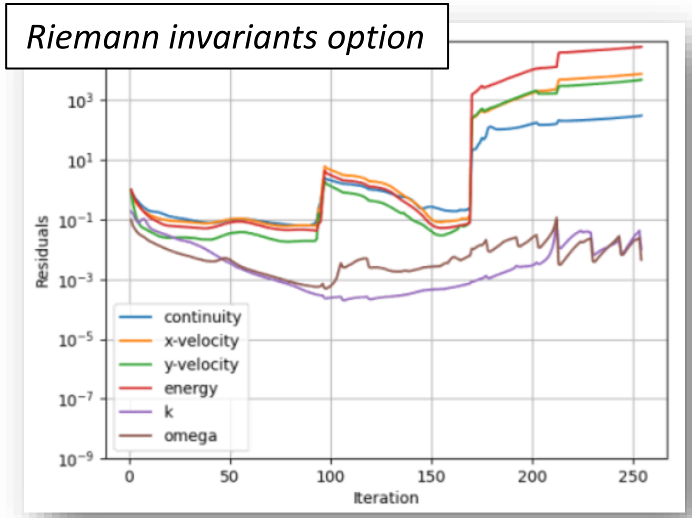
Numerics Enhancements for Density Based Solver

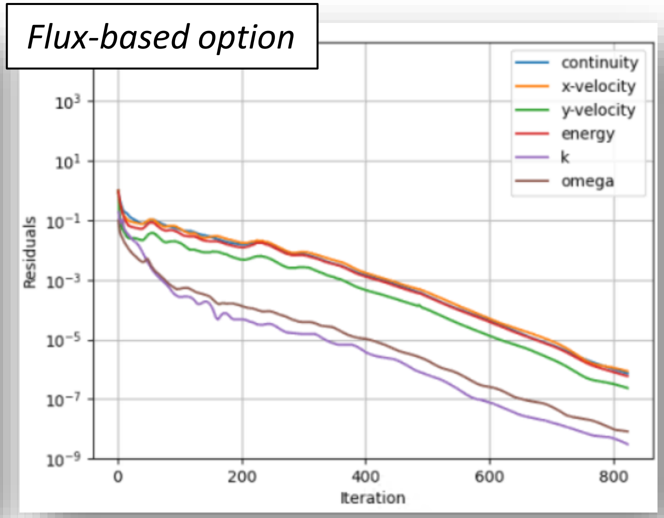
New pressure far field boundary condition option improves robustness

- New flux-based pressure far-field boundary
  - An alternative to current pressure far-field formulation
  - Uses numerical-flux instead of Riemann invariants
  - Activated from TUI:

```
> define boundary-condition bc-settings pressure-far-field type
Riemann-invariants = 0 Flux-based = 1
Type [0] 1
```







Numerics Enhancements for Density Based Solver

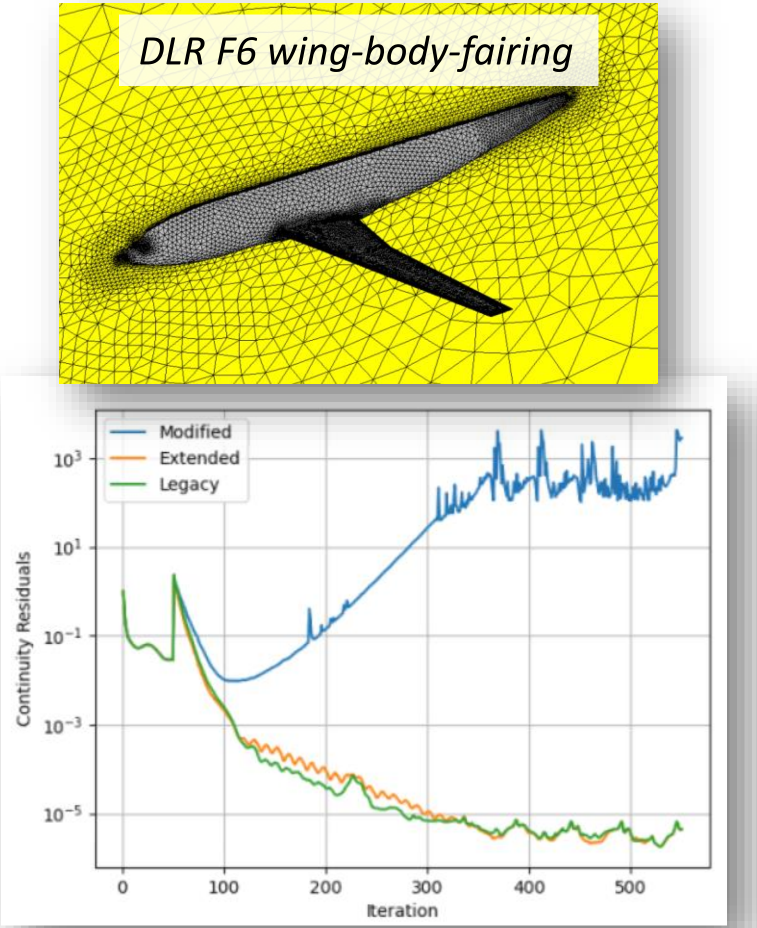
Node-based gradient improvements provide robust convergence on tetrahedral meshes

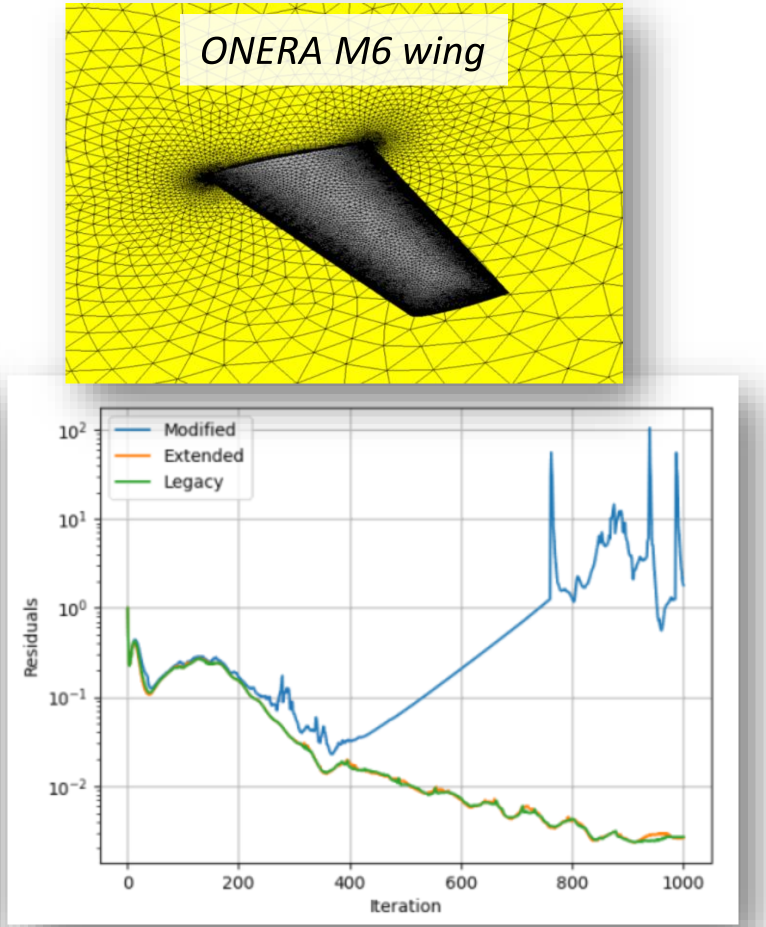
- Node-Based Gradient enhancements for DBNS solver
  - Addresses convergence difficulties on tetrahedral meshes when node-based gradients are used
  - The enhanced feature termed "Extended" is accessed via TUI under solve/set/nb-gradient-boundary-option?

```
> solve/set/nb-gradient-boundary-option?
 Options:
 Modified/Extended boundary treatment
 Legacy boundary treatment

 use modified/extended boundary treatment? [yes] yes
 use extended boundary treatment> [no] yes

 extended boundary treatment will be used
```



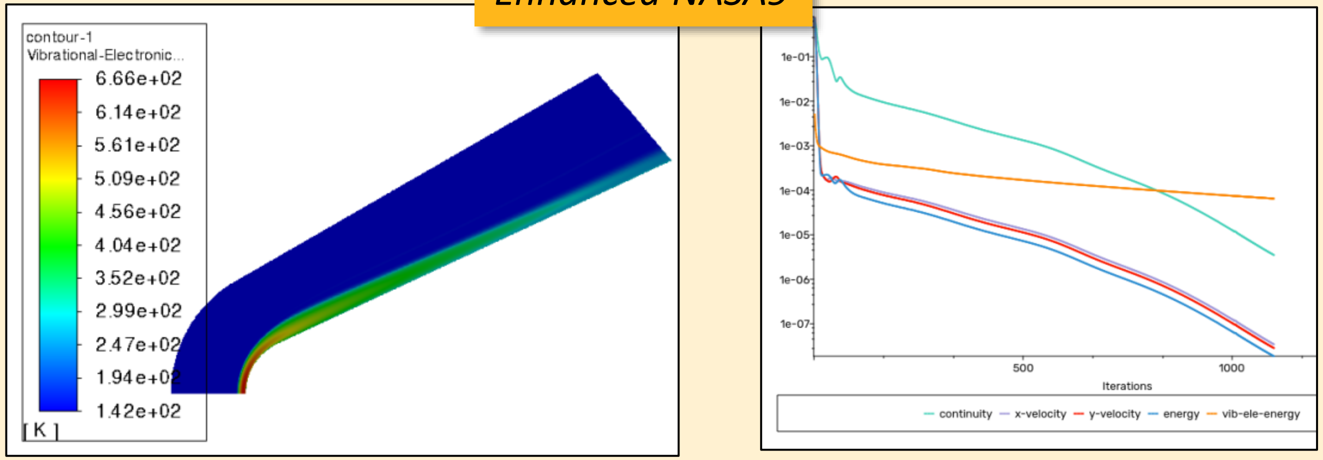


Two Temperature Equation Model Enhancements

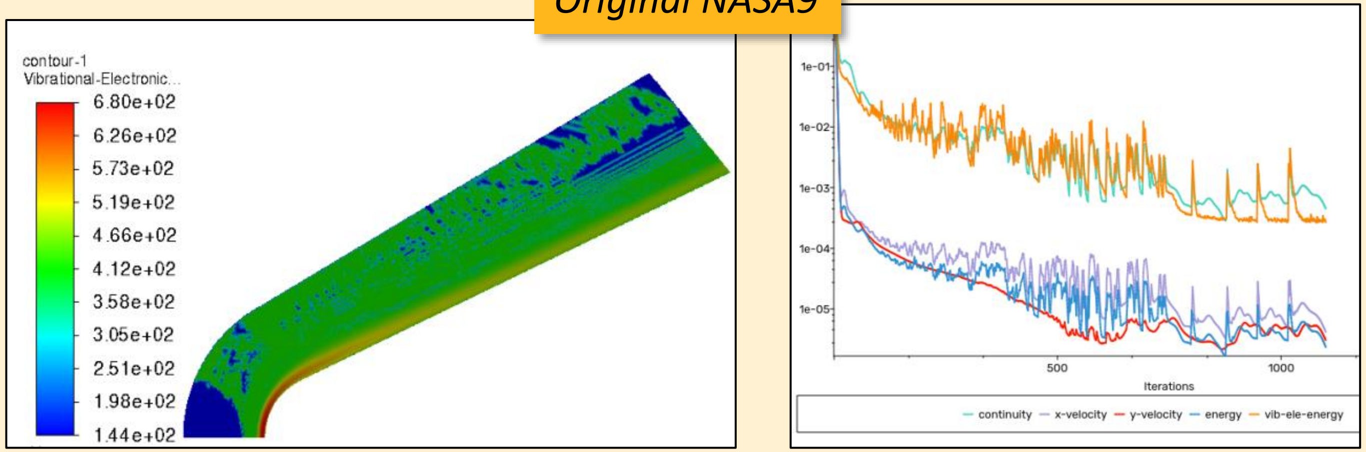
Robustness improvements

- Improved Two-Temperature equation model robustness with enhancements to the NASA 9-coeff for Cp (Specific Heat)
  - Smooths out anomalies in the polynomial curves for Cp
  - Improves model robustness for the entire flight range
  - Enabled by default

Enhanced NASA9

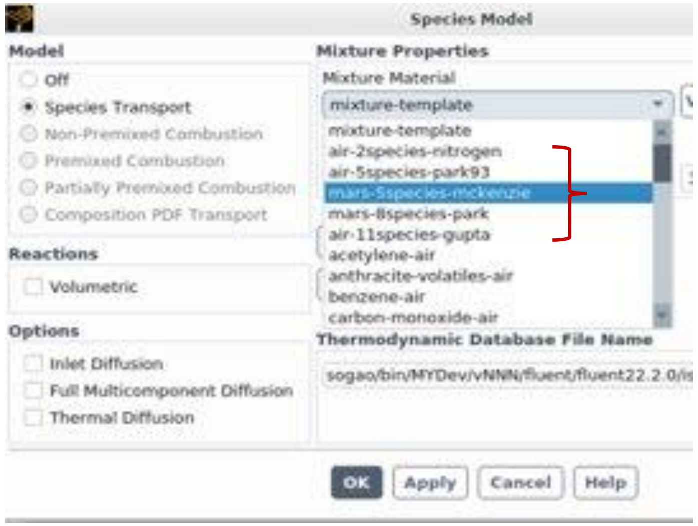


Original NASA9



Two Temperature Equation Model Enhancements

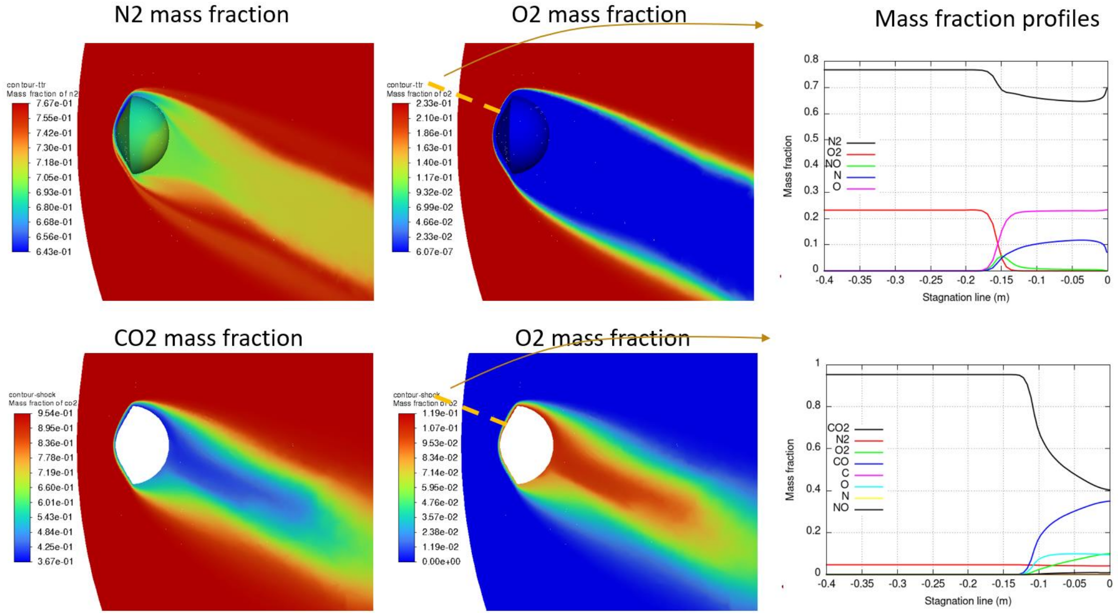
Included mechanisms suitable for Earth and Mars re-entry missions



- For Earth atmospheric re-entry two popular reaction mechanism area available for use
  - **5-species Parks93** reaction mechanism
  - **11-species Gupta** reaction mechanism
- Tri-atomic molecule to support for Martian entry missions with two reaction mechanism
  - **5-species McKenzie** reaction mechanism
  - **8-species Parks** reaction mechanism

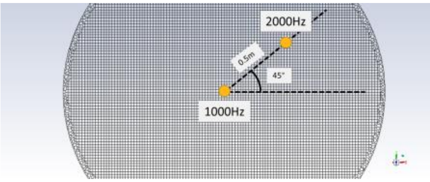
Earth

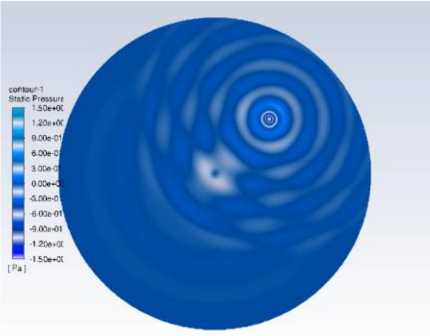
Mars



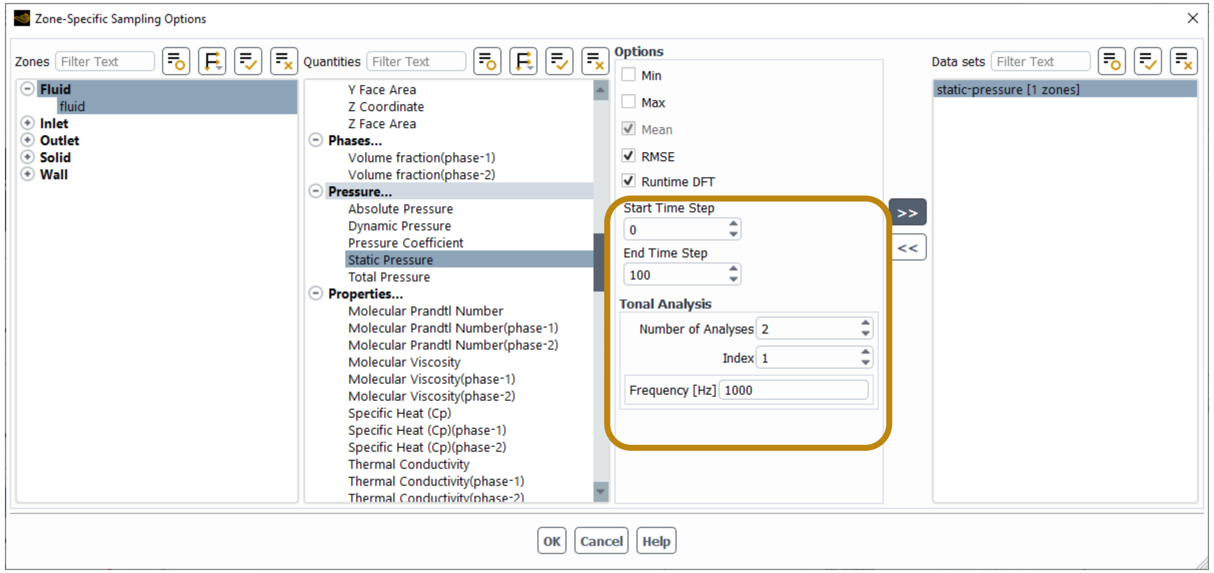
Runtime Discrete Fourier Transform

- Perform DFT Tonal Analysis during solution
  - Compute magnitude and phase information of selected frequency(-ies) during solution for later postprocessing
- Enabled in **Zone-Specific Sampling Options**
- Example: Two monopoles





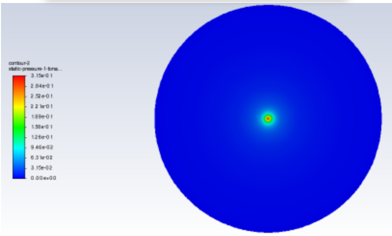
Instantaneous Pressure

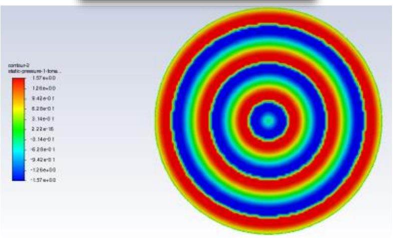


Pressure Magnitude

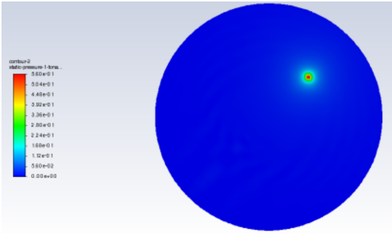
Pressure Phase

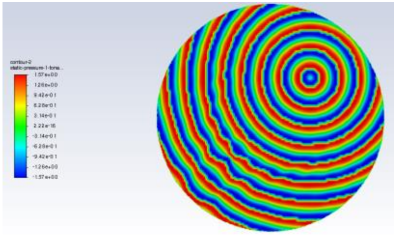
1000 Hz





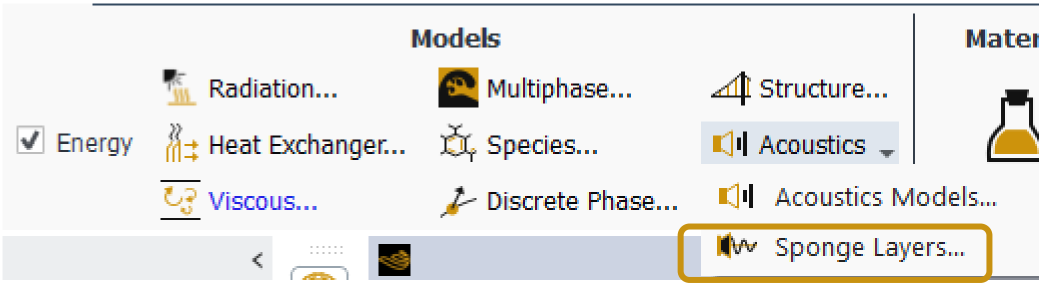
2000 Hz

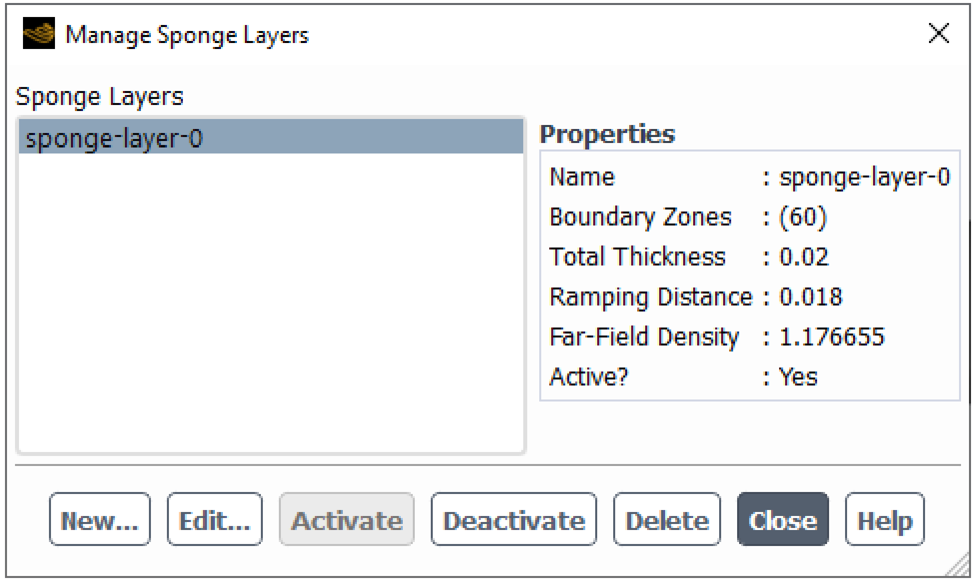




Acoustic Sponge Layer Creation

- Best practices may require a numerical “sponge layer” to avoid non-physical reflections near boundaries
- Density is blended between solver value and a specified far-field value over the **Ramping Distance**
- Multiple sponge layers can be defined, but only one can be **Active** at a time
  - A single sponge layer definition can be used for multiple boundaries

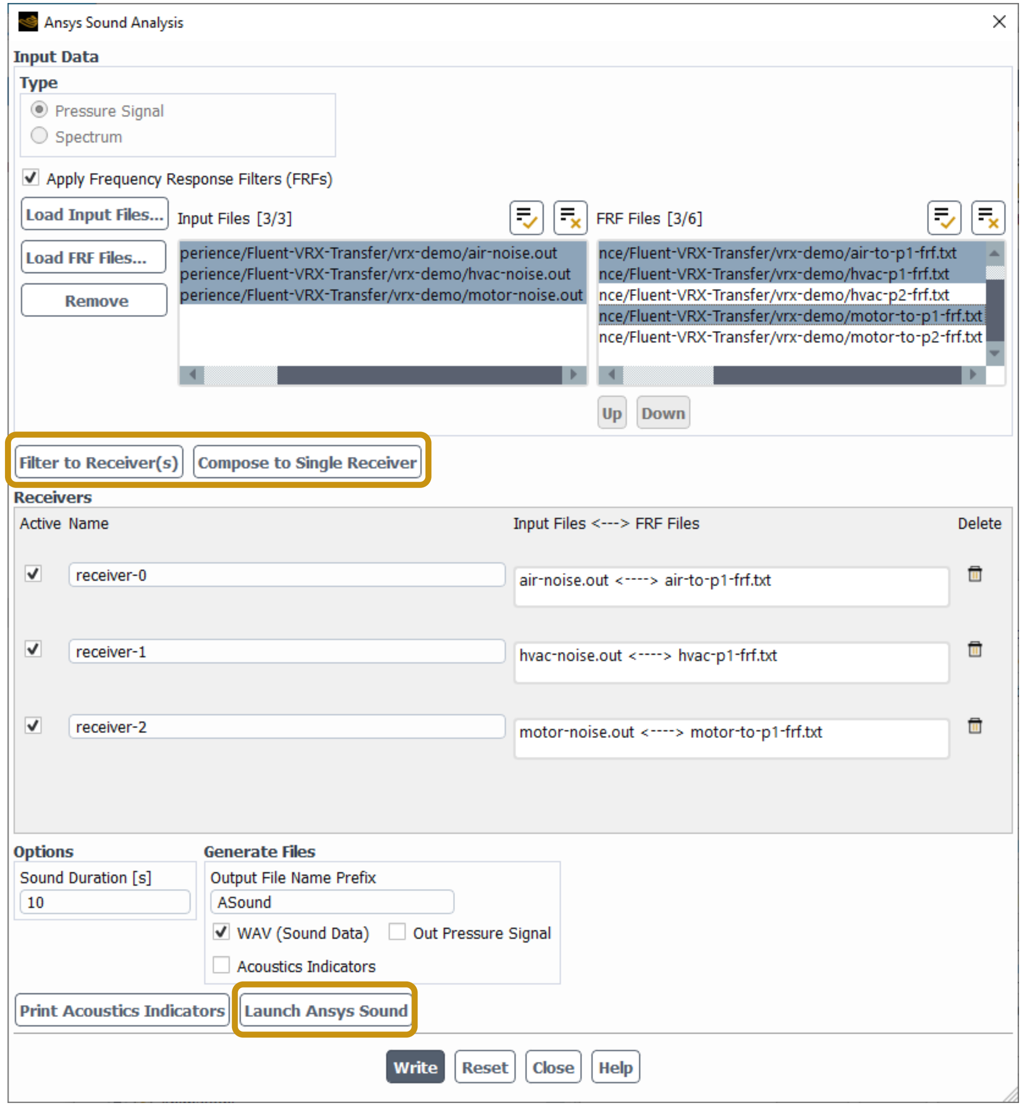




Improved Workflow with Ansys Sound

Launch Ansys Sound directly from Fluent;
Extensions to sound composition analysis

- **Launch Ansys Sound** starts Ansys Sound and transfers files.
- Multiple selection for computing sound at receivers
  - **Filter to Receiver(s)**:
    - Maps ***one*** selected input to ***N*** selected receivers, OR
    - Maps ***N*** selected inputs to ***N*** selected receivers
  - **Compose to Single Receiver**:
    - Maps ***N*** selected inputs to ***one*** selected receiver



User-Defined Structural Boundary Conditions

Two new UDF macros for specifying nodal forces or displacements on wall boundaries

- Provide ability to use UDFs for arbitrary nodal displacement / force conditions on walls in intrinsic FSI simulations

```
DEFINE_WALL_NODAL_DISP(name, f, t, v, m)
DEFINE_WALL_NODAL_FORCE(wall_nodal_force, f, t, v, m)
```

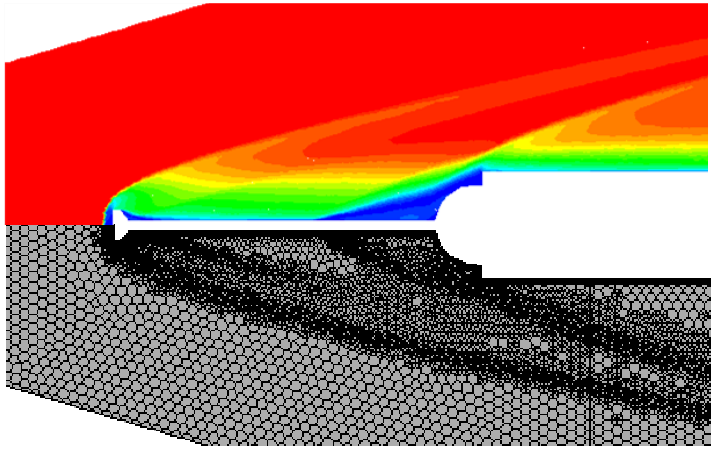
Mesh Interfaces

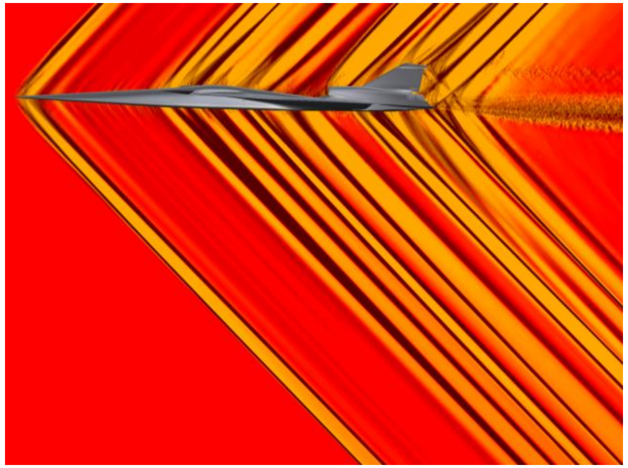
- Adjustable tolerances for auto-pairing to better control which selected zones are paired and intersected
  - Helpful for cases with gaps, thin layers, or complex geometrical features
- When creating a mesh interface, you can select any external parent zone type for pairing, not just interfaces or walls.
  - Resulting non-overlapping regions will inherit the same type as the originally-selected zone (except for interface-type which will become walls).

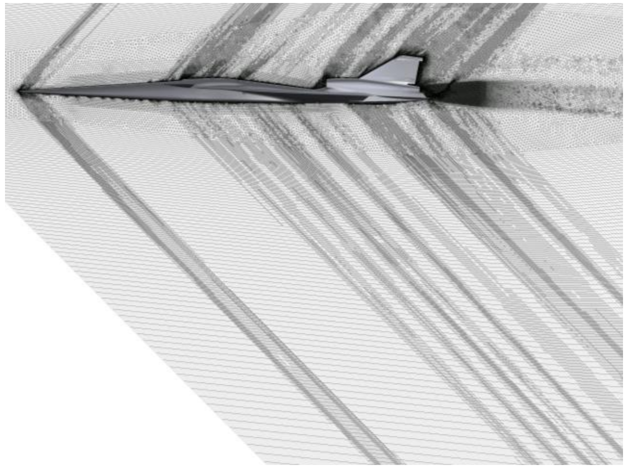
Adaption: Hessian-Based Error Metric

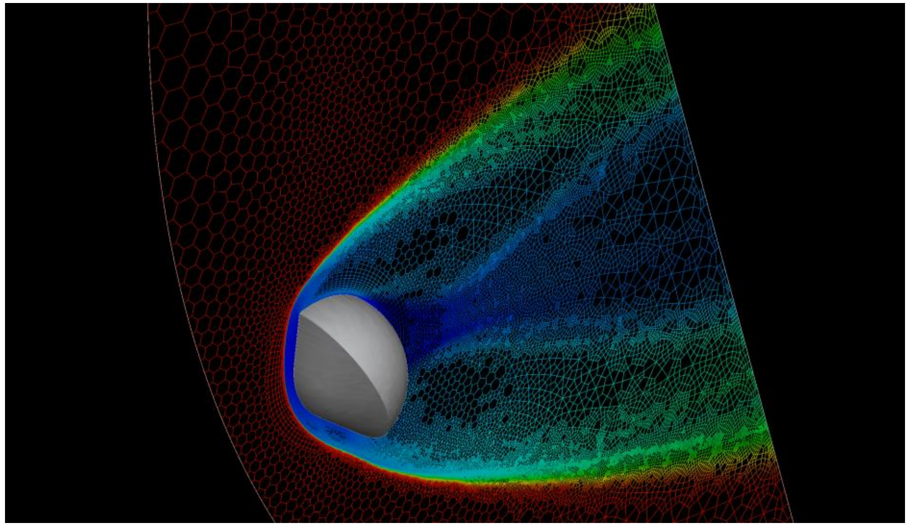
More efficient and effective mesh adaption metric for high-speed aero flows

- A new multiscale mesh adaption metric based on the solution error
- More efficient refinement selection than gradient-based methods
- Equally capture weak and strong flow features
- No heuristic adaption metric thresholds
- 10x+ savings in mesh size, computational time, disk storage, post processing effort for the same level of accuracy of a fine initial mesh









Unified Remeshing

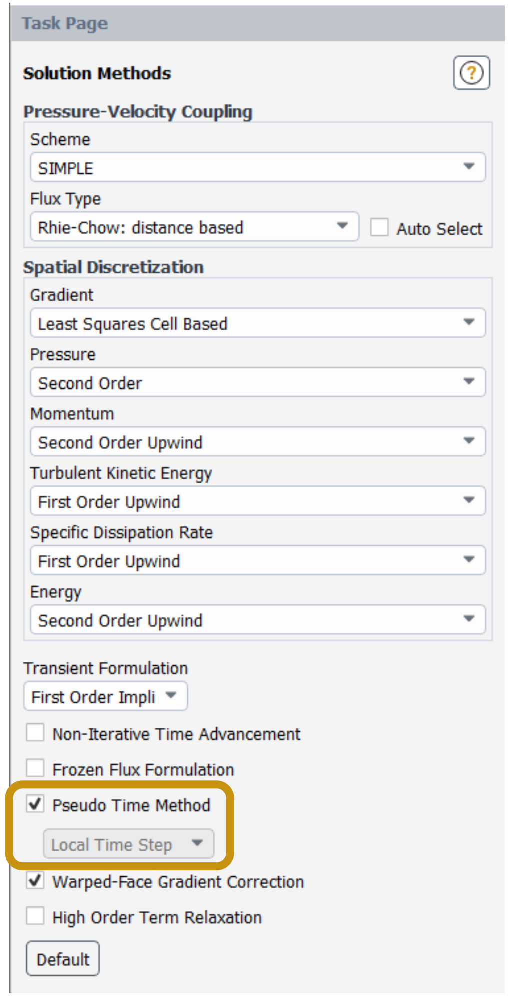
New capabilities for prism-layer remeshing and size field support

- Prism layer remeshing
  - Prism cells are marked based on quality and height providing new remeshing capabilities
    - E.g. remeshing prisms adjacent to non deforming face zone
  - More robust and simplified workflow
    - E.g. handles out of the box prism layers on adjacent zones without domain decomposition or requiring specific smoothing settings
  - Automatically detects prisms parameters
- Size field improvements
  - Added I/O functionality for size fields ensuring consistent restart
  - Added size field support for System Coupling and User-Defined dynamic zones

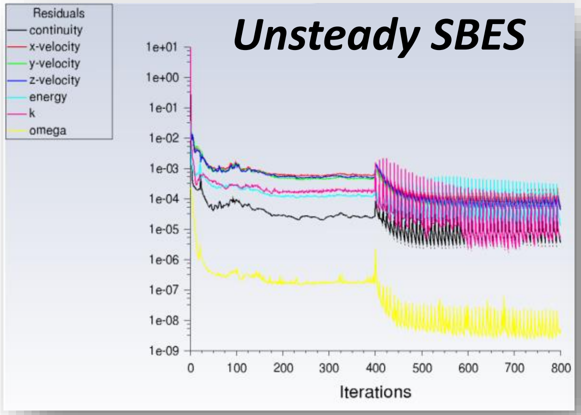
Dual Time-Stepping

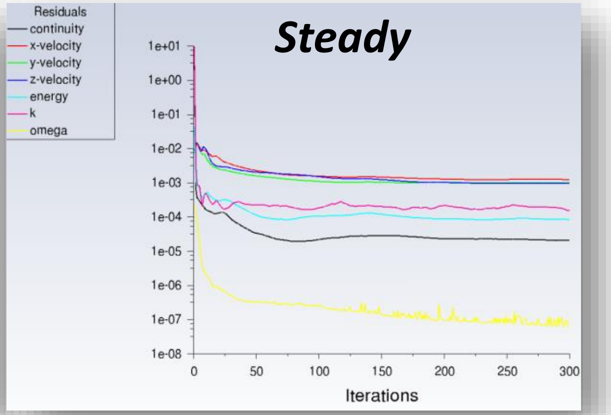
Dual time-stepping algorithm provides robust convergence for many difficult industrial cases

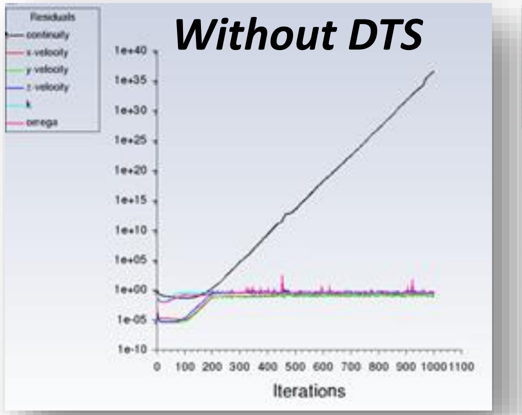
- Available for pressure-based segregated solver in 2022 R1
- Transient and steady formulations
  - Transient: fixed pseudo-time CFL that controls local URFs
  - Steady: local pseudo-time stepping method (similar to existing Pseudo-Transient, but with *local* time scale)
- Can be enabled for all equations or selectively
  - Momentum, energy, turbulence, combustion scalars
- Potential to also improve throughput via better convergence and fewer iterations required

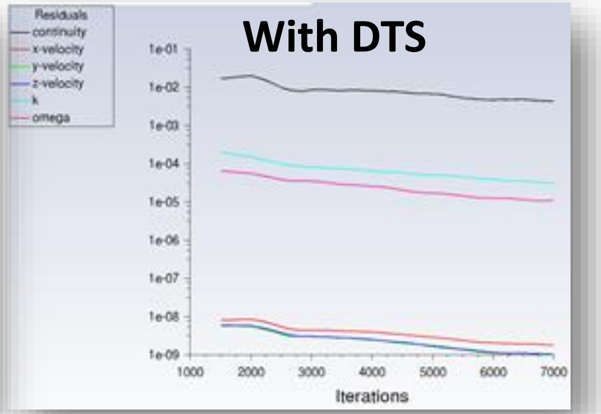


Solver Robustness: Dual Time-Stepping (DTS) Examples









Industrial Aero Case #1

- **Without Dual Time-Stepping**
  - SIMPLEC, steady → diverged
  - SIMPLEC unsteady → diverged
  - SIMPLEC unsteady *with PMN for 2M cells* → converged, very high cost
- **With Dual Time-Stepping**
  - SIMPLEC, steady and unsteady → all converged
  - Default PMN

Customer ext. aero #2

- **Without Dual Time-Stepping**
  - SIMPLEC, steady → diverged
  - SIMPLE, steady → diverged
- **With Dual Time-Stepping**
  - SIMPLE/SIMPLEC, steady/unsteady → all converged
  - No PMN required

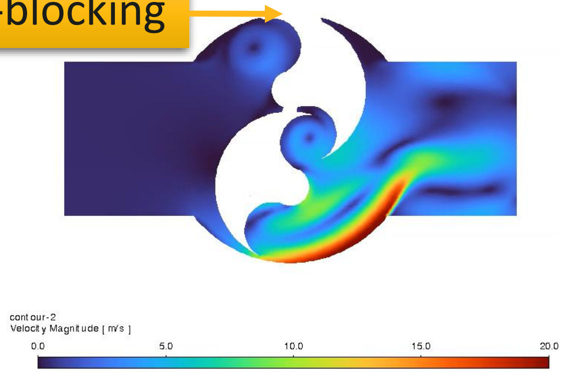
*PMN = Poor Mesh Numerics

Flow Modeling in Thin Gaps

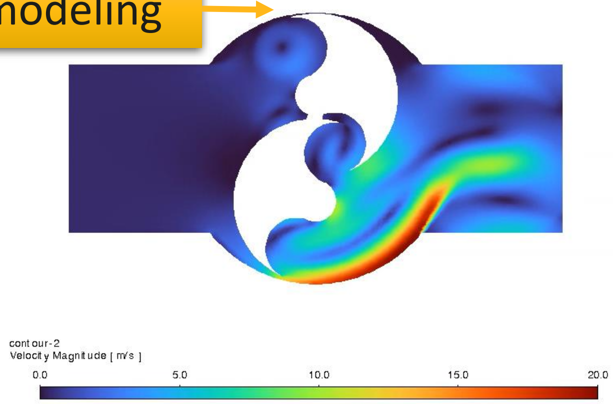
Provides a modeling solution for flow through narrow gaps in moving mesh simulations

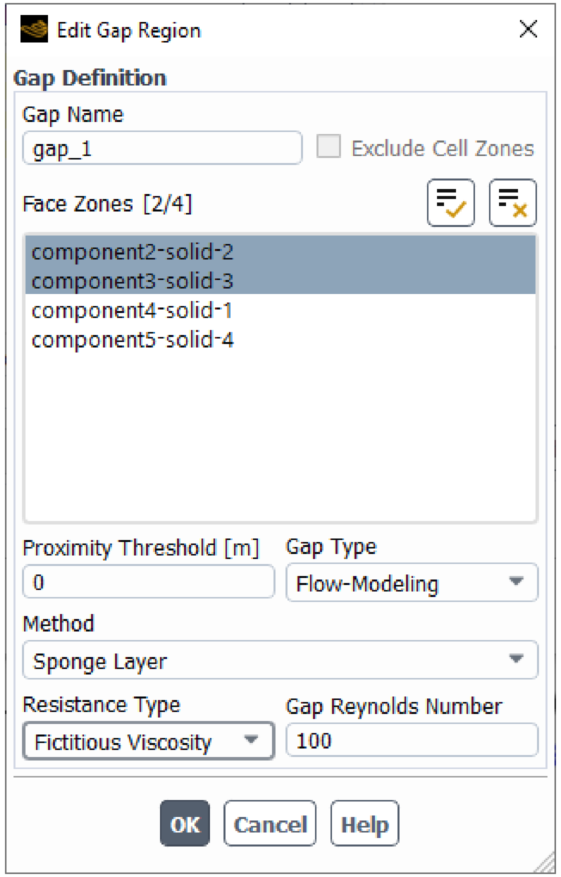
- Alternative to fully-blocking flow in narrow gaps
- Sponge-layer method artificially increases viscosity based on user-supplied Reynolds number in the gap region
  - **Fictious Viscosity:** scaled viscosity in discretization of momentum equations only. Other equations use scaled velocities from momentum equations.
  - **Real Viscosity:** scaled viscosity effects are observable in all equations

Flow-blocking



Flow-modeling



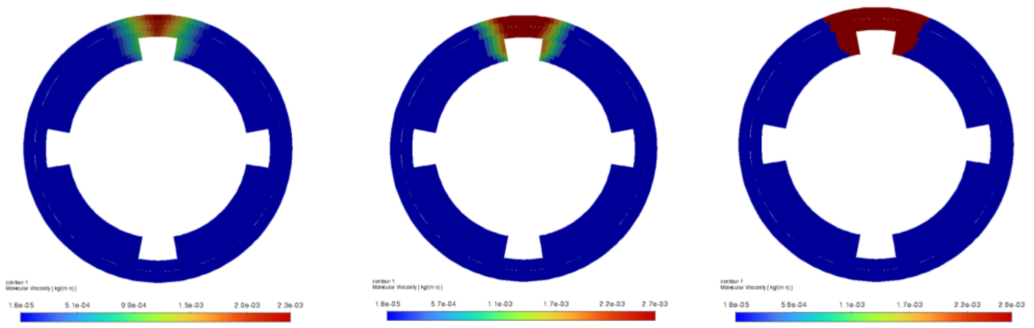


- User-Defined Source method with DEFINE_GAP_MODEL_SOURCE

Gap Model: Other Options and Enhancements

- Variable viscosity for flow modeling treatment
  - By default, a **fixed viscosity** treatment is used which applies the scaled viscosity uniformly in the gap region
  - Optionally, a variation factor [0..1] can be applied which applies a linear variation through the region
    - **1** ≡ Fixed: provides best results for very small gaps
    - **[0..1]** ≡ Variable: appropriate for larger gaps and can improve solver stability
  - `/define/gap-model/advanced-options/sponge-layer`
- Option to extend mark regions
- Robustness improvements in marking algorithm

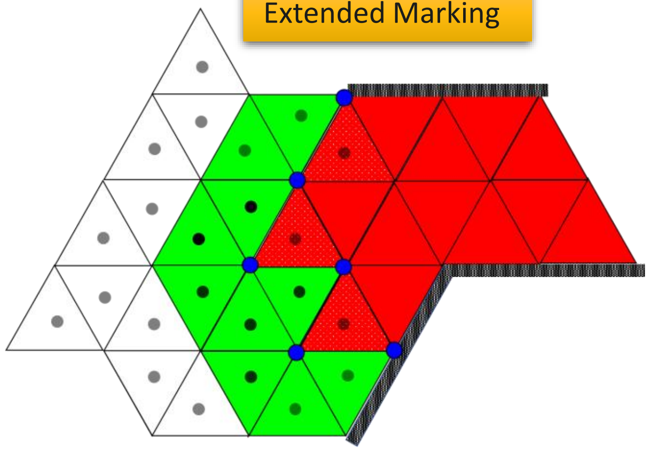
`Set viscosity variation factor ([0.0] Fully Variable, [1.0] Fixed) (0. 1.) [1] 0.5`



Variation factor = 0

Variation factor = 0.5

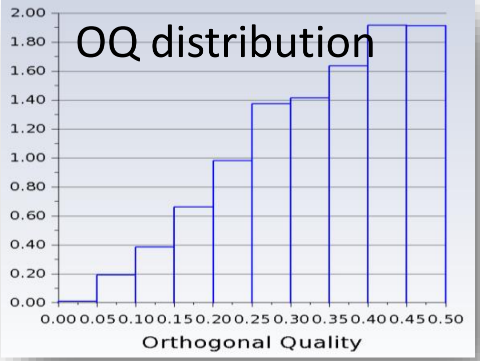
Variation factor = 0.5



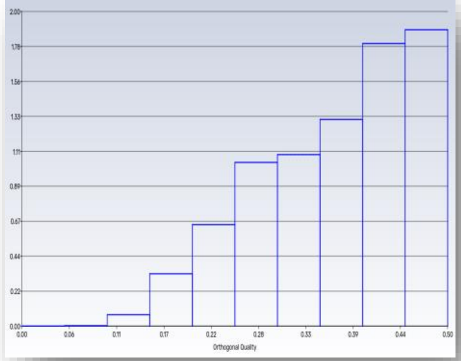
Solver Robustness: Orthogonality Enhancing Metrics

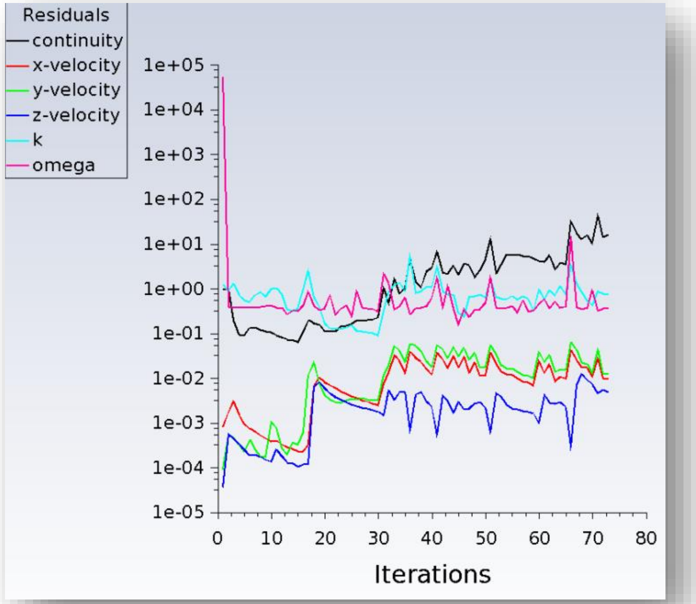
Orthogonality Enhancing Metrics (OEM) improve robustness on cases with poor-quality cells

- Part of the Poor Mesh Numerics (PMN) portfolio
- Tweaks effective cell-centroid of poor-quality elements
- This improves orthogonal quality and solver robustness

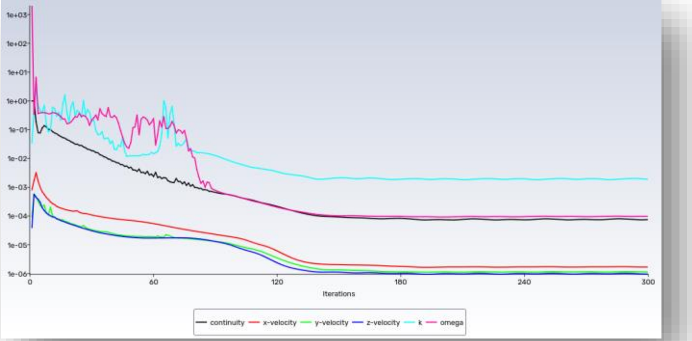


with OEM





SAE body convergence:
default vs. OEM

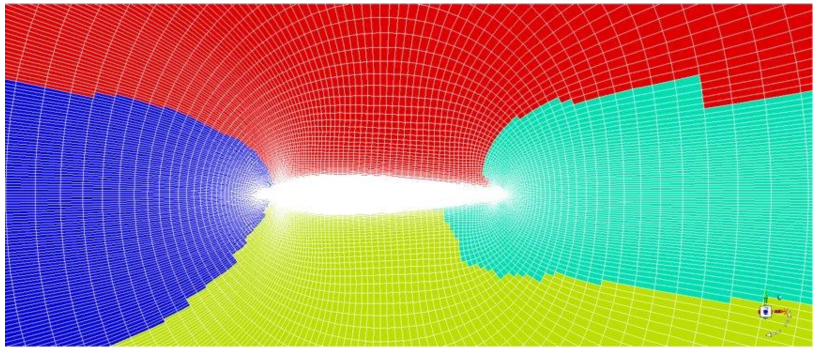


- Activate in TUI with specified Orthogonal Quality threshold

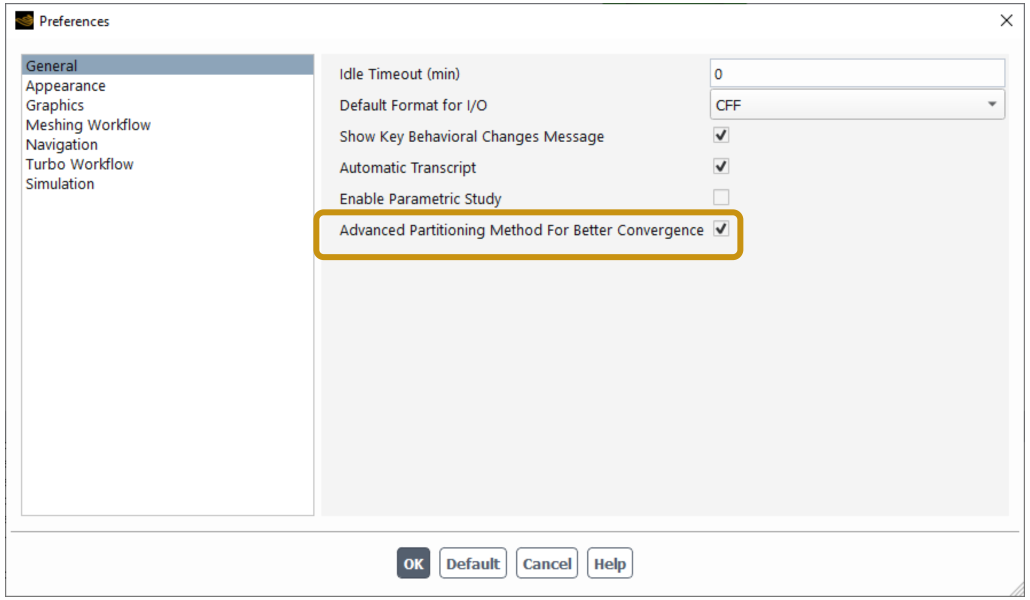
```
/solve/set/poor-mesh-numerics> orthogonality-enhancing-cell-centroids?
Relocate select cell centroids, to improve orthogonality metrics and solution stability? [no] yes
Orthogonal Quality Threshold [0.1]
```

Partitioning Enhancements for Solver Robustness

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



- Enhancements for extruded cases
  - Correct the stretching aspect ratios in extruded prism cells
  - Leads to significant improvement in load balance for extruded 3D cases
- Provide a user-preference to always enable Laplace partitioning
  - Laplace partitioning generally provides improved convergence, robustness, and scalability
  - With preference enabled, cases that are read will be repartitioned with Laplace Smoothing if required



Multiple Reference Pressure (MRP) Optimization

Significant memory reduction for cases using multiple reference pressure formulation with many cell zones

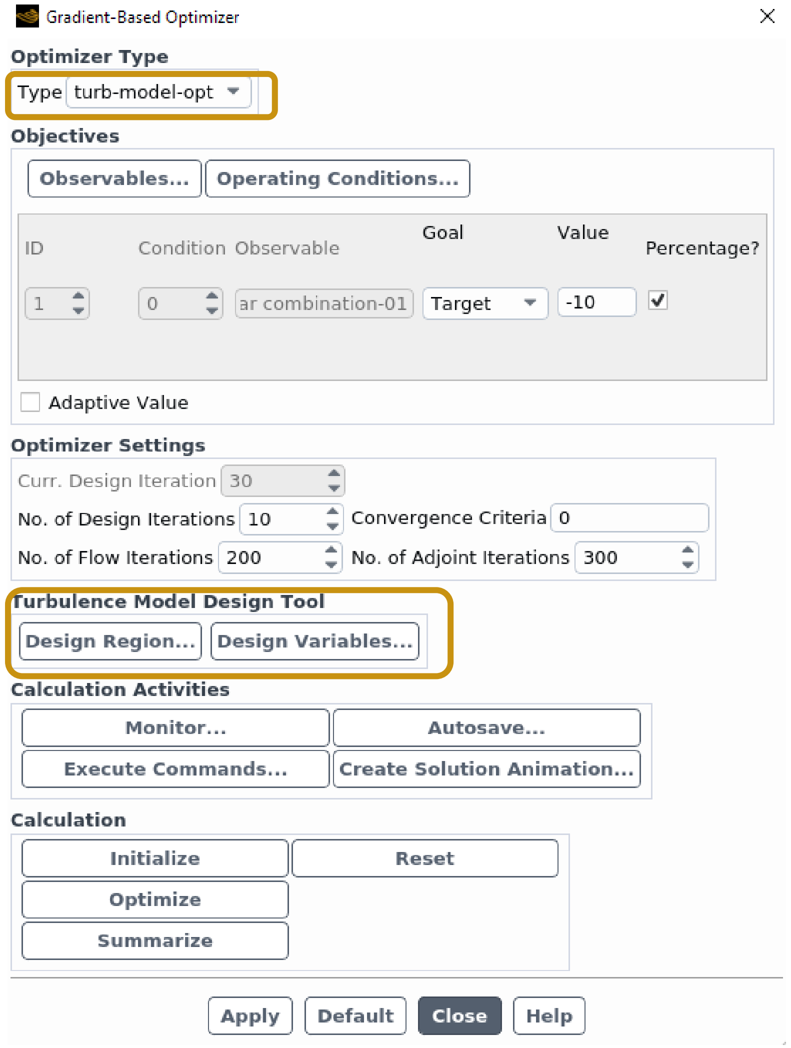
- Consolidated and enhanced the reference pressure handling
  - Enhanced robustness, memory usage, and performance
- Most noticeable for cases with many cell zones

480 core run	Virtual Memory (GB)	Resident Memory (GB)
2021 R2	1187	503
2022 R1	903	438
Improvement	**31%**	**15%**

Turbulence Model Optimizer

Train the GEKO turbulence model to yield results more like a high-fidelity, high-cost scale-resolving simulation

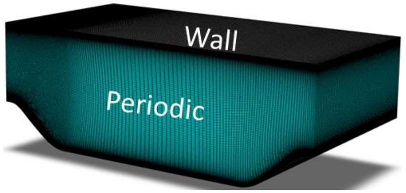
- Workflow:
  - Training:
    - Optimize turbulence coefficients to match data from high fidelity simulation/experiment
      - Scalar quality data, e. g. drag coefficients
      - Field data, e. g. time-averaged velocity from SBES Simulation.
    - Devise Neural Network to generalize the correlation between the optimized GEKO coefficients and flow features.
      - Design variables: GEKO coeff.: CSEP, CMIX, CNW, Blending function
      - Flow features: Pre-selected and customizable
  - Deployment:
    - Incorporate the devised neural network into other similar simulations to reproduce tuned-GEKO behavior



Turbulence Model Optimizer Example

Training:

3D Periodic Hills Case



$Re_H = 10400$

Deployment

2.5D Hump Case

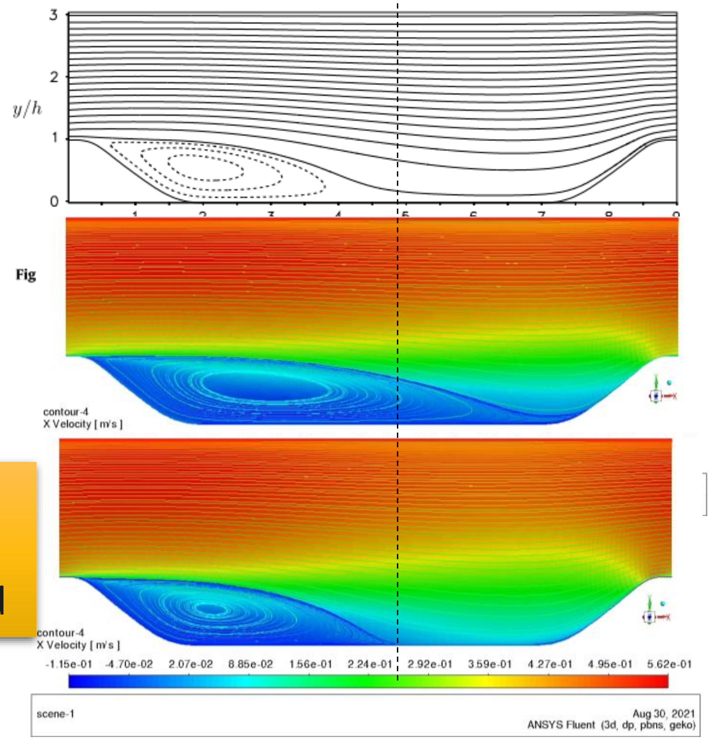
$Re_c = 968279$

M. Breuer et al

LES Reference

GEKO Default

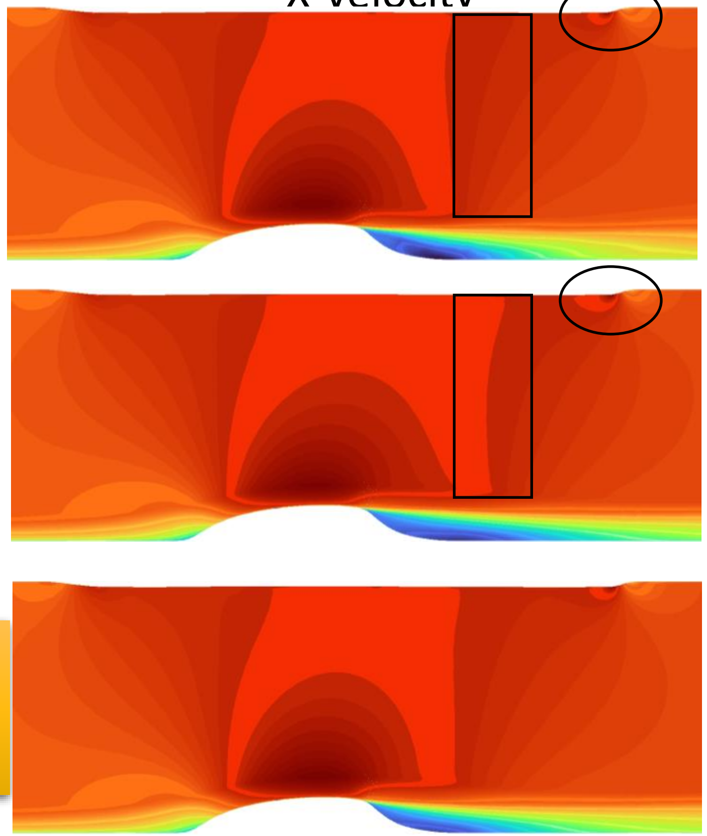
Neural Network Augmented

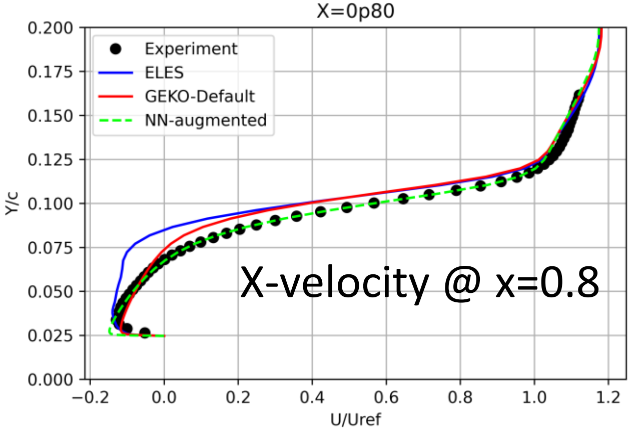


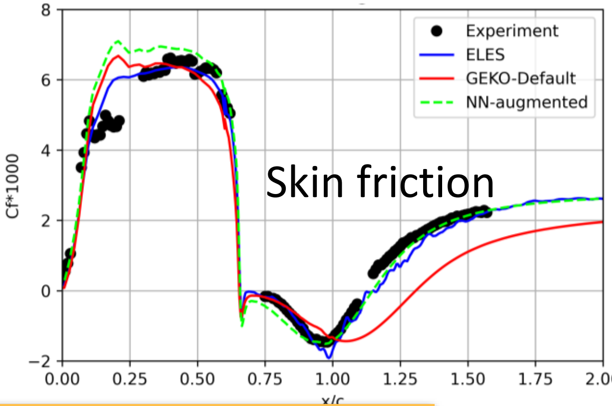
ELES Reference

GEKO Default

Neural Network Augmented





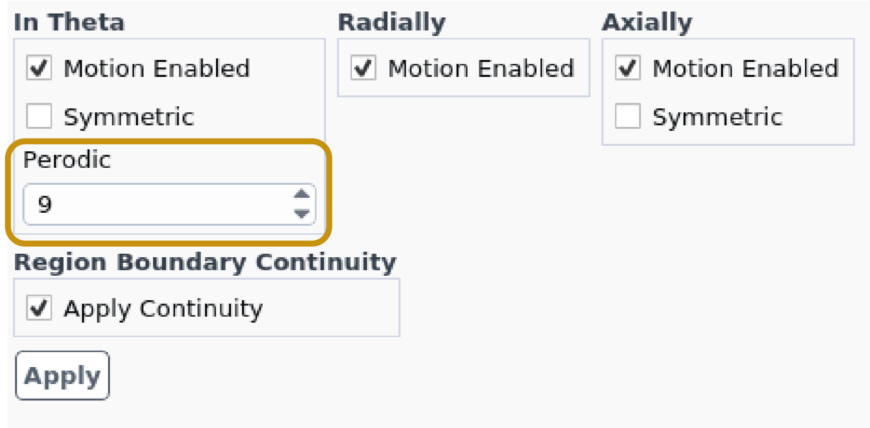


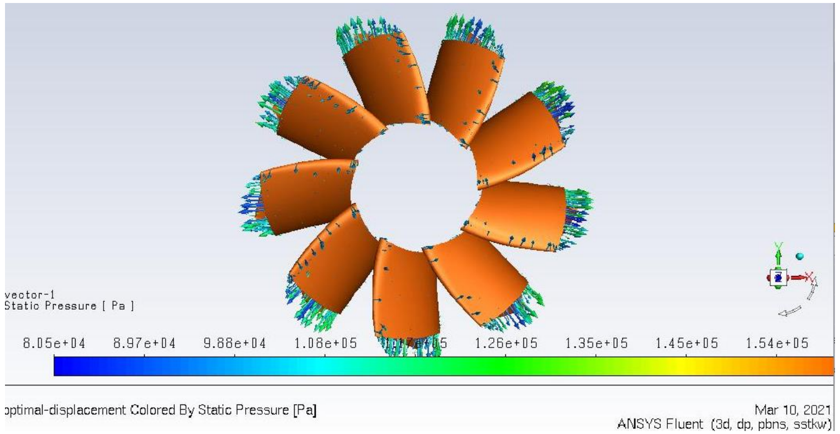
NN augmented GEKO velocity nearly match ELES/Exp.

Other Adjoint Optimization Improvements

Periodic morphing support enables optimization of turbomachinery.

- **Periodic Morphing**: enforce morphing that is periodic, with N repeats in the morphing region.
  - Key requirement for optimizing turbomachinery equipment where all blade shapes need to be identical
- Support of distance-based Rhie-Chow flux in the adjoint solver
  - Consistency in available Rhie-Chow formulations between core flow solver and adjoint solver





Ansys