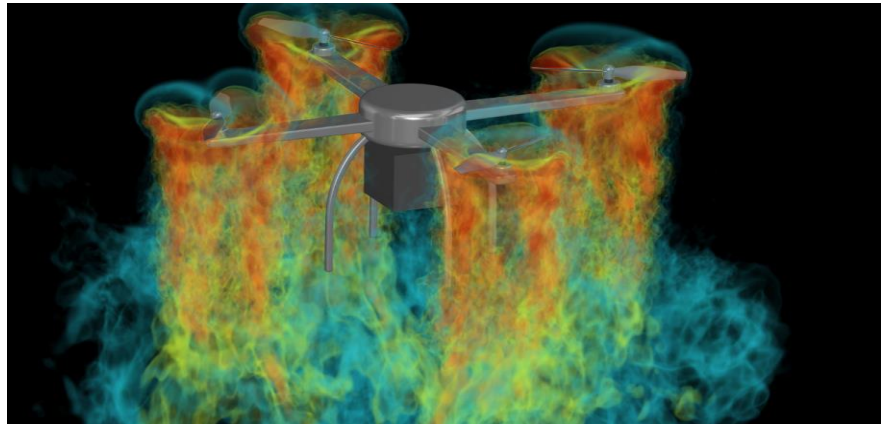
Release 2023 R1 Highlights
Ansys Fluent

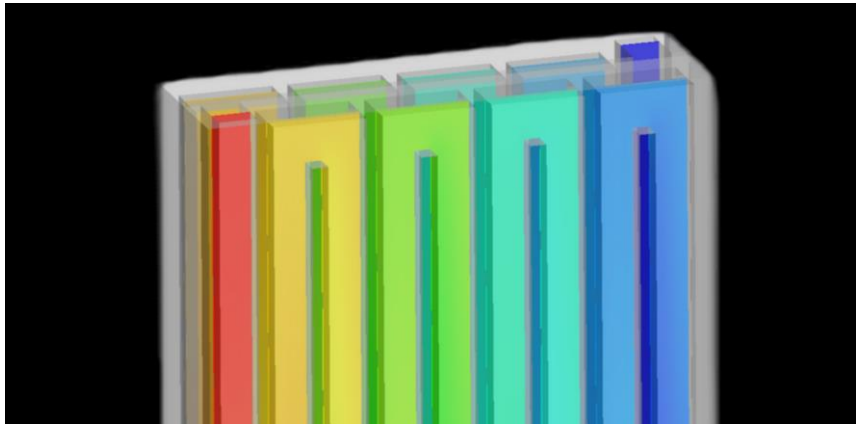
Top 2023 R1 Highlights



Full Release of Multi-GPU Solver

- ✓ Multi-GPU Solver in Fluent is now fully released
- ✓ Support for species transport, moving reference frames and enhanced numerics for LES simulations
- ✓ Beta support for reacting flows, sliding mesh & compressible flows
- ✓ Reduce simulation solve time (6 high-end GPUs > 2,000 CPUs) and total power consumption
- ✓ Addresses a broad spectrum of automotive, aerospace and energy industrial applications
- ✓ Thorough validation with canonical & industrial cases for both experiment data and CPU solver

Learn More →



Models for Hydrogen Value Chain

- ✓ Simulate green hydrogen production through electrolysis with new and accurate Proton Exchange Membrane (PEM) and Alkaline models
- ✓ Simulate hydrogen consumption with enhanced Polymer Electrolyte Membrane Fuel Cells (PEMFC) and Solid Oxide Fuel Cells (SOFC) models
- ✓ Address challenges in hydrogen fuel cell design for transportation and energy storage
- ✓ Sustainable, water-emitting hydrogen fuel cells help address critical climate challenges

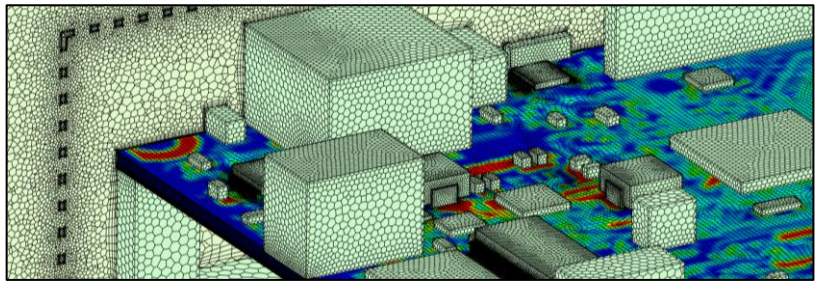
Learn More →

PyFluent Expansion

- ✓ Increase end-user productivity by building customized solutions using PyFluent
- ✓ A new embedded PyConsole brings PyFluent directly into the Fluent interface
- ✓ Support for centralized Python journaling and full scheme/TUI to python journal translation
- ✓ New usability features like auto-complete and quick search of Fluent commands (beta)

Learn More →

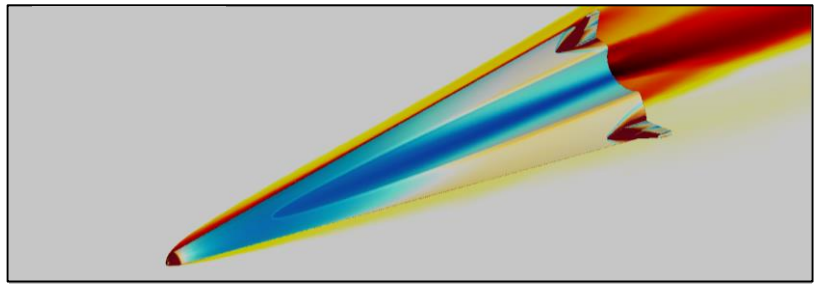
Additional 2023 R1 Highlights



Direct ECAD workflow for PCB

New workflow for printed circuit board (PCB) that enables the import of an ECAD directly from the Fluent interface without the need to use additional products

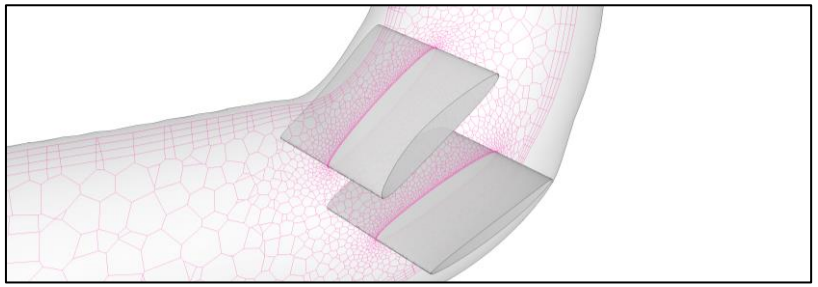
Learn More →



High-speed Numerics in Fluent Aero

Access High-Speed Numerics (HSN) within the Fluent Aero Workspace

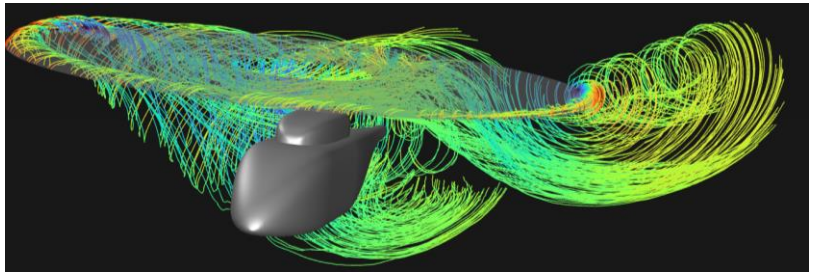
Learn More →



Embedded Parametric Workflow

A new workflow for parametric design optimization and morphing embedded in the Fluent interface

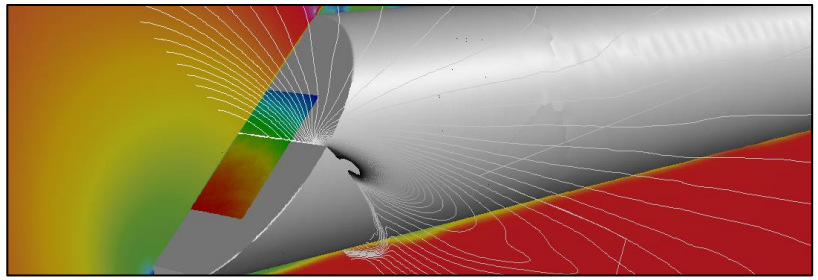
Learn More →



Virtual Blade Model (β)

A Virtual Blade Model (VBM) replace 3D rotors with actuator disks by introducing their effect as source terms in the governing equations

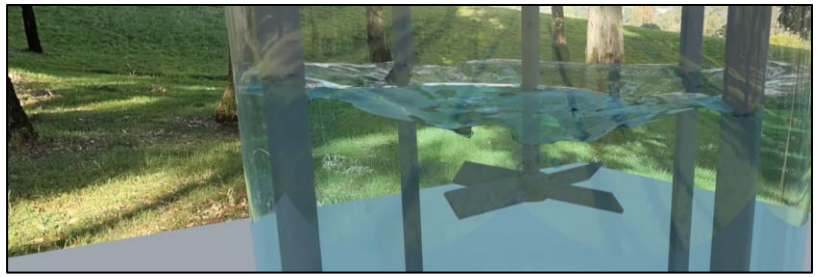
Learn More →



Built-in AeroOptical workflow (β)

Compute aberrations effects with a new embedded workflow to evaluate Optical Path Length / Optical Path Difference (OPL/OPD) directly in Ansys Fluent

Learn More →



OSPRay raytracing engine (β)

A new OSPRay raytracing engine available is now available for post-processing directly in Fluent

Learn More →

Licensing

CFD Pro, Premium and Enterprise with Fluent

CFD Pro

Entry level physics capabilities

- Steady state flow
- *NEW* Transient flow
- Heat transfer
- CHT
- Porous media
- Massless/inert particle tracking
- Multi-stream component mixing
- Rotating reference frames
- 2-equation RANS turbulence
- Compressible and incompressible
- Non-Newtonian fluids
- Watertight Meshing workflow
- Materials Processing – entry level extrusion, co-extrusion, fiber spinning, blow molding and thermoforming

CFD Premium

Full range of physics models

- All physics models in the Fluent CPU solvers
- Full Fluent Meshing capabilities, including Fault Tolerant Meshing workflow

CFD Enterprise

GPU Solver and additional modules

In addition to CFD Premium features:

- Fluent GPU solver
- Fluent Aero workspace
- Fluent Icing workspace
- All physics models in the Materials Processing workspace

Transient flow modeling is now available in Fluent with a CFD Pro license

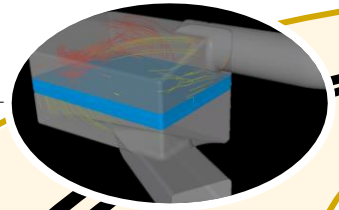
GPU Solver

Fully Native Multi-GPU Solver in Fluent

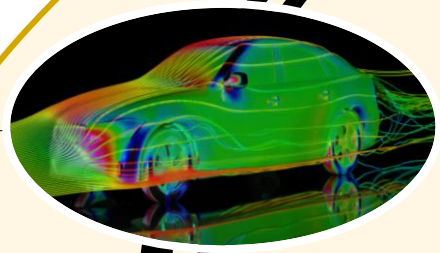
UTILIZE THE POWER OF MULTIPLE GPUS TO ACCELERATE YOUR CFD SIMULATIONS

Beta support for :
- Single/multi-GPU (shared / distributed memory)
- Steady & transient simulations
- Incompressible & subsonic compressible flows
- All mesh types
- Ideal Gas and Materials with variable properties
- Turbulence: standard k-e, SST, GEKO, RKE, SBES
- Solid conduction and CHT
- Porous media
- Windows and Linux

2022/R1

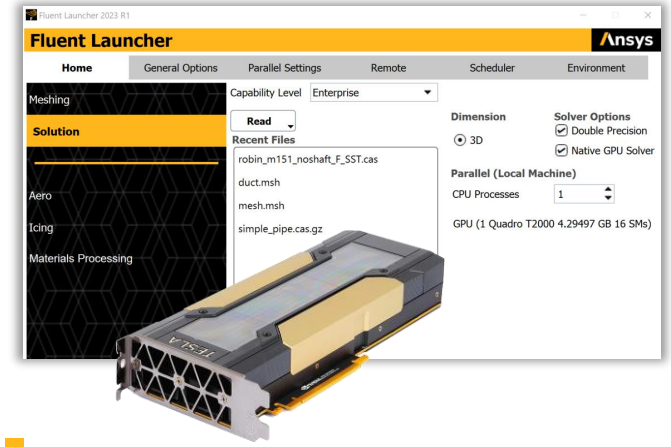


2022/R2



Beta support for :
- Transient scale-resolving simulations
- Non-conformal mesh interfaces
- Moving walls & Moving Reference Frame

2023/R1



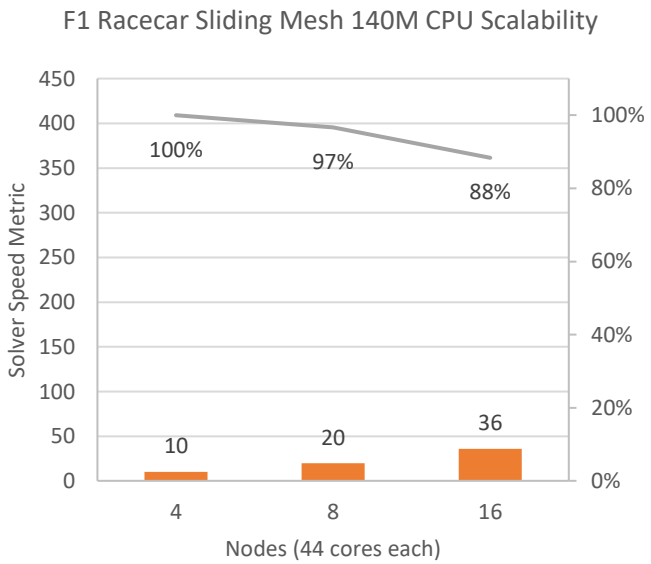
All previous features released, along with :
- Enhanced numerics
- Enhanced RANS for robustness and accuracy
- Species transport
- Enhanced LES
- Enhanced monitors: Uniformity, HTC, Porous force, Moments, Volume averaging
- Velocity directions for pressure inlet , Porous jump, Intake fan, Outlet vent , Wall thickness
- Enhanced Handling of unsupported features
- Profiles: Primitive variables for boundary
- Setup time reduction & simplified UI
- Enhanced Materials: Piecewise-linear, Polynomial, Piecewise-Polynomial , Viscosity: Sutherland
- Initialization workflow enhancement
- Launcher enhancement

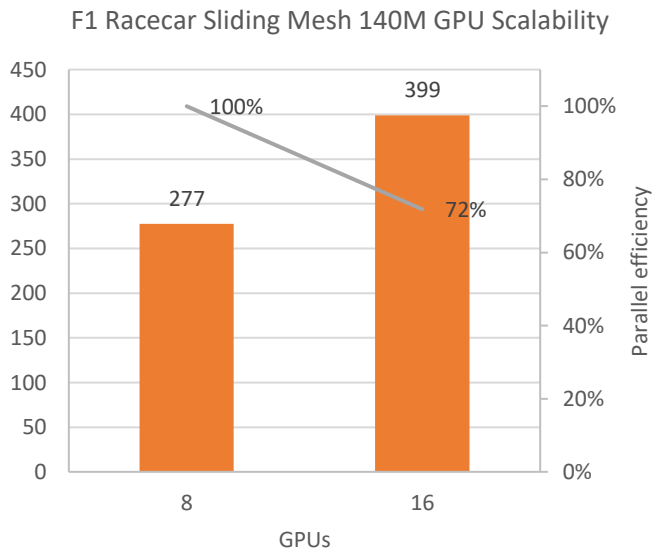
Beta support for :
- Sliding meshes
- Non-stiff reacting flows

GPU Solver: Sliding Mesh (beta)

F1 Racecar 140M cells

- SST k-omega, transient
- Computing resources
  - Intel Platinum 8168 dual-socket cluster
    - 44 cores per node
  - Nvidia A100 40GB 8-GPUs per node
- 1 GPU delivers the performance of 16 CPU nodes (704 cores)

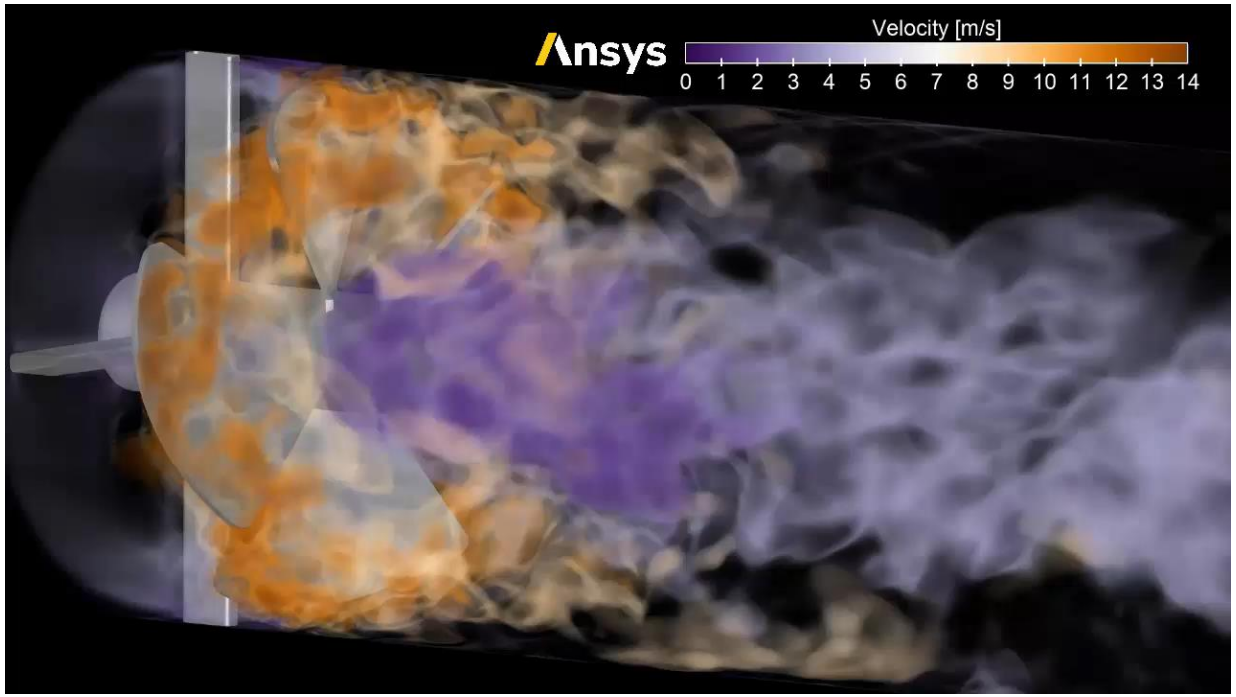




1000 timesteps in 40 minutes on 8 GPUs!

GPU Solver: Sliding Mesh (beta)

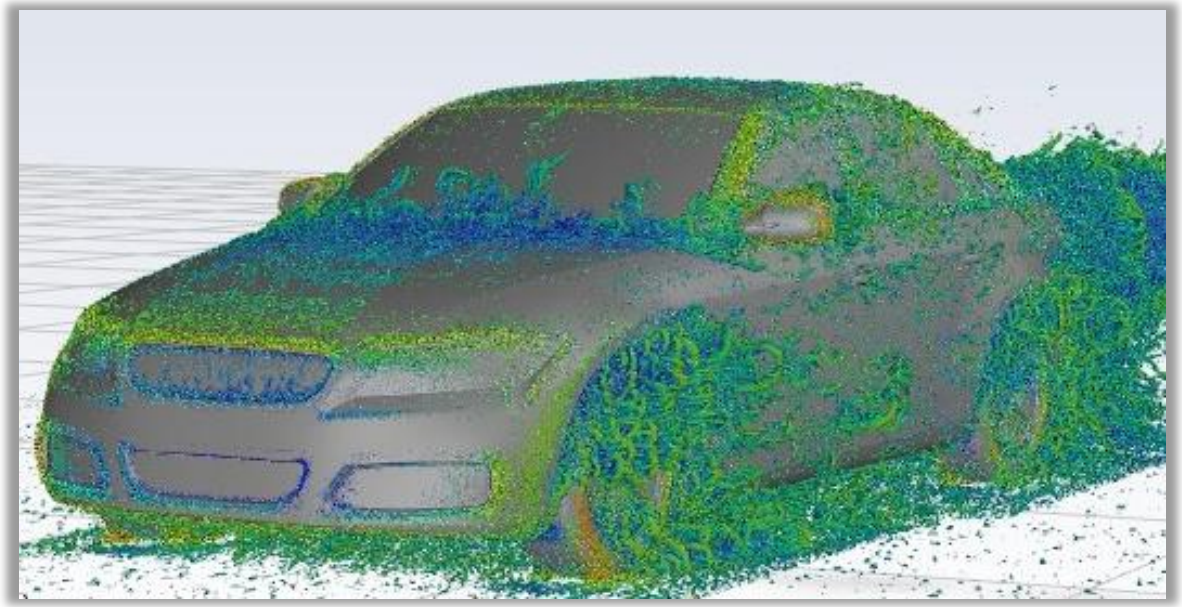
Drone Simulation with Sliding Mesh on Multiple GPUs

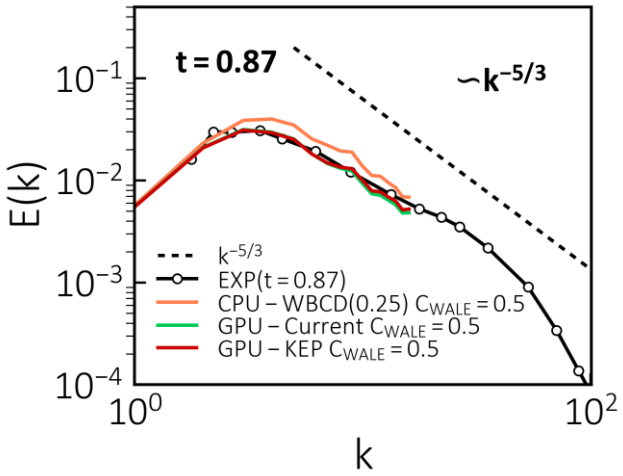


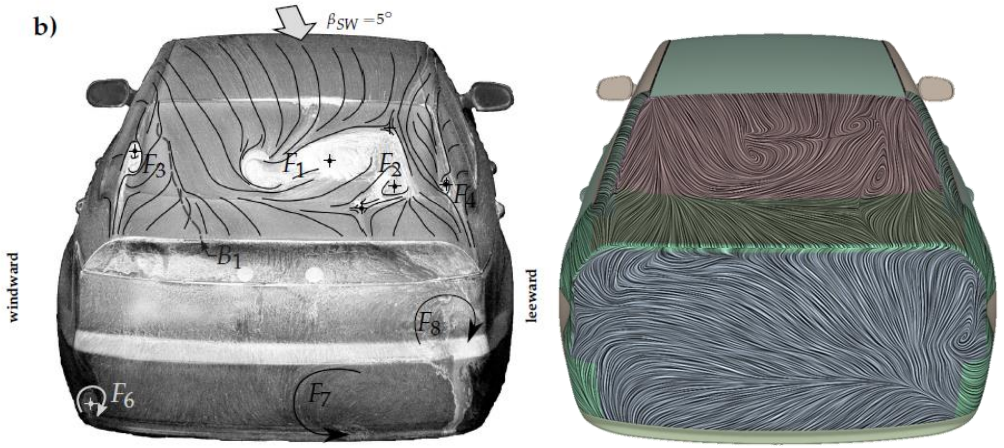
Axial Fan Simulation with Sliding Mesh on Multiple GPUs

GPU Solver: Enhanced LES Numerics

- Excellent match to experimental results and CPU results on real-world cases







Flow topology well captured for DriveAer test case

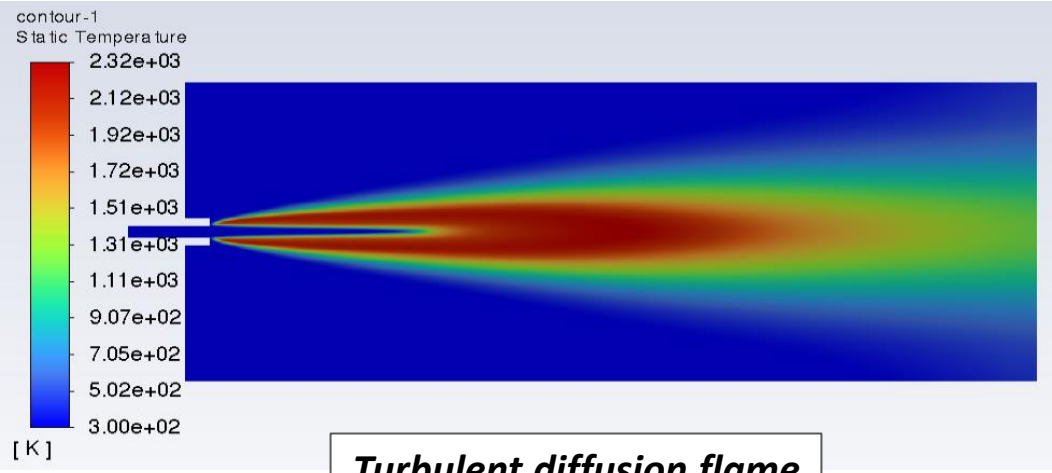
GPU Solver: Species Transport and Reacting Flows

Species Transport is now available

- Generic Species Transport Equations
- Property methods supported
  - Polynomials, Mixing Law, Kinetic theory, Volume weighted mixing law
- For variable density transient flows, e.g., bluff body, high swirl etc., robustness needs further enhancement

Chemical reactions

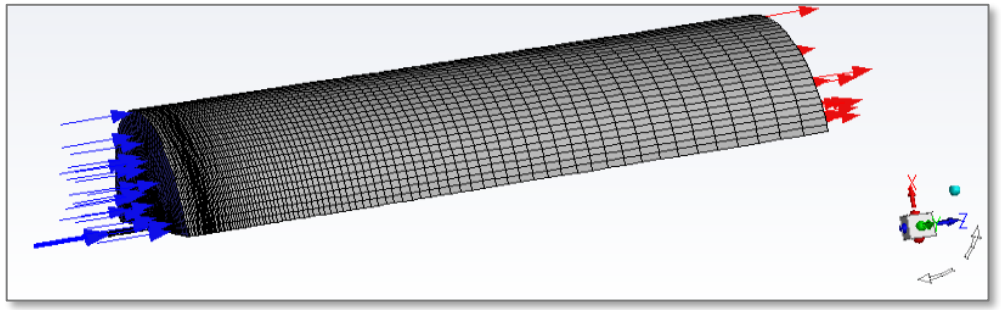
- Non-stiff solver - Eddy dissipation Combustion model (beta)

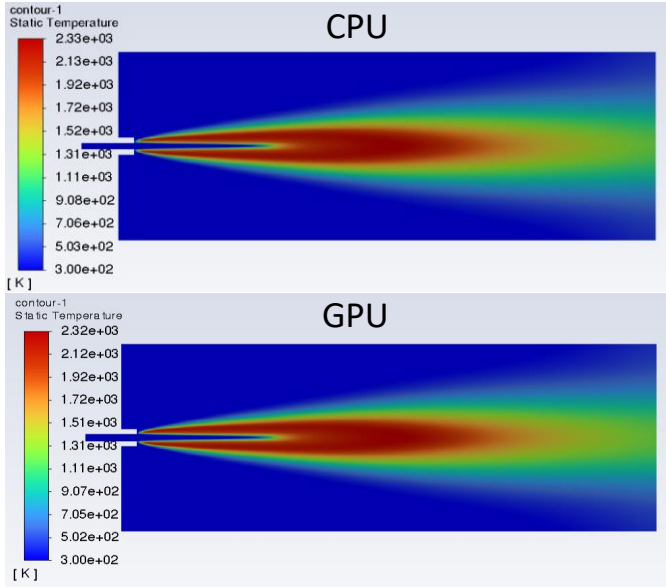


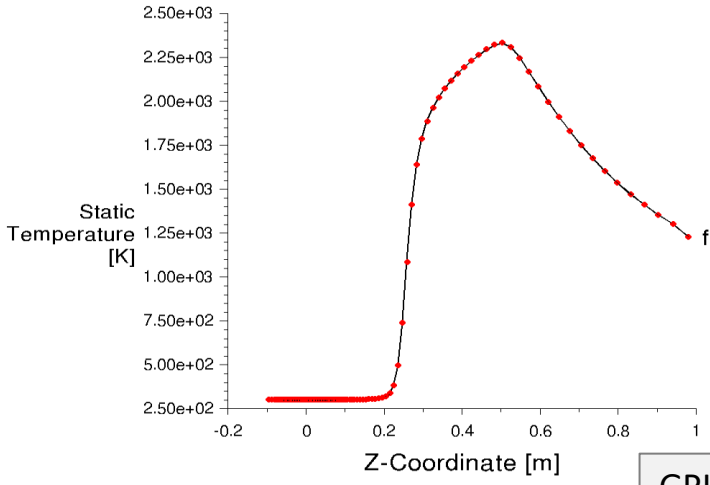
Turbulent diffusion flame

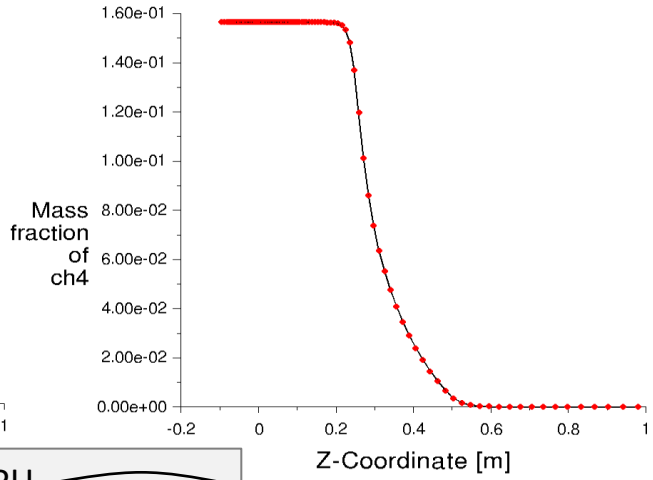
GPU Solver: Sandia Flame D Test Case

- Geometry
  - Central fuel jet with diameter = 7.2mm
  - 18.2 mm pilot (currently treated as wall)
  - Outer air inlet, Adiabatic Walls
- Modeling
  - Turbulence: Standard k-epsilon
  - Combustion: Eddy Dissipation (beta)
  - Reaction: A single step reaction is used
    - **CH4 + 2O2 -> CO2 + 2H2O**
  - Default reaction parameters (mixing rates) are used from Fluent







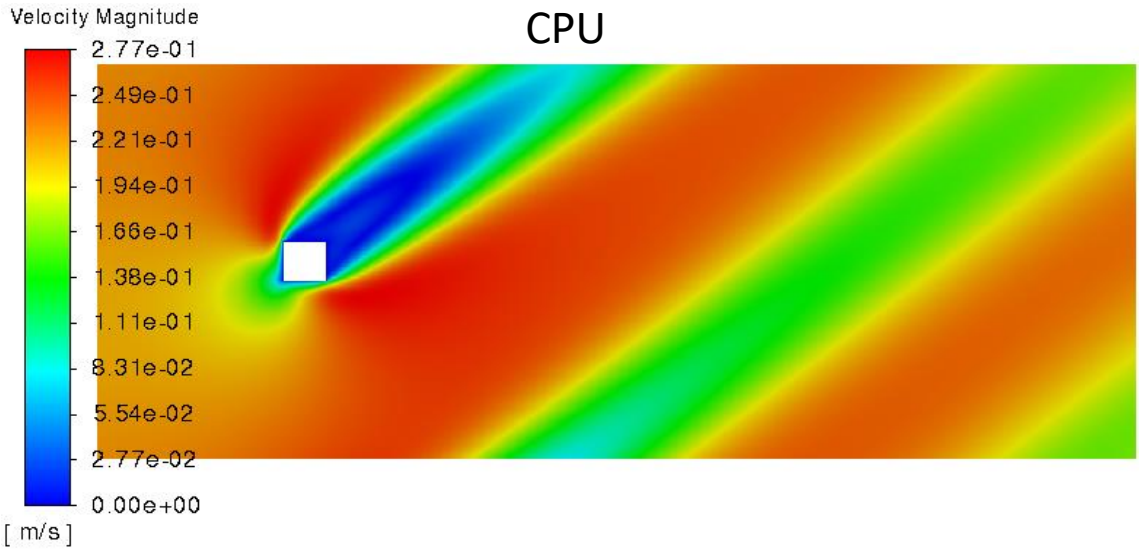


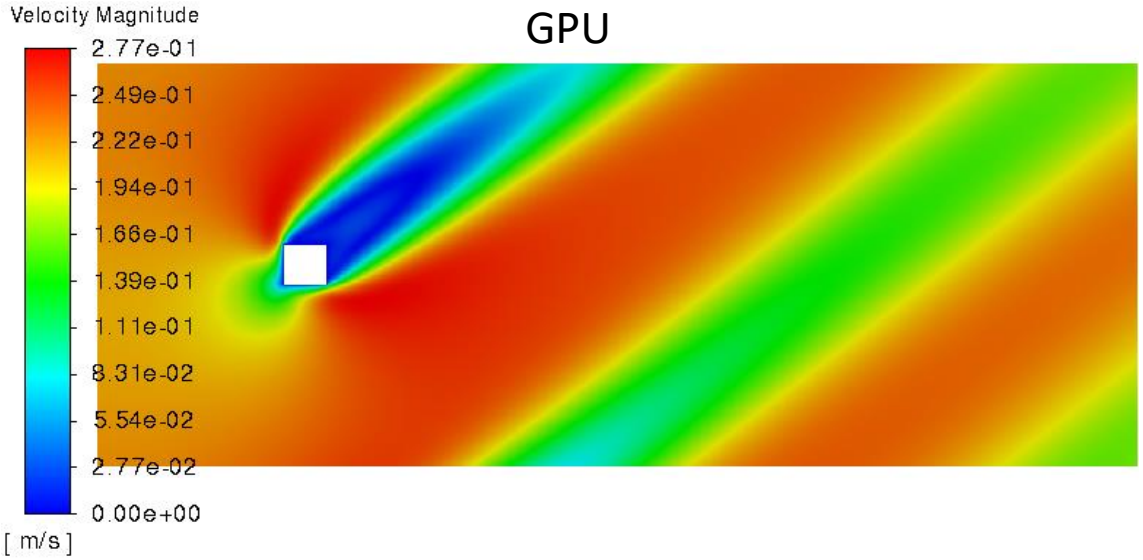
GPU
CPU

GPU Solver: More Boundary Conditions

Support for many new boundary types

- Porous jump
- Velocity directions for pressure inlet
  - Cartesian directions
    - Example, flow pass bluff body →
      - 45-deg flow direction for pressure inlet
      - Top/bottom periodic
  - Cylindrical directions
  - Local Cylindrical directions
- Intake fan
- Outlet vent





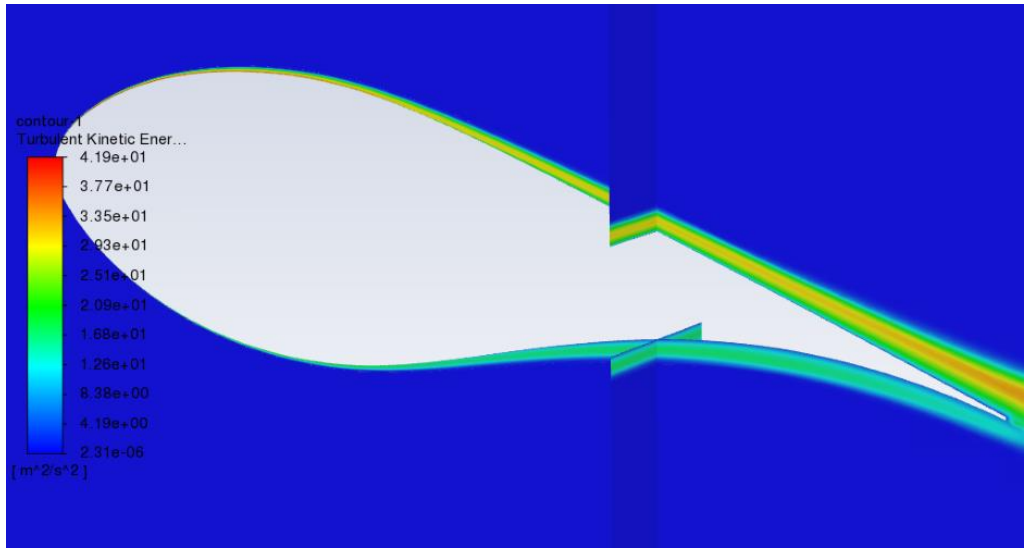
GPU Solver: Boundary Profiles

You can now set spatial profiles for boundary condition values

- Supports u, v, w, t, p, k, epsilon and omega

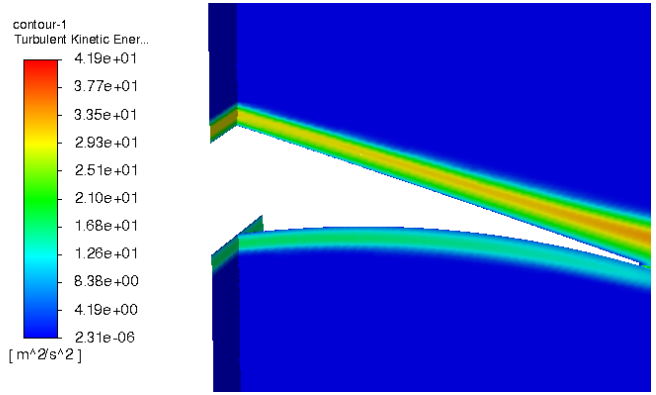
- Validation case:
  - Airfoil with SST k-w
  - Solve the full domain case then compare to a partial domain solution that uses profile boundary conditions extracted from the full domain solution

Turbulent Kinetic Energy Contour with Full Domain



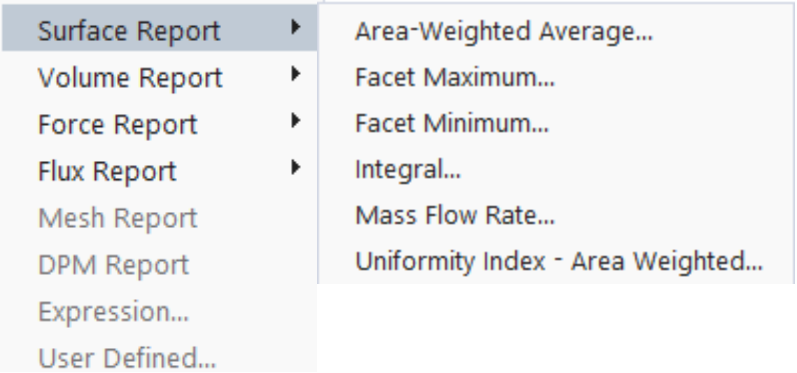
Turbulent Kinetic Energy Contour with partial domain and profiles as input from full domain case

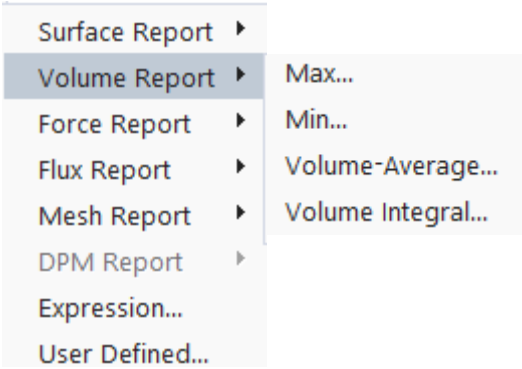
Courtesy T. Hybschmann

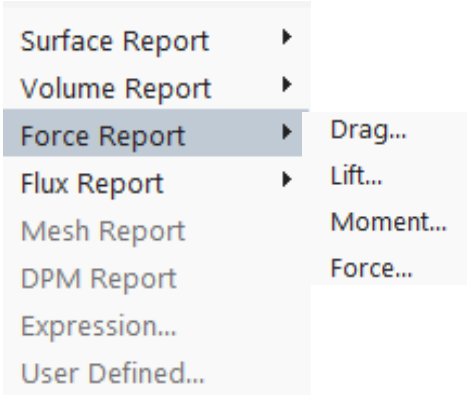


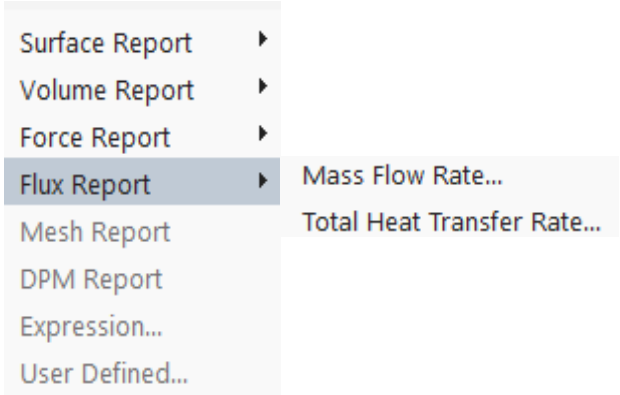
GPU Solver: Monitors Supported

- Monitors on normal zones supported
  - New for 2023 R1: Uniformity, HTC, Porous force, Moments, Volume averaging



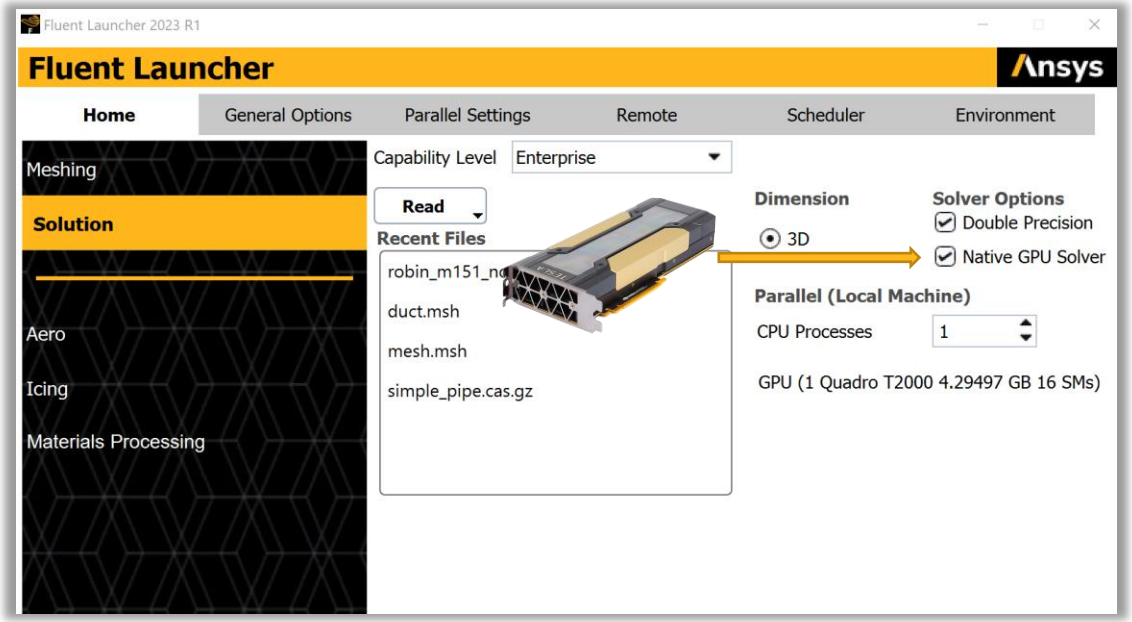






GPU Solver: User Interface

The GPU solver is now launched from Solution mode with updated launcher



Unsupported feature handling

- Most unsupported features are hidden in the GUI
  - All top-level unsupported features are hidden
  - Most sub-model level unsupported options are hidden
- Minimal TUI changes, i.e., most unsupported features still available in TUI (but cannot be invoked)
- Automatic compatibility check and case conversion is carried out:
  - At case read
  - When settings are changed by users
  - At GPU solver invocation

PyFluent

PyFluent – Opensource Accessibility for Fluent

- **PyFluent Capabilities**
  - Use Fluent within, or alongside, any other Python environment
    - In conjunction with other Ansys Python libraries/packages
    - With other external Python products
  - Access to all Fluent TUI commands for meshing, solving, and post-processing
  - Extract post-processing (field) data and use with standard Python tools
    - PyVista, NumPy, SciPy, Matplotlib, etc…
  - Build custom workflows

```
/define> materials

/define/materials> change-create

material-name> air
material name [air] air
air is a fluid
change Density? [no] yes

Density
methods: (constant ideal-gas incompressible-idea
expression compressible-liquid user-defined)
new method [constant] ideal-gas
no data required.

change Cp (Specific Heat)? [no] no
change Thermal Conductivity? [no] no
change Viscosity? [no] no
```

PyFluent (TUI API) [**22R2 release**]

```
session.tui.solver.define.materials.change_create('air','air','yes','ideal-gas','no','no','no','no','no','no')
```

PyFluent (Settings API) [**22R2 beta, expanded for 23R1 beta**]

```
root.setup.materials.fluid['air']={'density':{'option':'ideal-gas'}}
```

PyFluent Source Code, Documentation, and Help

https://github.com/pyansys

PyAnsys

PyAnsys - Ansys Python development organization

https://docs.pyansys.com/ Part of ANSYS, Inc.

Overview | Repositories 11 | Projects 3 | Packages | People 5

Source Code

README.md

Py/Ansys

Python + Ansys = PyAnsys

Welcome to the PyAnsys project.

The PyAnsys project is a collection of Python packages that enable the use of Ansys products through Python.

This project originated as a single package, `pyansys`, and has expanded to several main packages:

Documentation

- PyAEDT: Pythonic interface to AEDT
- PyDPF-Core: Pythonic interface to DPF (Data Processing Framework) for building more advanced and customized workflows
- PyDPF-Post: Pythonic interface to DPF's postprocessing toolbox for manipulating and transforming simulation data
- PyMAPDL: Pythonic interface to MAPDL
- PyMAPDL Reader: Pythonic interface to read legacy MAPDL result files (MAPDL 14.5 and later)
- PyFluent: Pythonic interface to Ansys Fluent
- PyFluent-Parametric: Pythonic interface to Ansys Fluent parametric workflows
- PyFluent-Visualization: Pythonic interface to visualize Ansys Fluent simulations using Python
- PyPIM: Pythonic interface to communicate with the PIM (Product Instance Management) API
- Granta MI BoM Analytics: Pythonic interface to Granta MI BoM Analytics services
- Shared Components: Shared software components to enable package interoperability and minimize maintenance

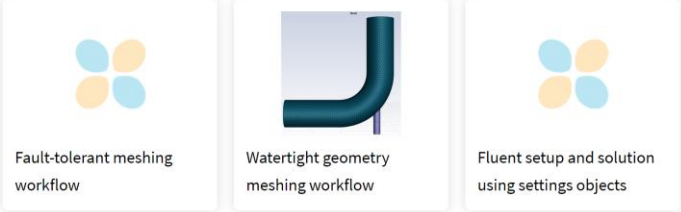
PyAnsys » PyFluent » Examples

Examples

End-to-end examples show how you can use PyFluent. If the PyFluent `ansys-fluent-core` package is installed on your machine, you can download these examples as Python files or Jupyter notebooks and run them locally.

Simulation Examples

These examples show you how you can use Fluent capabilities from Python to perform Fluent simulations. This includes geometry import, Fluent'smeshing workflows, setting up and running the solver, and reviewing the results using Fluent's postprocessing capabilities.



Each module has documentation and examples

pyansys / pyfluent Public

Code | Issues 95 | Pu

New issue

Need support?
Open an issue!

PyConsole (beta)

- What is it?
  - Enables Python journaling directly in Fluent through an embedded console
  - Python journaling plus usability features like tab completion and quick search (WIP)
- Enables a single ecosystem
  - Identical Python interfaces in PyFluent and PyConsole, requiring a single set of scripts
  - Centralized Python journaling in Fluent generates scripts for both PyFluent and PyConsole
  - Full Scheme/TUI to Python journal translation built in to Fluent
  - Everything underpinned by a solid API platform, used across multiple initiatives

PyConsole (beta)

- PyConsole interactions will be familiar to PyFluent users
- Meshing workflow part of PyConsole is already a mature, full feature in Fluent Meshing
- PyConsole offers additional usability features

```
>>> inlet_6 = solver.setup.boundary_conditions.velocity_inlet["velocity-inlet-6"]
>>> inlet_6.vmag()
{'option': 'value', 'value': 21.0}
>>> inlet_6.vmag = 20.5
>>>
>>> inlet_6.turb_intensity()
0.06
>>> inlet_6.                    0.05
>>> inlet_6.                    tio()
10
>>> inlet_6.                    tio = 9
>>>
>>> inlet_6.                    tio()
9
>>> inlet_6.
>>> inlet_6.frame_of_reference
```

frame_of_reference
ke_spec
p_sup.
turb_intensity
turb_viscosity_ratio
velocity_spec
vmag.

Tab completion

boundary_conditions

Quick search

- Journaling in meshing and solution modes
  - In meshing, commands due to workflow GUI actions are journaled along with TUI-, Scheme- and Python-scripted commands:

```
workflow.InitializeWorkflow(WorkflowType=r'Watertight Geometry')
workflow.TaskObject['Generate the Volume Mesh'].ExecuteUpstreamNonExecutedAndThisTask()
meshing.execute_tui(r'''/file/export/stl "mesh_1" ''')
```

  - In solver, commands due to TUI-, Scheme- and Python-scripted commands are journaled:

```
solver.execute_tui(r'''/define/models/dpm/unsteady-tracking ''')
solver.setup.models.energy = {"enabled" : True, "pressure_work" : True}
```

- The journaling can be controlled both in PyFluent and in PyConsole, and the journals can be executed in both

Features common in PyFluent and PyConsole

- High-level object-oriented Python programming through identical interfaces
- Meshing workflow and solver Python object hierarchies exposing comprehensive capabilities

```
import_geometry = workflow.TaskObject['Import Geometry']
viscous_model = solver.setup.models.viscous
initialization = solver.solution.initialization
```

- Get and set properties. Either individual settings, or dictionaries of hierarchical settings

```
meshing.GlobalSettings.LengthUnit()
meshing.GlobalSettings = {'LengthUnit'='mm'}
viscous_model()
viscous_model.k_omega_model = 'sst'
```

- Get attributes

```
iterate.number_of_iterations.default()
velocity_inlet.turb_intensity.min()
meshing.GlobalSettings.AreaUnit.allowed_values()
```

- Command and query

```
import_geometry.Execute()
initialization.hybrid_initialize()
calculation.iterate(number_of_iterations=10)
```

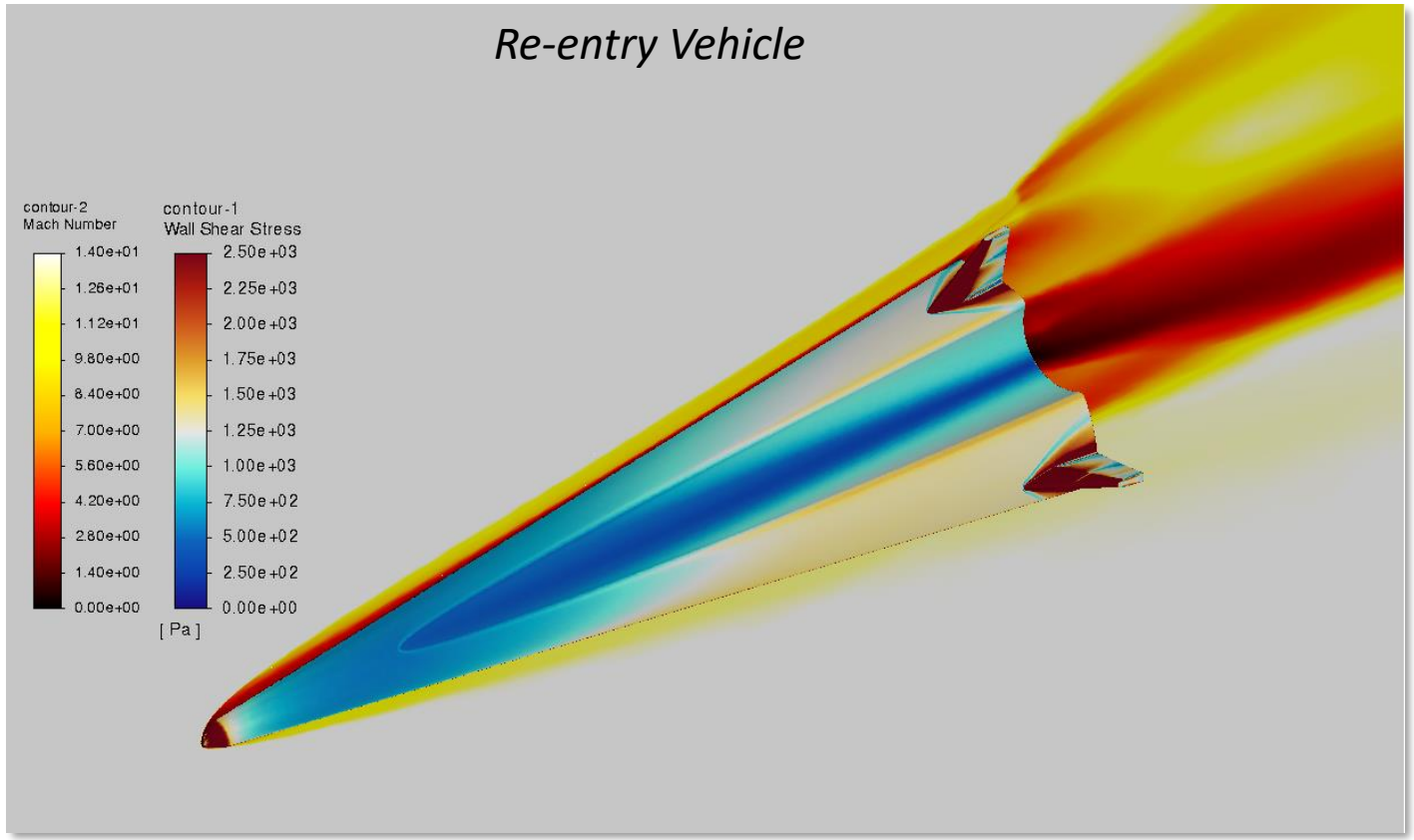
- Access to allow existing TUIs and Scheme functionality through Python

execute_tui(*Scheme code*)

Aerospace

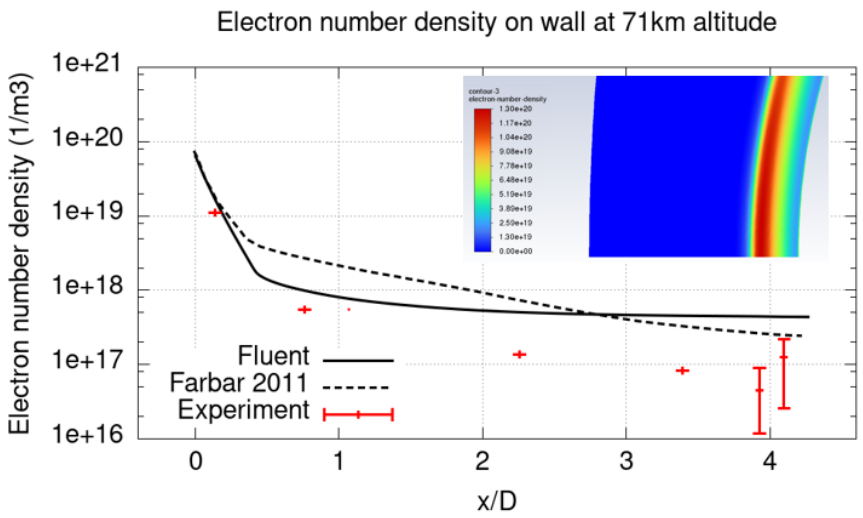
Density-Based Solver & Hypersonics

- **Numerics**
  - High-Speed Numerics now available in the GUI
  - Improved solver robustness
  - Improved robustness on high-skew meshes
  - Enthalpy preservation across shocks and discontinuities

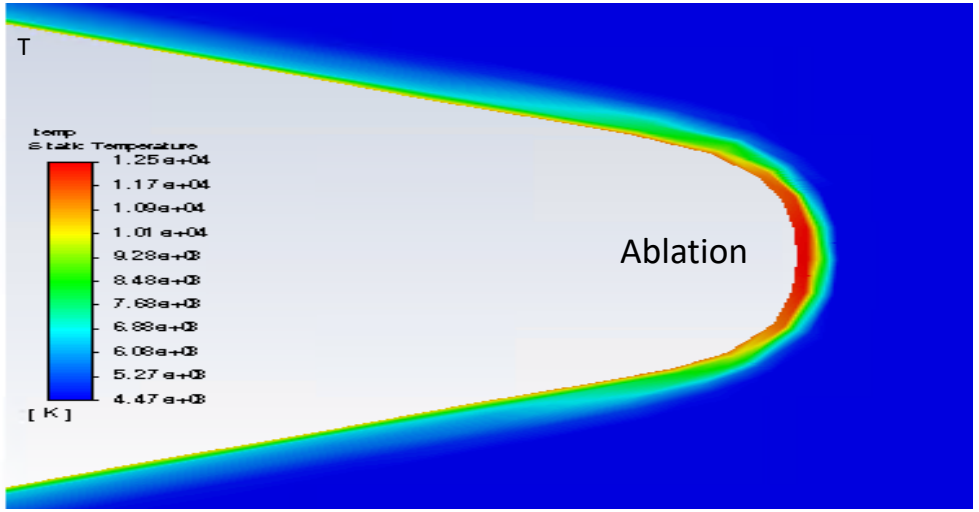


Density-Based Solver & Hypersonics

- **Two-Temperature Eq Model**
  - Weak Ionization (beta)



- **Vielle's wall ablation model**
  - Extended to include mass and heat transfer (blowing effect)
  - Only mesh-movement in earlier release

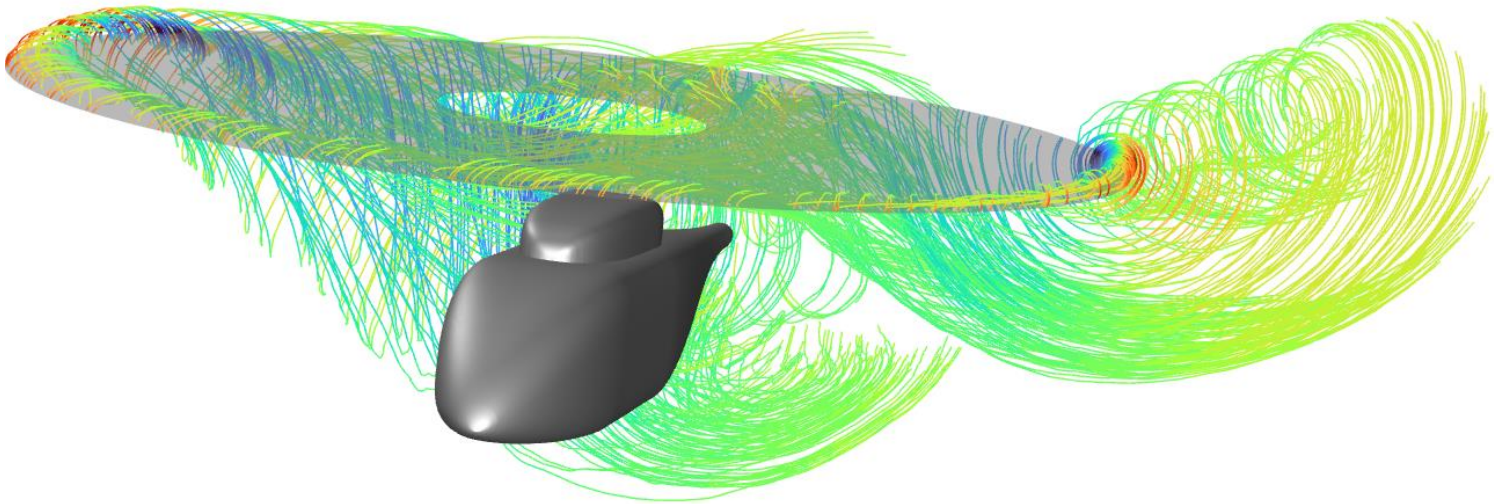


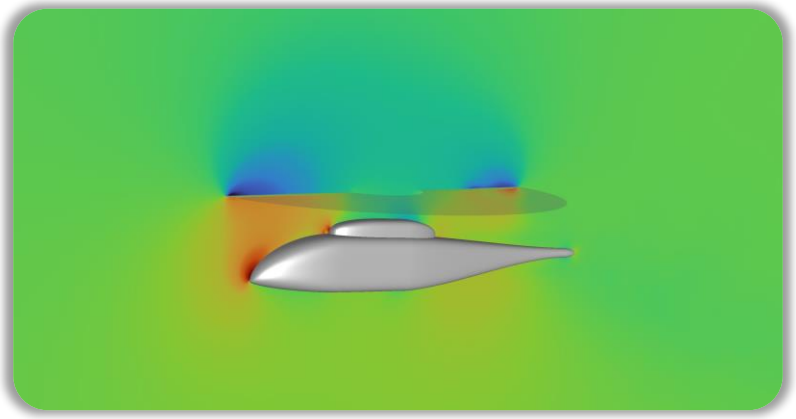
An updated license file is needed to use the Two-Temperature or Wall Ablation models. Please contact your Ansys representative.

Virtual Blade Model (beta)

The Virtual Blade Model provides an efficient approach to modeling rotating blades

- Does not resolve geometry details
- Provides bulk effects on the flow field
- **Floating disc capability**
  - No need to mesh the VBM disc as a cell zone
  - Mesh adaption can refine elements at disc if desired



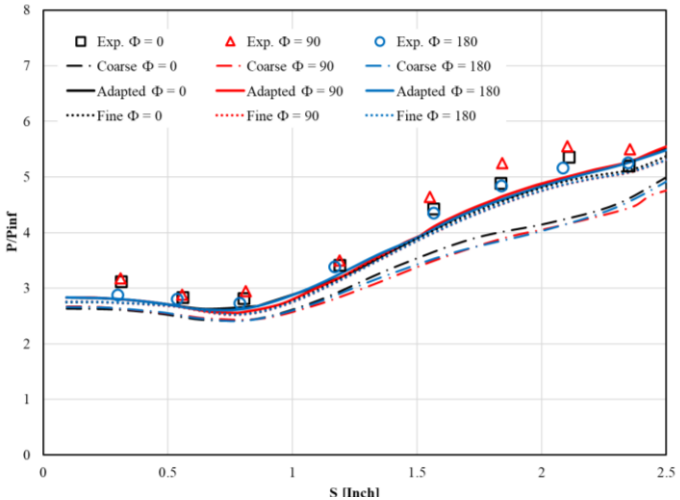


Mesh Adaption

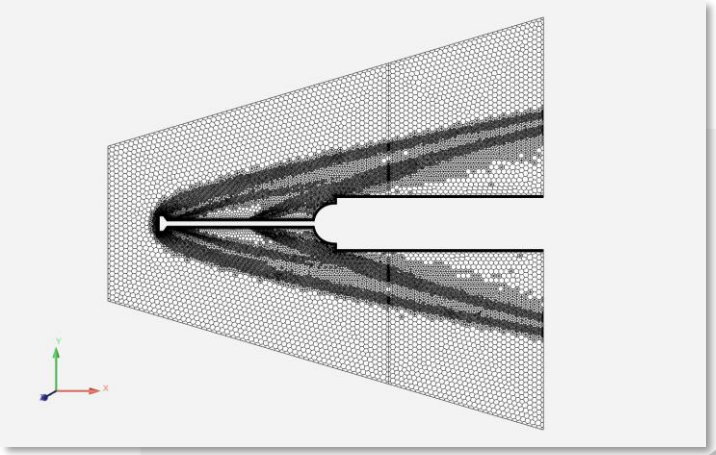
Hessian-based mesh adaption provides efficient and accurate results

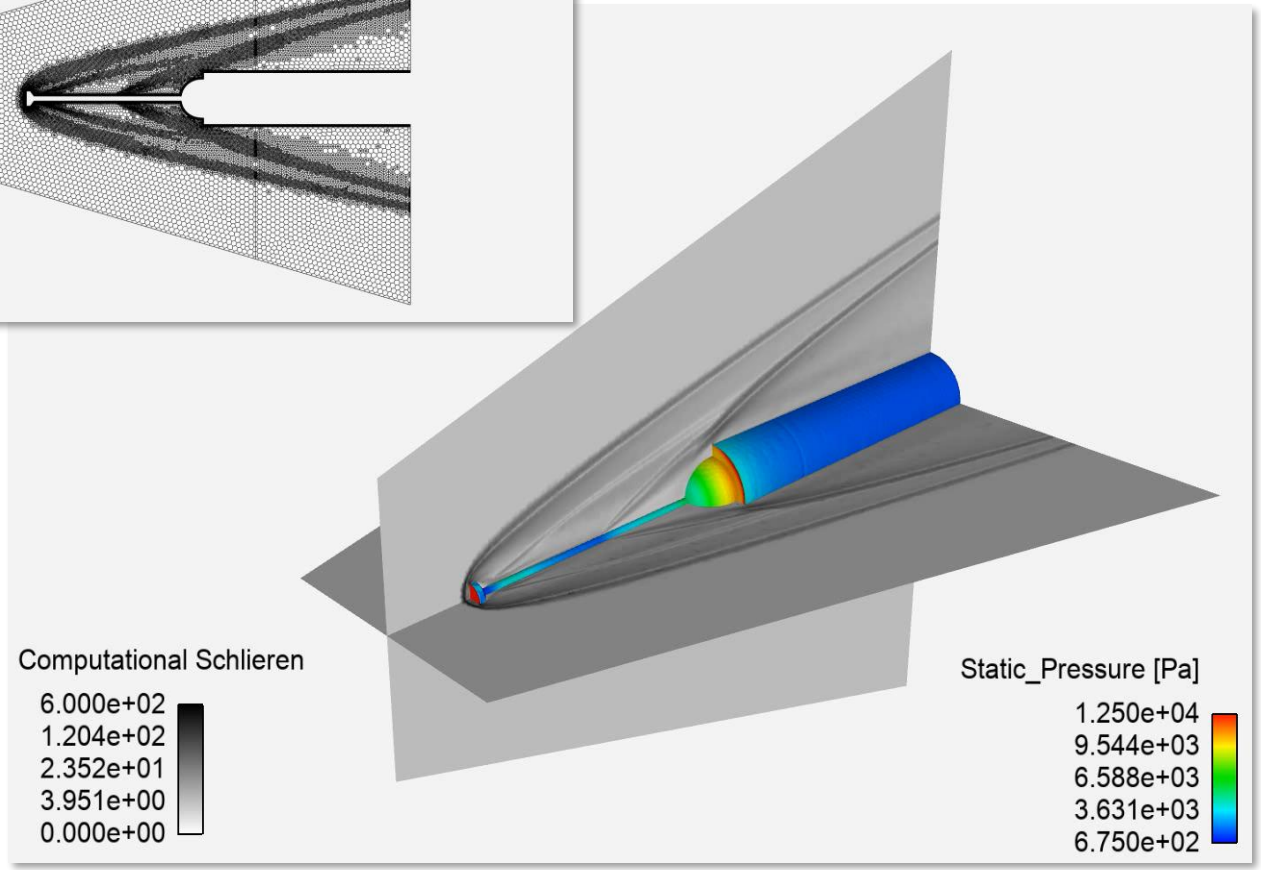
17x Speedup Compared to Fine Mesh with comparable accuracy!

Mesh	# Cells (millions)	# Cores	Run Time (s)
Coarse	0.123	64	88
Adapted	2.44	64	3094
Fine	15.65	64	53920



Experimental data from *Experimental results on the feasibility of an aerospike for hypersonic missiles.* Huebner et. al, 1995

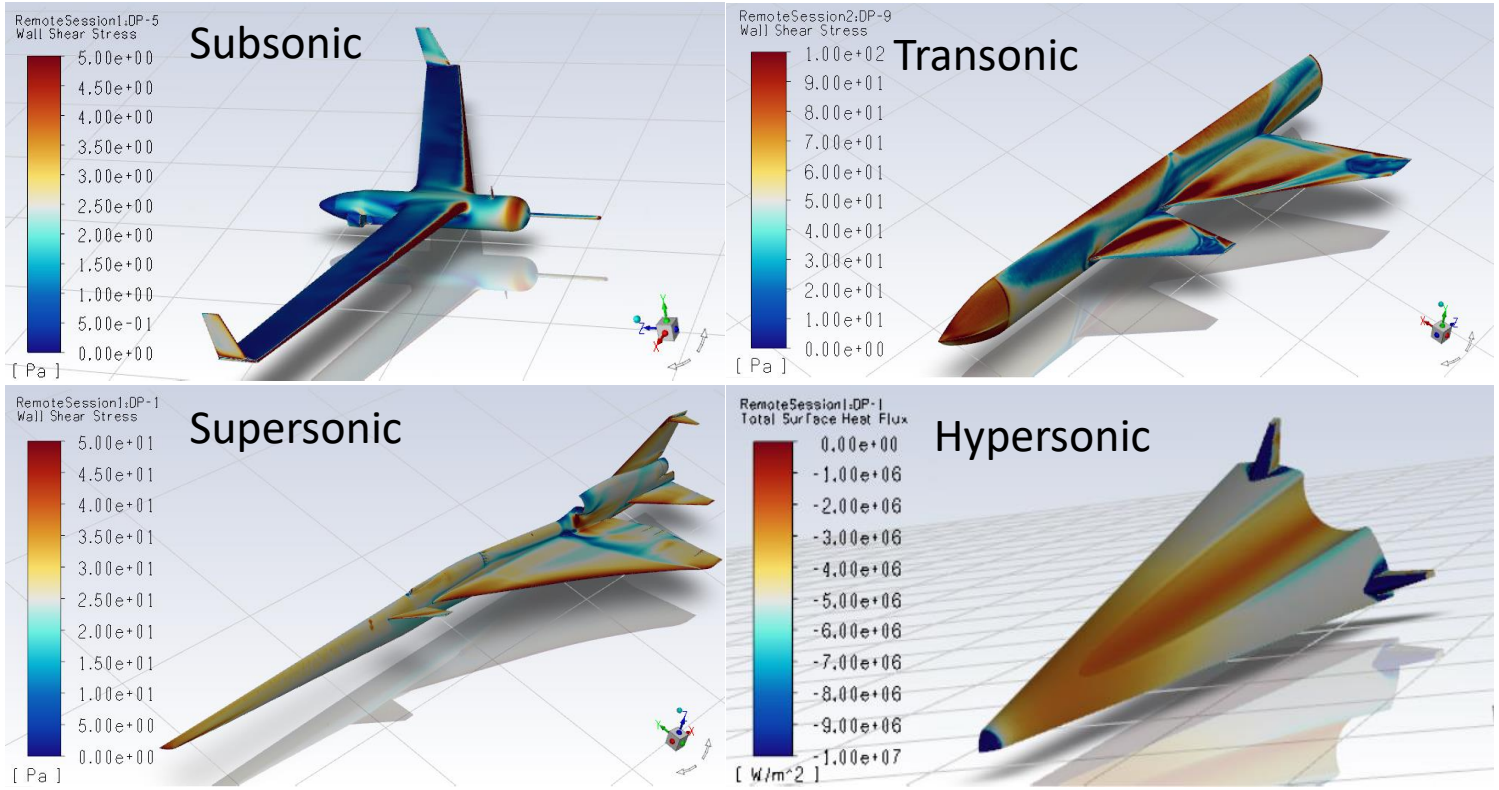




Fluent Aero Workspace

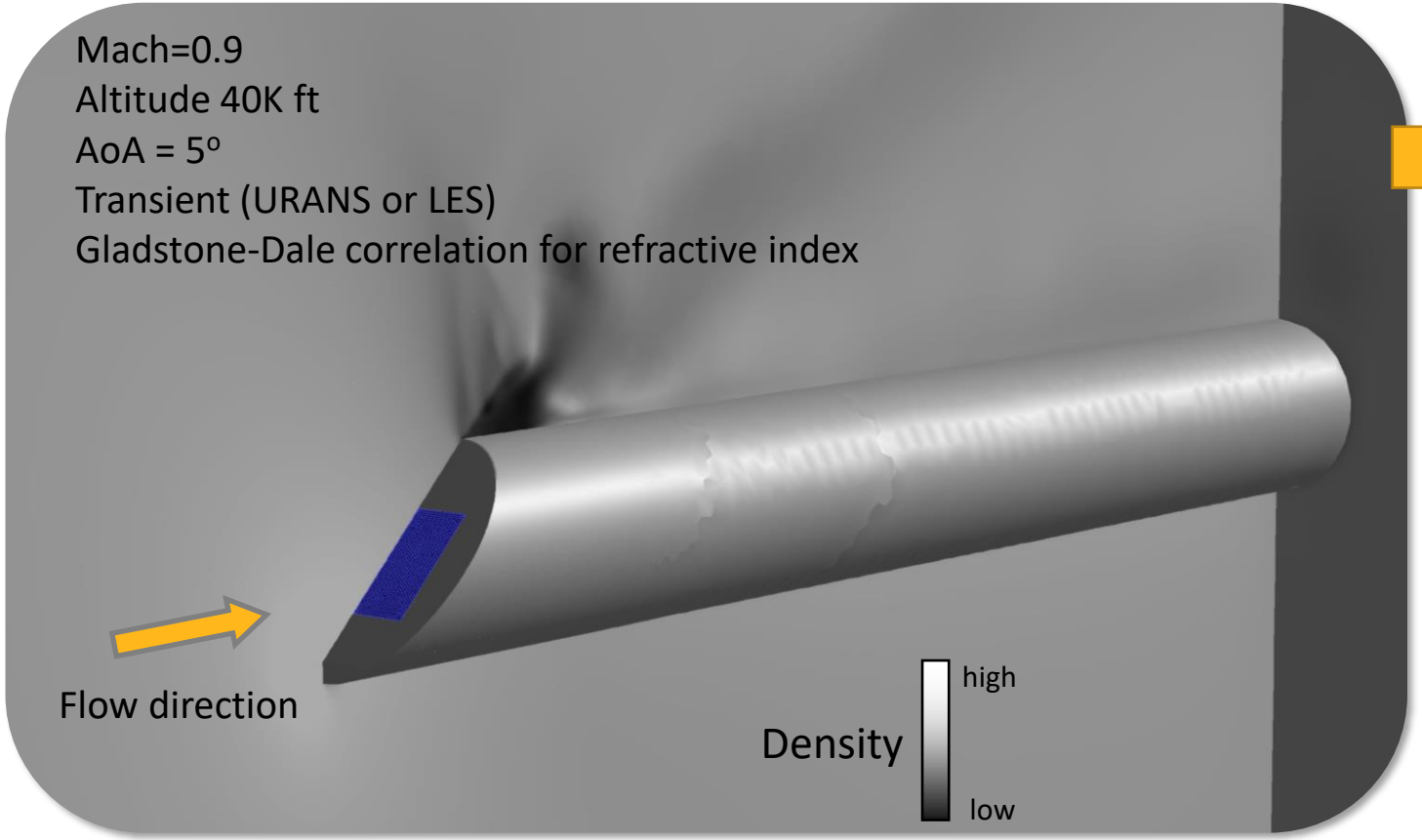
- Automation Enhancements
  - Full batch mode for the entire Design Point Matrix
- Best Practices
  - Recalibration of DBNS default convergence settings for Subsonic to Hypersonic simulations
  - Expose the latest convergence settings for DBNS
- Aerodynamic Extraction Tool
  - Improve robustness of CAD and meshing scripts for airfoils
  - Improve workflow to create the aerodynamic airfoil data for Virtual Blade Model
- Airflow Properties & Physics
  - Wall thermal boundary conditions supported

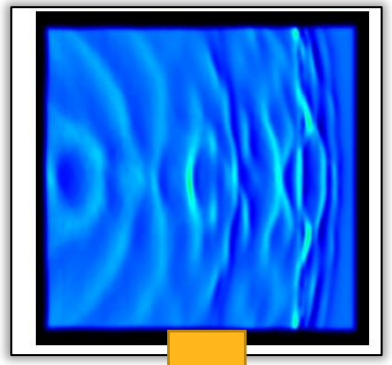
Best Practices Results using Default and Robust Convergence Settings



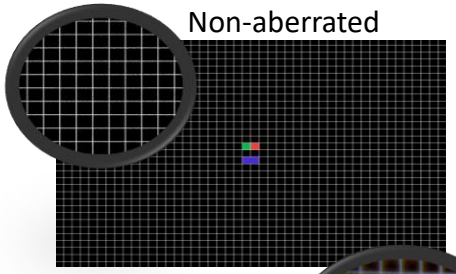
Built-in AeroOptical workflow for computation of optical aberrations (Beta)

Flow field and Optical Path Length / Optical Path Difference (OPL/OPD) computed in Fluent

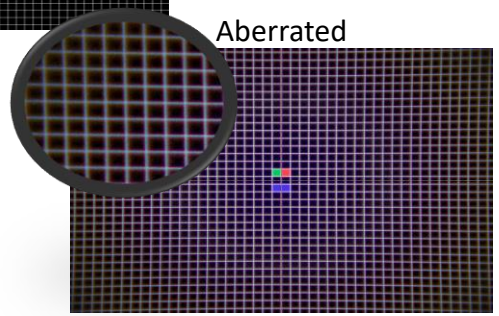




Visualization of OPD on sensor window in Fluent



Export of OPL/OPD from Fluent to OpticStudio for optical system analysis



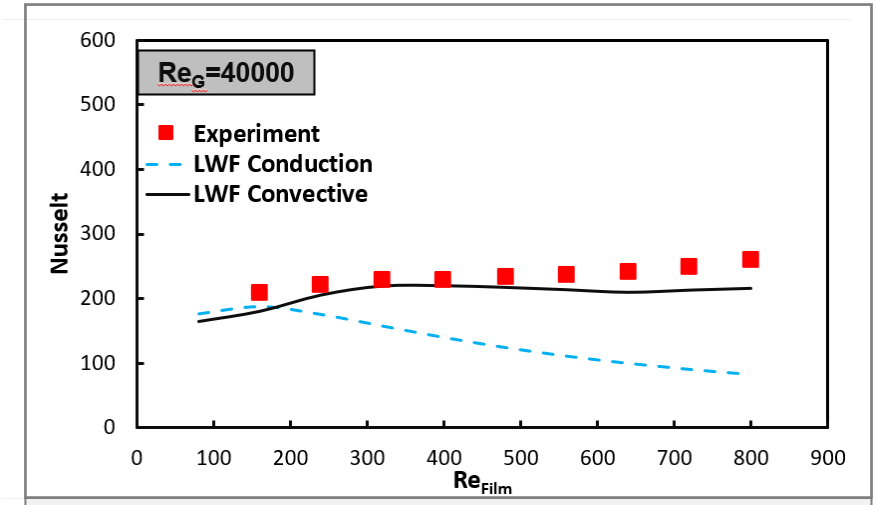
Multiphase

Lagrangian Wall Film: Convective HTC Model

Thin liquid films appear in numerous applications, including pre-filming atomizers, Selective Catalytic Reactors in exhaust systems, and oil cooling systems for automotive electric motors

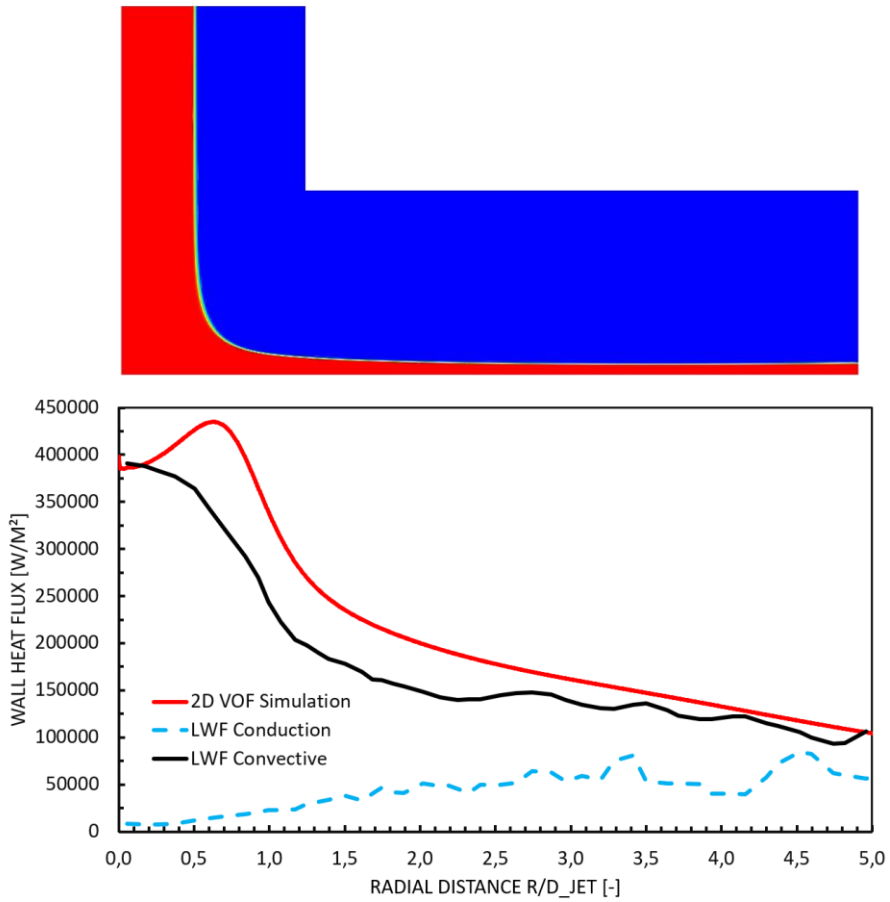
The new Convective Heat Transfer Coefficient model for Lagrangian Wall Film (LWF) modeling accuratley captures wall heat transfer

- Results are comparable to a fully-resolved VOF simulation for an impinging jet
- Excellent comparison to experiments of gravity driven and shear driven film for a large range for Film Reynolds numbers



Experimental comparison for shear driven film

2D VOF Simulation: Impinging Jet

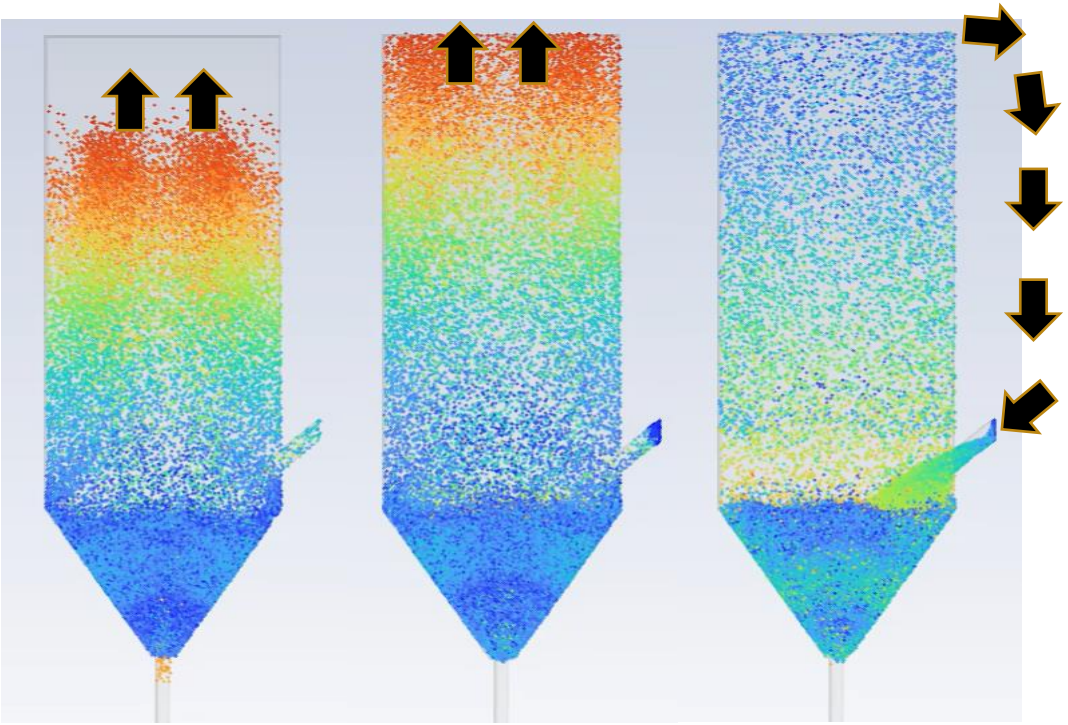


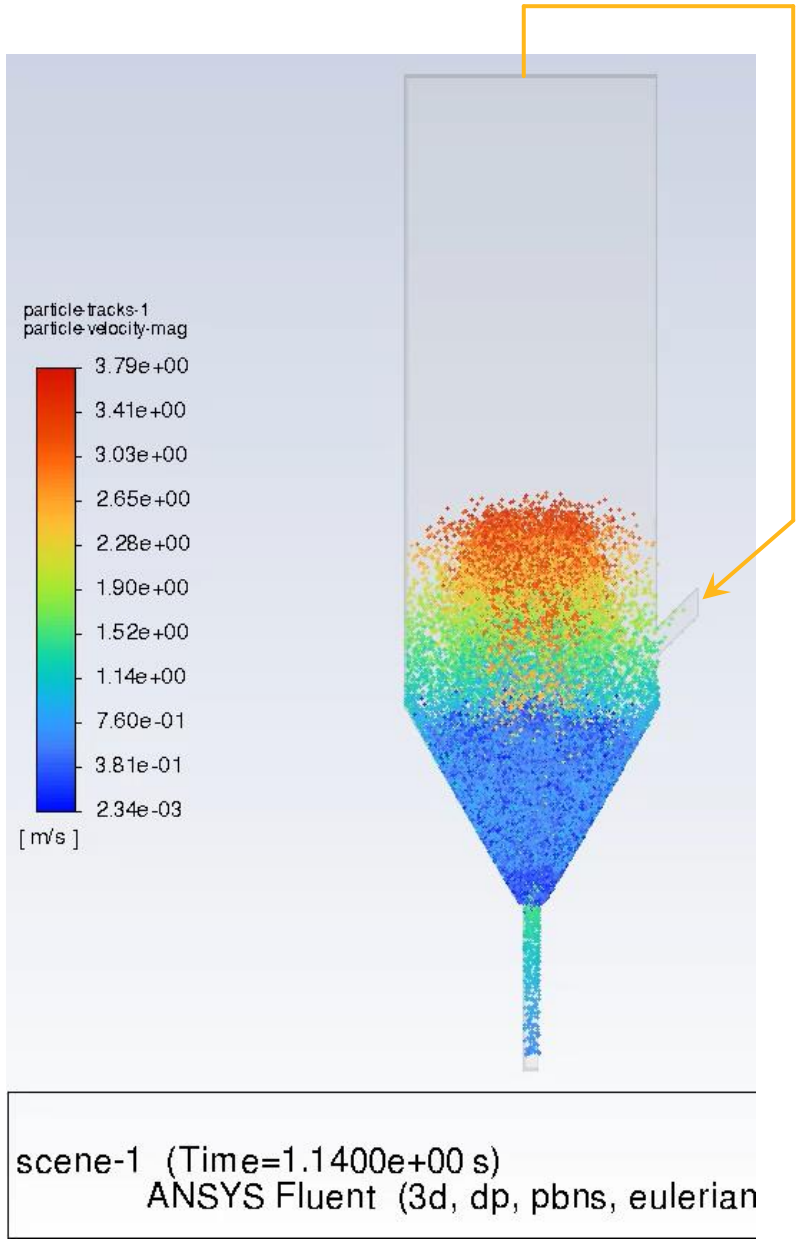
Comparison of new LWF Convective HTC model to fully resolved VOF results and the LWF Conductive model

Gas/Particulate Flows

New Particle Reinject Boundary Condition

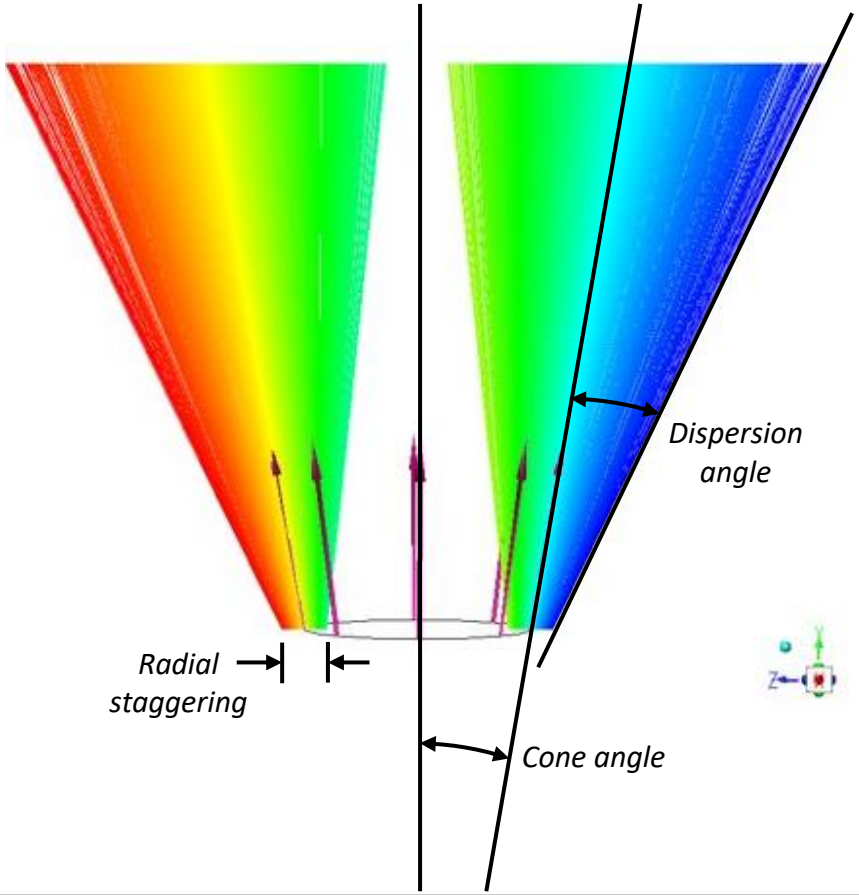
- Particle recirculation boundary condition, allowing particles leaving the domain to be reinjected again
- Particle temperature, diameter and composition are preserved when leaving the domain, but location and velocity are specified for reinjection
- Applications: carbon capture, proppant transport, fluidized bed reactors





New options for Hollow Cone Injection with DPM

More accurately model hollow cone particle injection

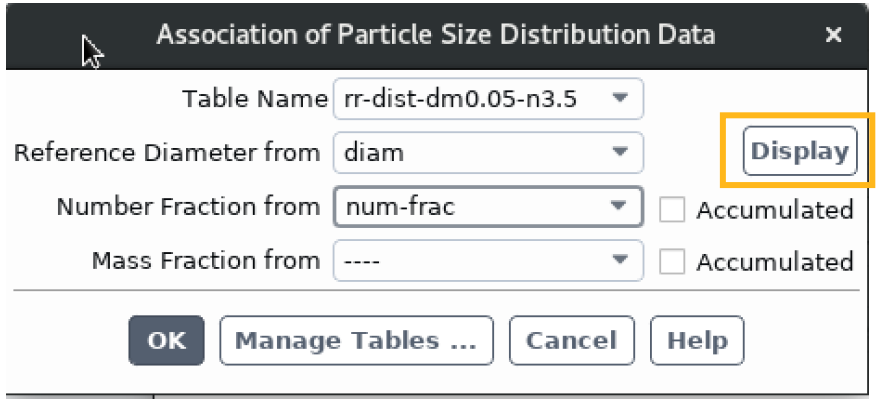


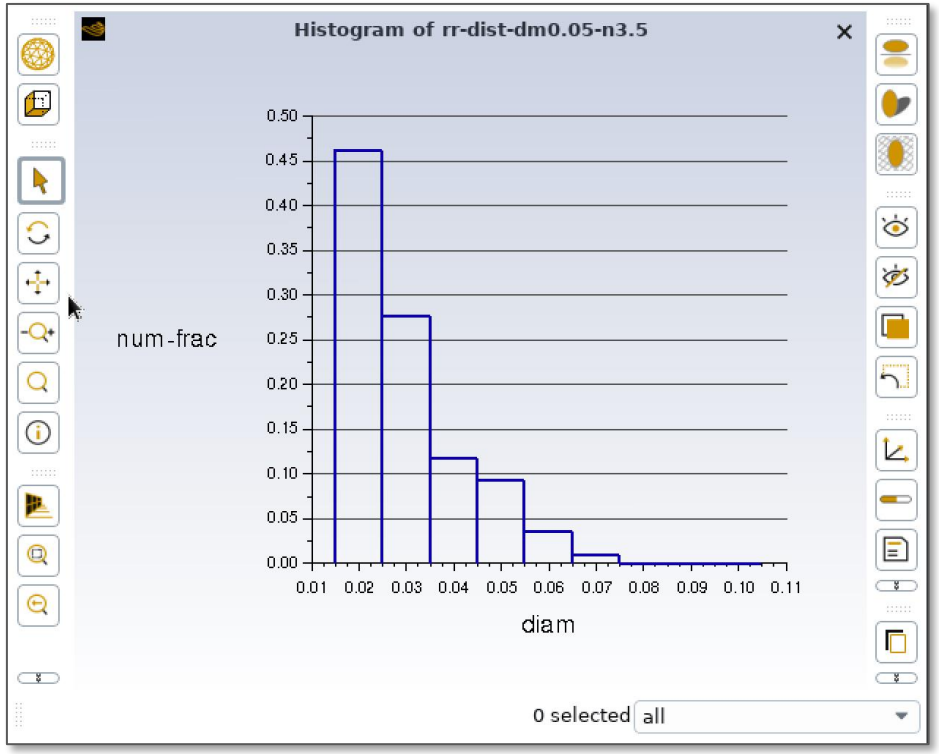
- A Dispersion Angle can now be set, in addition to the Cone Angle
  - The Cone Angle is interpreted as the mean angle
  - The Dispersion Angle is added/subtracted to the mean angle to define the cone shape
- A spread can be set in the initial radial injection position (radial staggering)
  - "*Stagger in Injection Plane only*" option in the UI

Discrete Phase Modeling (DPM) Usability Improvements

Histogram plots for tabulated size distribution of particles

- The Reference Diameter is plotted over Mass Fraction or Number Fraction





DPM summary report now includes zone names:

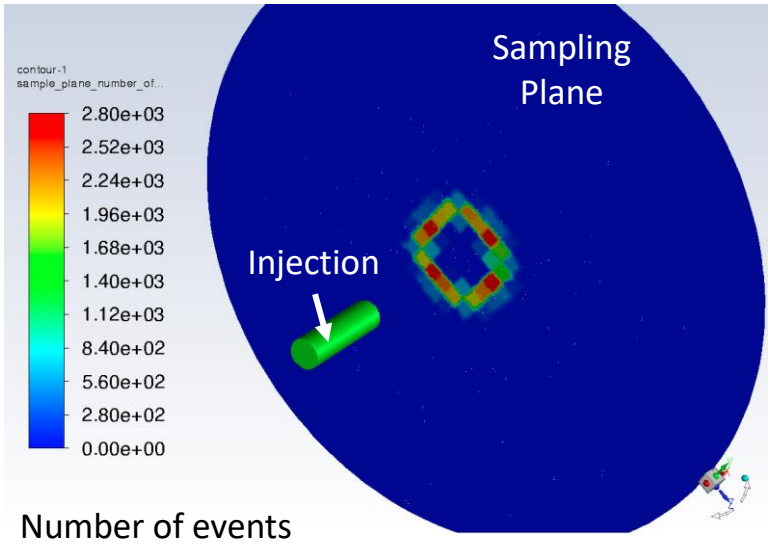
(*) - Multicomponent Droplet - (*)

Fate	Zone Name	Zone Id	Species Names	Species Content (kg/s) Initial	Final	%Conv
Evaporated			c5h12-droplet<1>	1.667e-002	0.000e+000	100.00
Evaporated			c7h16-droplet<1>	3.333e-002	0.000e+000	100.00
Evaporated			h2o<1>	0.000e+000	0.000e+000	0.00
Escaped	inlet	2	c5h12-droplet<1>	1.667e-002	2.585e-004	98.45
Escaped	outlet-low	4	c7h16-droplet<1>	0.000e+000	0.000e+000	0.00
Escaped	outlet-high	6	h20<1>	3.333e-002	1.134e-002	65.99
Trapped	wall-top	3	c5h12-droplet<1>	3.333e-002	0.000e+000	100.00
Trapped	wall-bot	5	c7h16-droplet<1>	3.333e-002	0.000e+000	100.00
Trapped	wall-left	7	h20<1>	3.333e-002	0.000e+000	100.00

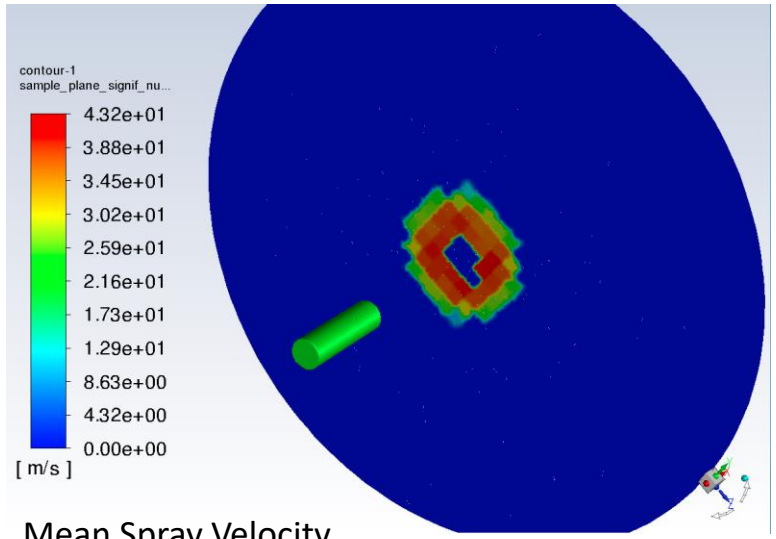
Contour Plots on Sampling Planes (beta)

Generate contour plots for sampled particle data

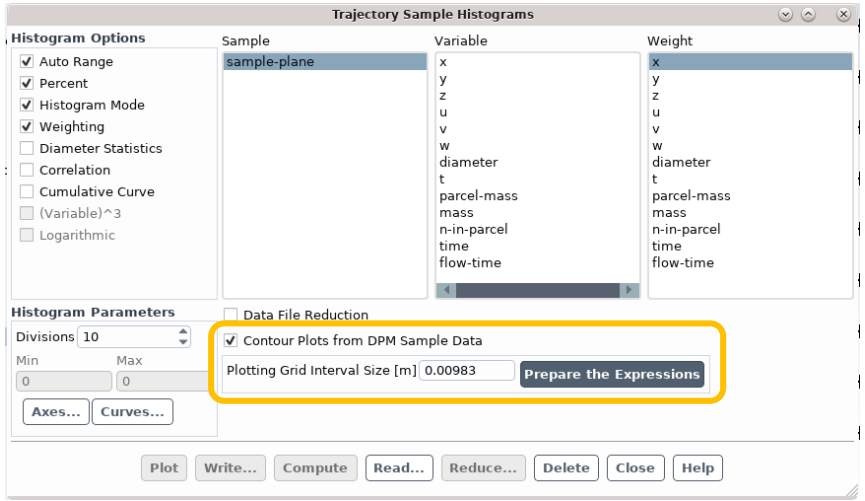
- Sampling takes place on a user defined plane
- The sampling plane is discretized into a 2D mesh
- Particle data recorded on the sampling plane is assigned to a cell of the mesh and “averaged”
  - Number weighting, mass weighting
- Allows mass per unit area

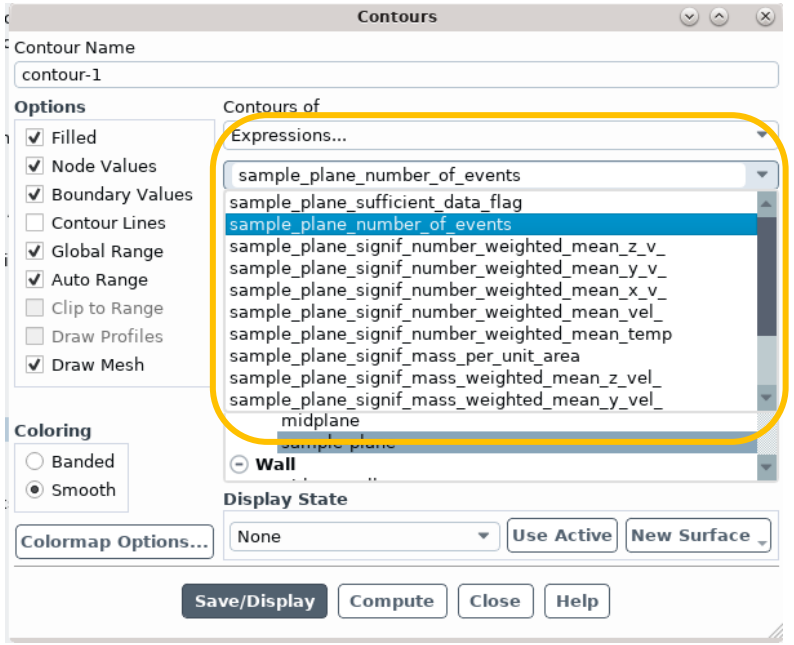


Number of events



Mean Spray Velocity



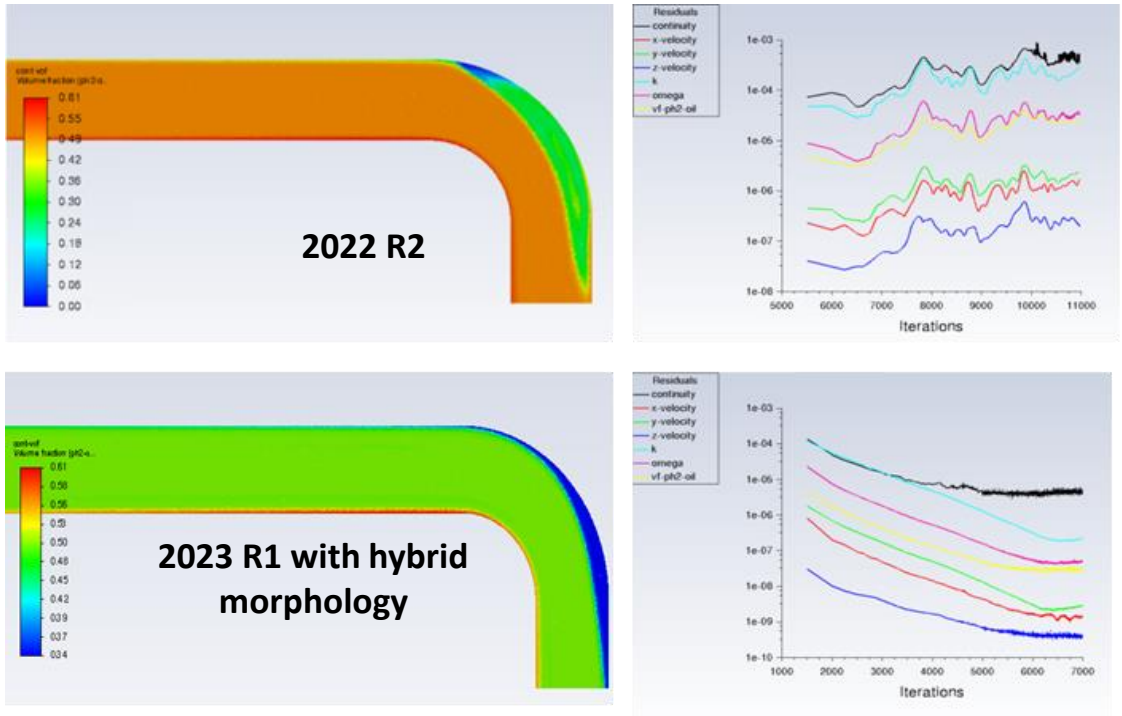


Eulerian Multiphase

Regime transition within the Mixture Model

- The Eulerian AIAD model provides a generalized approach to multiphase flows across flow regimes/morphologies, but the simpler Mixture Model is faster
- 2023 R1 introduces regime transition within the Mixture Model, using the AIAD framework to identify the flow morphology (hybrid morphology) and automatically apply an appropriate drag correlation
- Provides faster solutions for gear-box applications, separators and other mixed-morphology multiphase flow applications

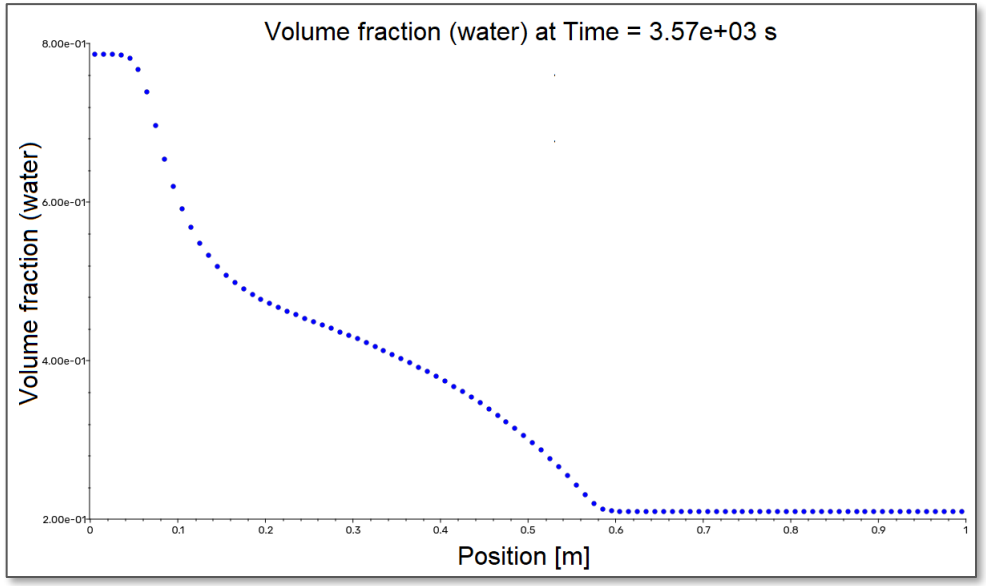
Gravity-driven Oil-Gas Separator with the Mixed Model



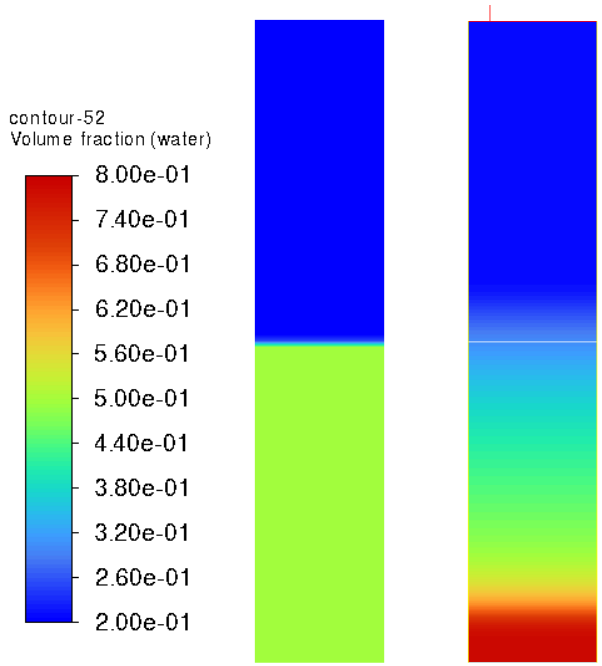
Eulerian Multiphase

More robust Capillary Pressure formulation for porous media applications

- Capillary Pressure has been reformulated for better stability and convergence
- Consistent sources for overall continuity and phasic energy



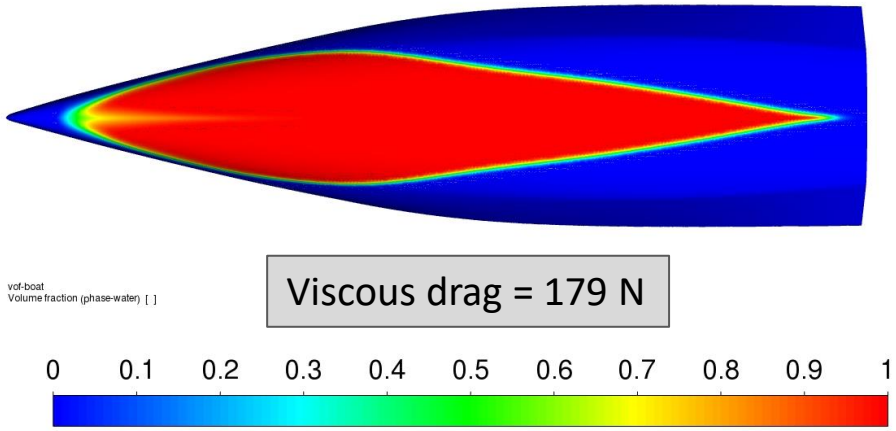
Capillary Pressure / Gravity Equilibrium



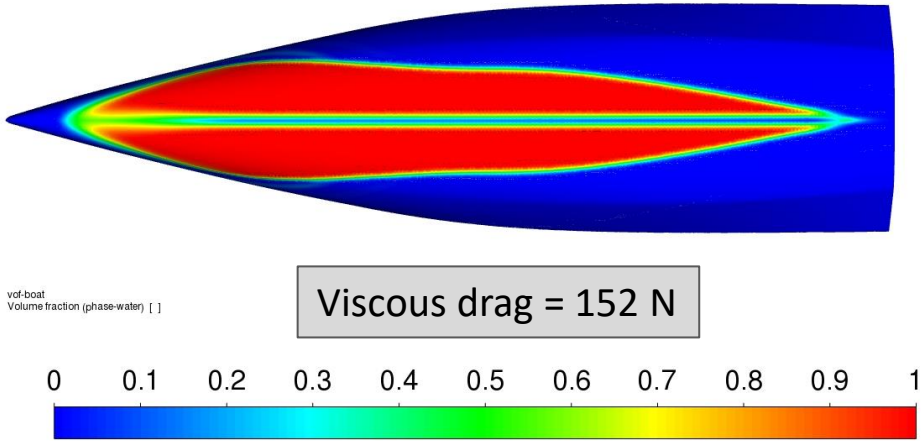
Initially condition (left) shows water at 0.5 volume fraction in the bottom half of the column. Gravity pulls the water down, while capillary pressure acts in the opposite direction, until an equilibrium is reached (right)

Minimizing Numerical Ventilation in Planing Hulls (β)

- Numerical ventilation is the unphysical entrainment of air under a hull, and is a major source of error in planing hull simulations
- New expression-based treatment removes air or water in the first cell of the boundary layer using a sink term which depends on the local volume fraction
- Predicted drag is much close to experiments
- Could be beneficial on other applications as well like filling, electric motor cooling, e-coating etc.



Sink Term Numerical Treatment

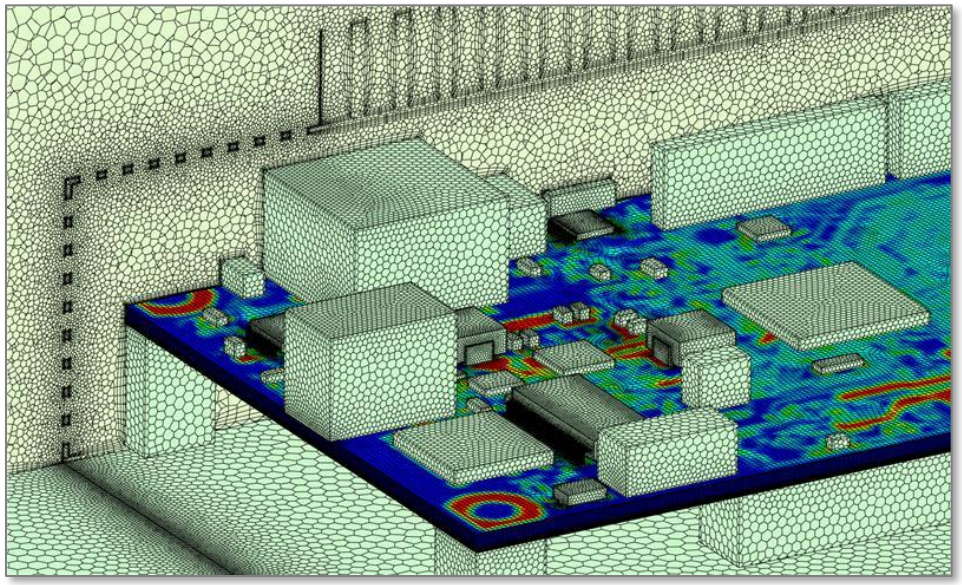


Default Compressive Scheme without Sink Term Numerical Treatment

Solver & Physics

PCB direct ECAD workflow

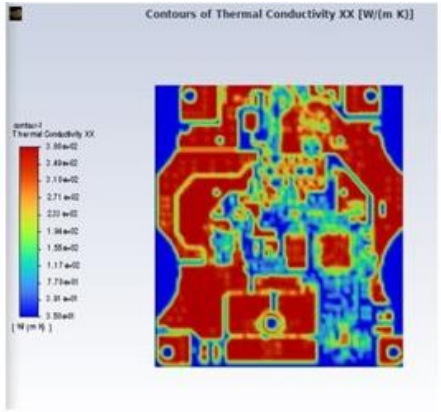
Fluent Meshing with Multizone meshing for PCB



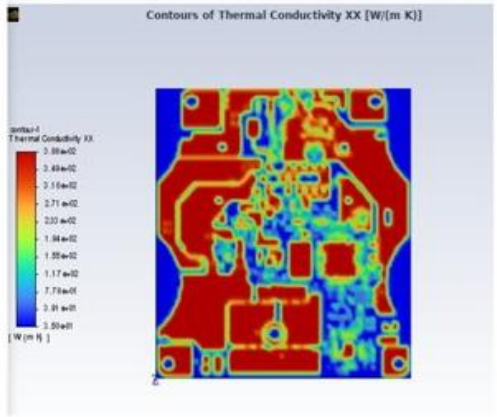
From ECAD -> Fluent Meshing -> Fluent solver

- Detailed geometry representation is needed to resolve thermal fields in products having complex geometries (PCB + complex surrounding assembly).
  - Support of Multizone meshing for PCB bodies.
  - Read ECAD Power Map data to capture accurate PCB thermal behavior in Fluent
- Include Wall Film modeling for condensation on display instruments, fogging/defogging, and evaporation modeling for embedded apps
- Capability to extend to system simulation via ROM creation and Ansys Twin Builder coupling.

Validation with IcePak



K Distribution using IcePak Generated Input Files



K Distribution using PCB Workflow Generated Input Files

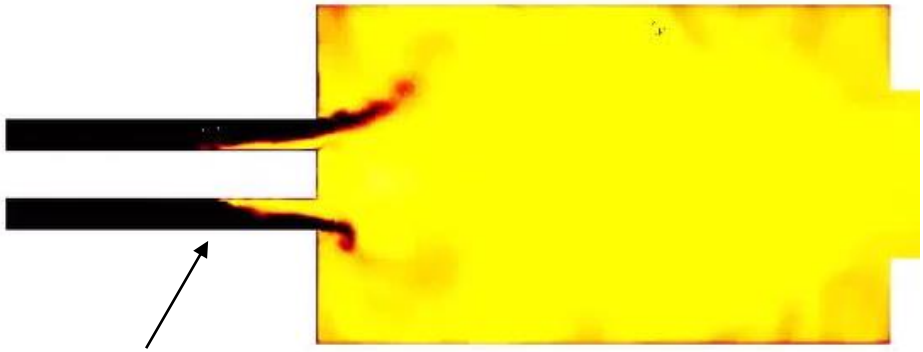
Hydrogen Combustion Flashback

Validation cases have established new best practices for hydrogen combustion flashback modeling

- The Thickened Flame Model (TFM) with Automatic Mesh Refinement (AMR) is recommend for H2 flashback

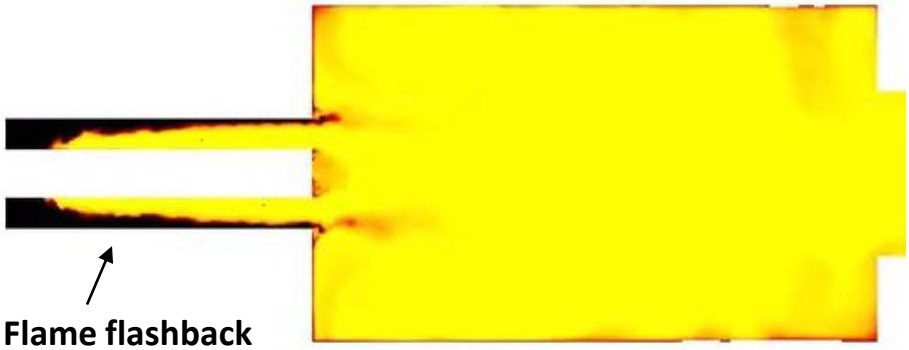
Note: for lifted lean H2 flames and for stable combustion (swirler or bluff body), FGM is recommended

65% H2



Sustained flame anchored

80% H2



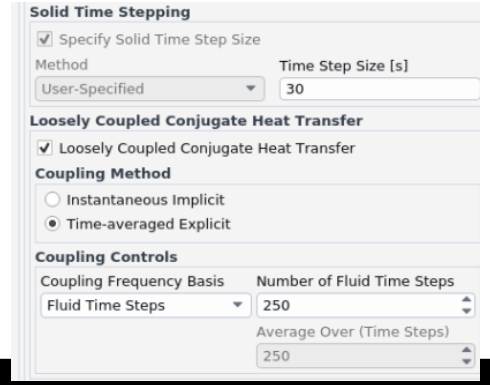
Flame flashback

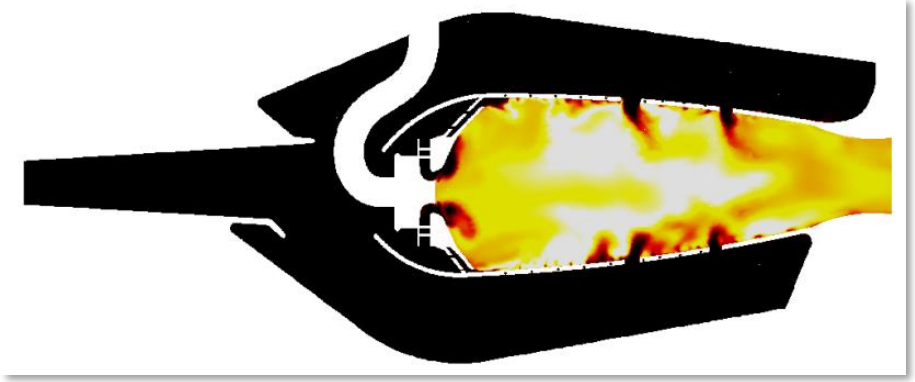
Flashback is correctly captured

Gas Turbine Linear Modeling with MDA

Fluent's Multi-Domain Architecture (MDA) provides a flexible and efficient approach for gas turbine liner temperature assessment

- Support <u>conformal and non-conformal</u> interfaces between fluid and solid zones
- Solid zone is only solved at a few time instants, increasing simulation speed
  - Loosely coupled approach: periodically solve fluid and solid fully coupled, solving only the fluid zone in the intermediate time steps
- Time-averaged temperature passed from fluid to CHT solid

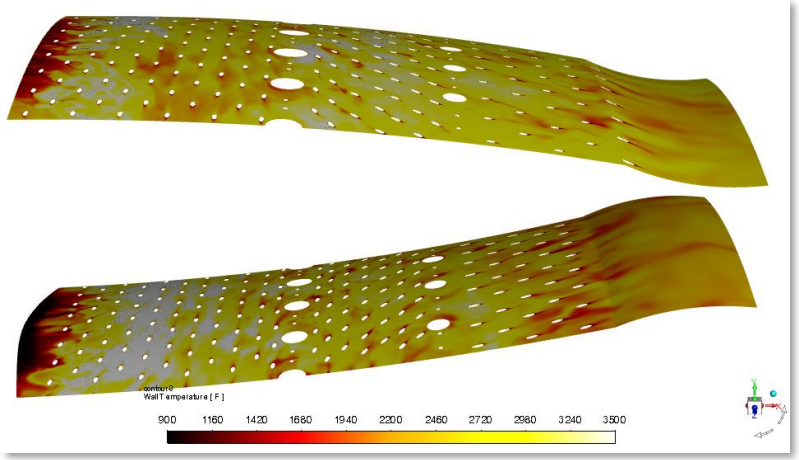




Transient flow field, with time-averaged fluid temperatures stored at the solid interface

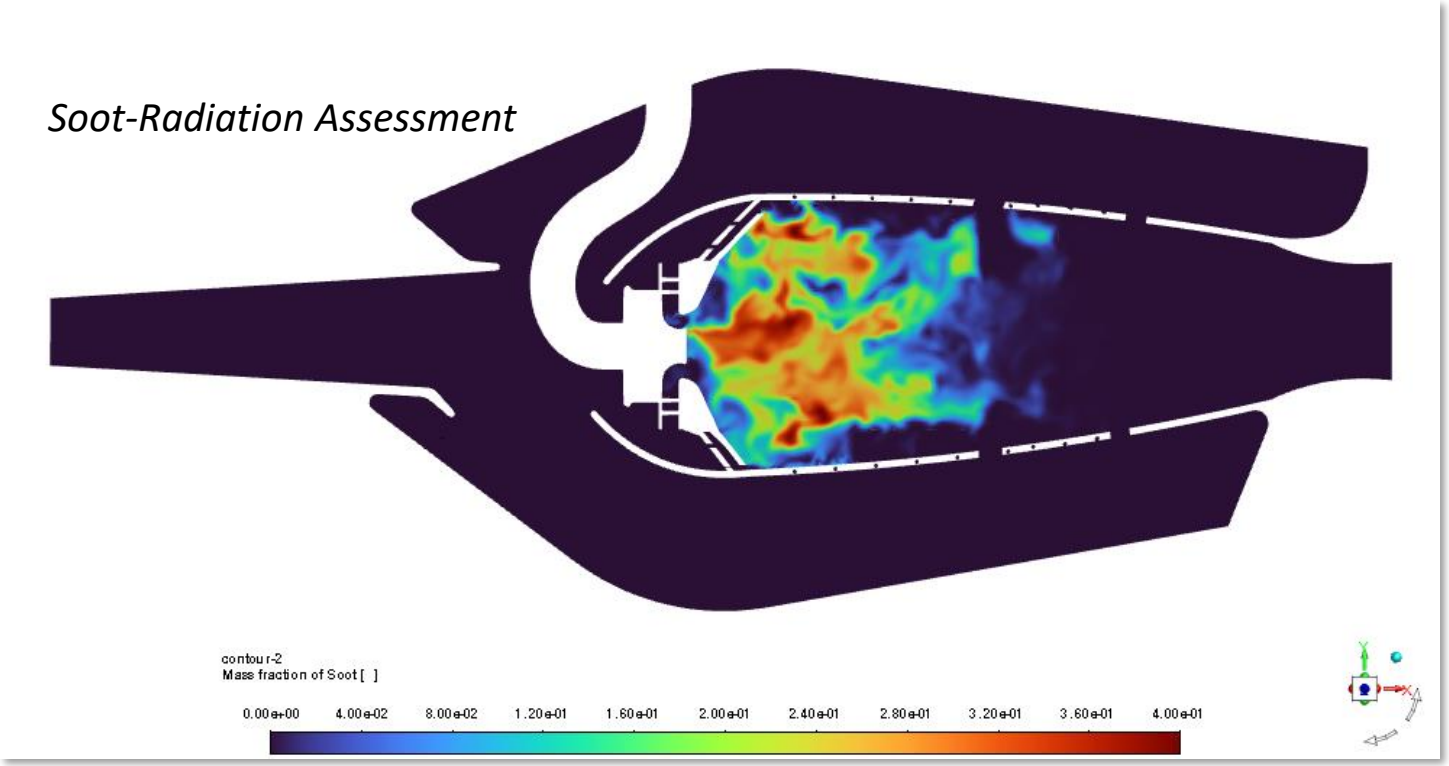
Solid temperature solved periodically, using a large time step to accelerate convergence

Volumetric liner temperature or surface temperature and HTC can be passed to Mechanical for durability analysis



Gas Turbine Combustion

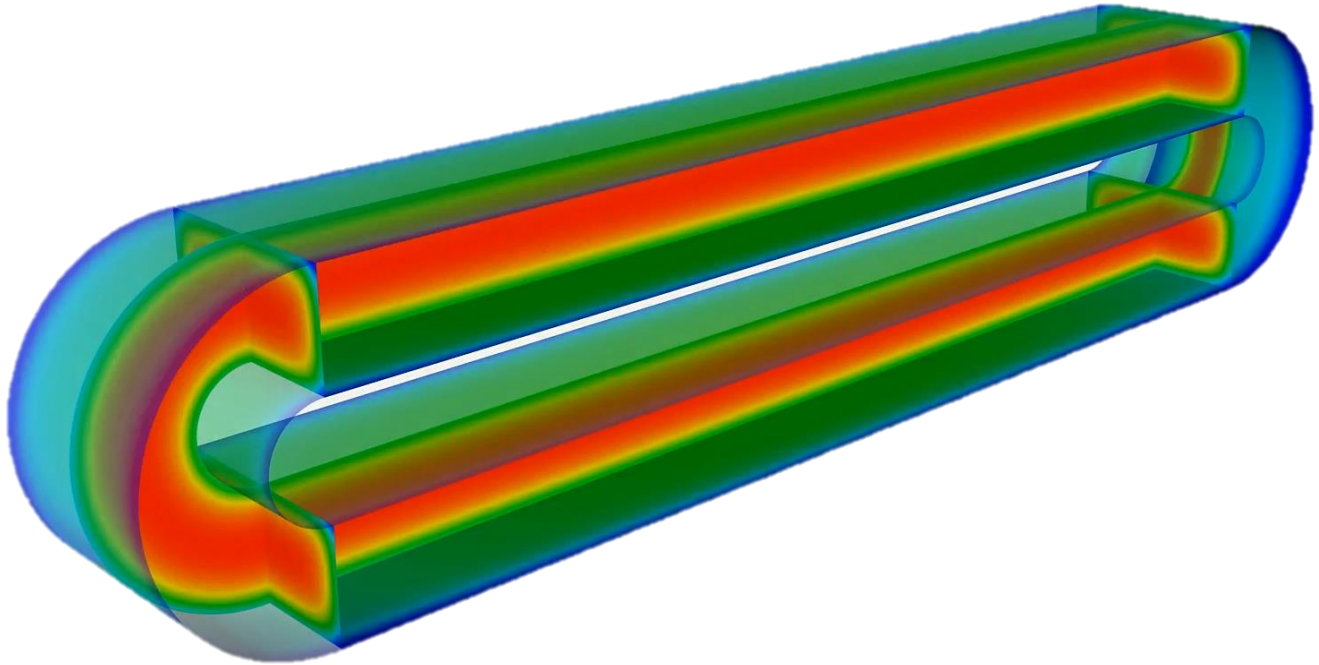
- Discrete Ordinates (DO) radiation acceleration is now compatible with soot and particle interactions
  - Allows for faster radiation calculations when using soot or particle tracking



Heat Transfer

Improved accuracy for thermal modelling of Electric Motors

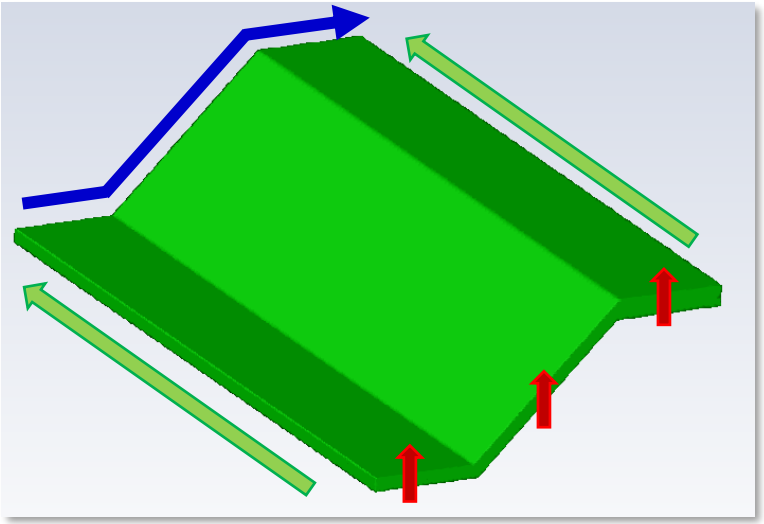
- Anisotropic thermal conductivity numerics improvement
  - New default blended flux improves accuracy and robustness
  - Local neighbor gradient option further improves convergence and accuracy, and is 2-3 time faster (β)

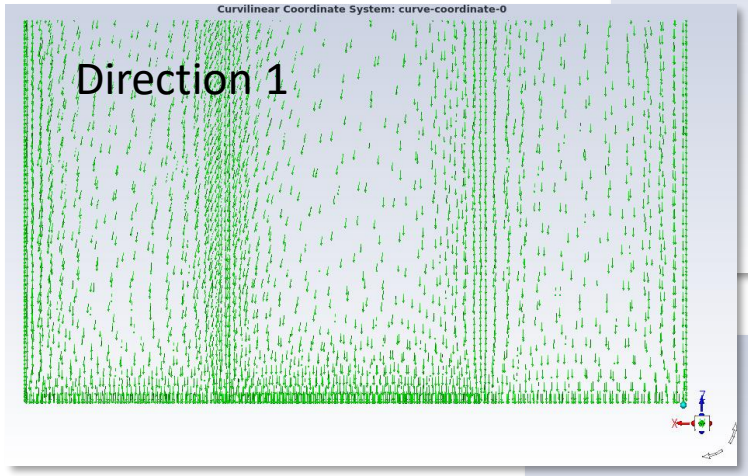


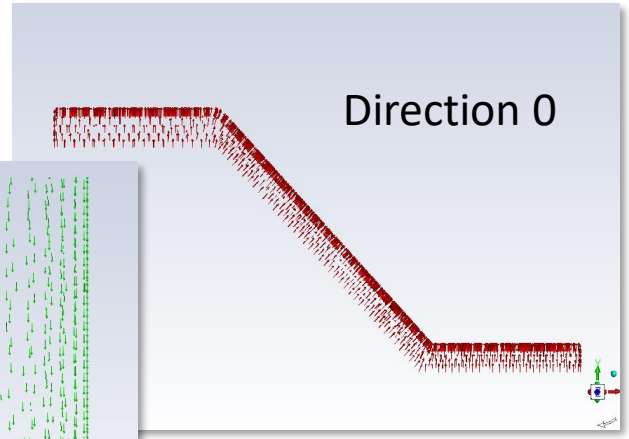
Temperature in E-Motor Winding with Anisotropic Thermal Conductivity

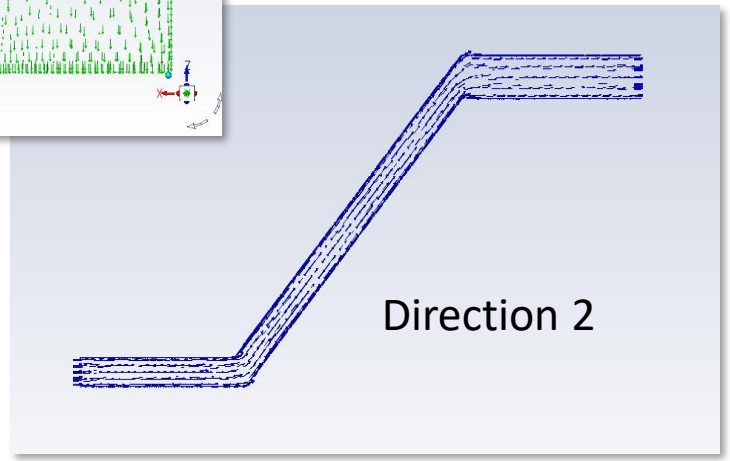
Porous Modeling with Curvilinear Coordinates

Model complex porous shapes (e.g. filters) with anisotropic resistances using Curvilinear Coordinate Systems (CCS)







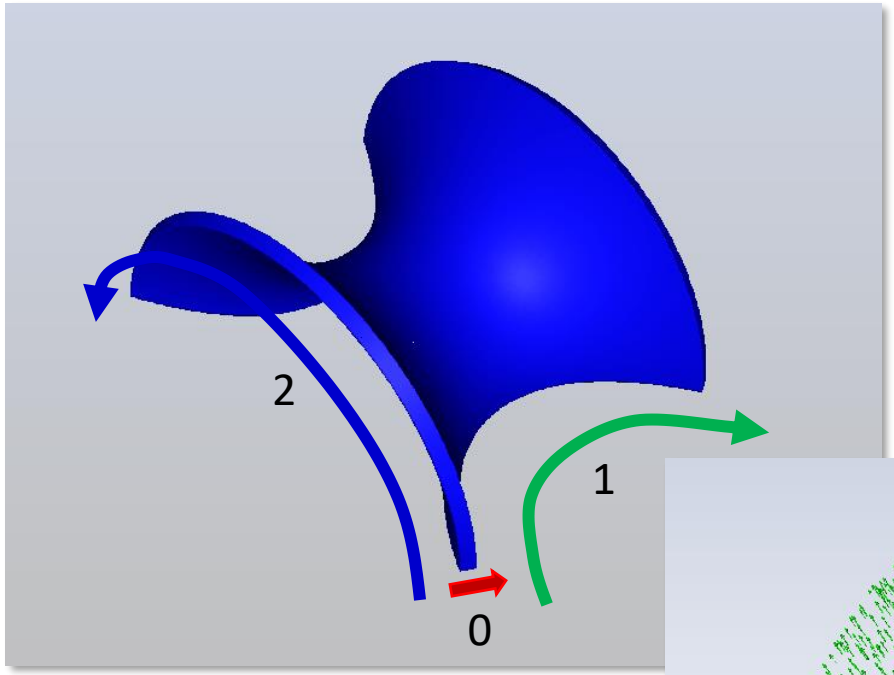


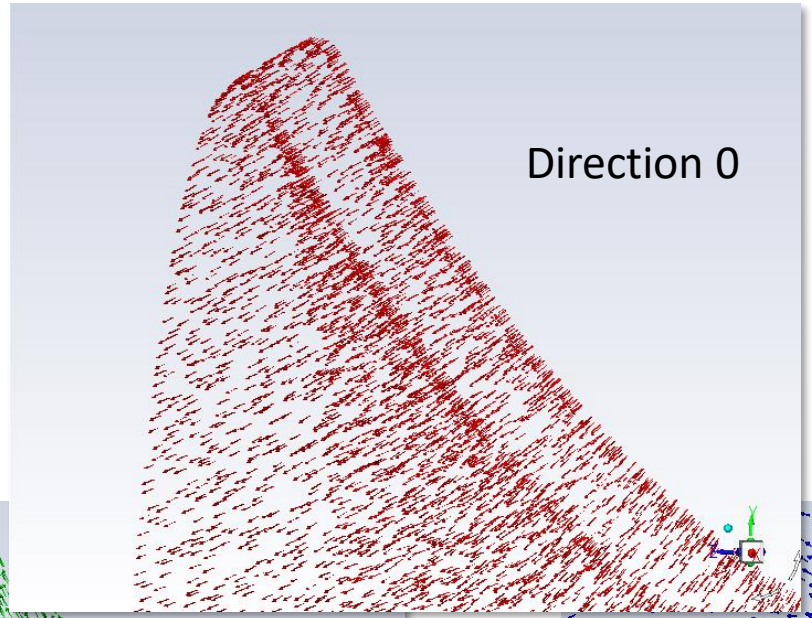
Direction 0: select start and end faces to define the primary direction

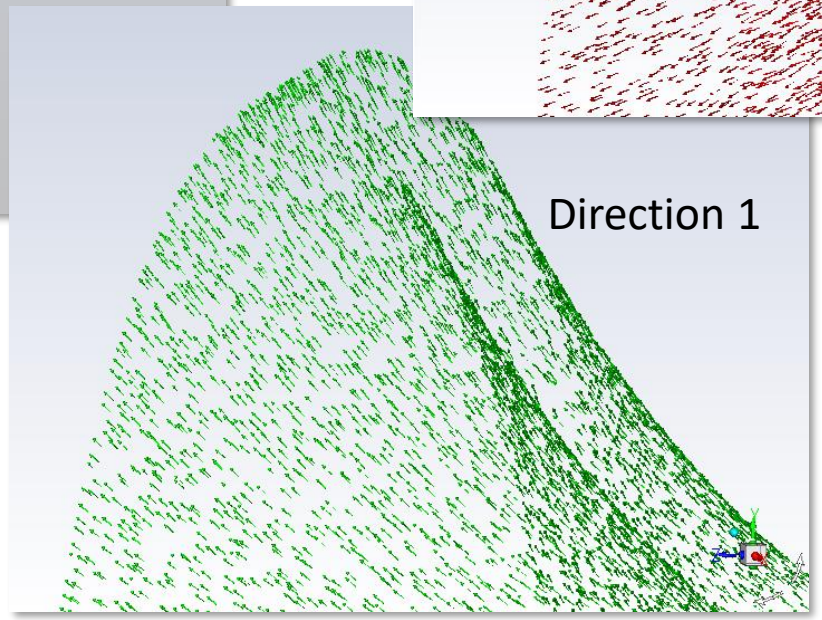
Direction 1: select start and end faces to define the secondary direction

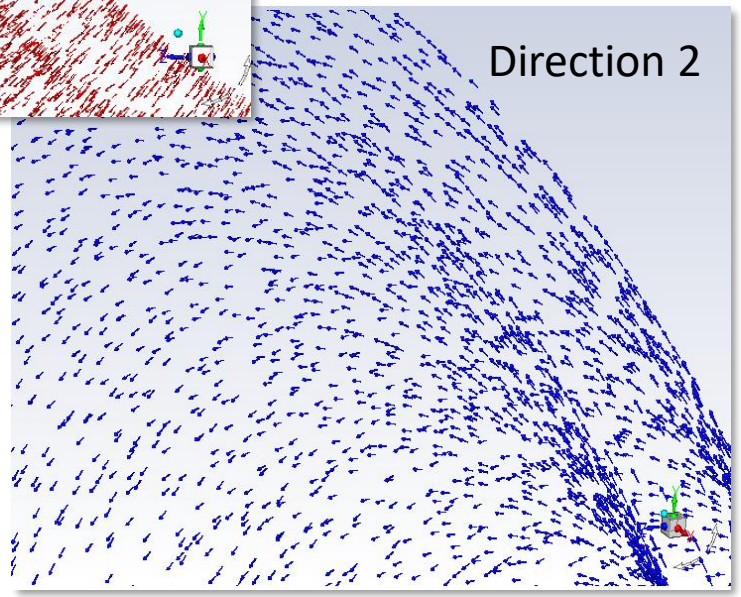
Direction 2: the third direction is calculated using CCS

Porous Modeling with Curvilinear Coordinates





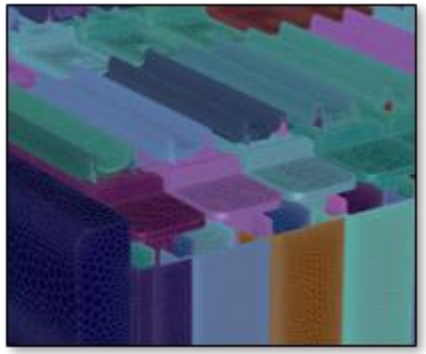


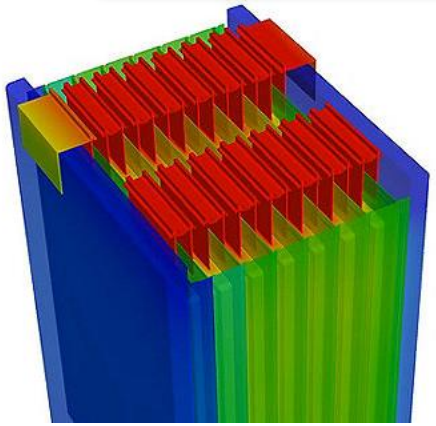


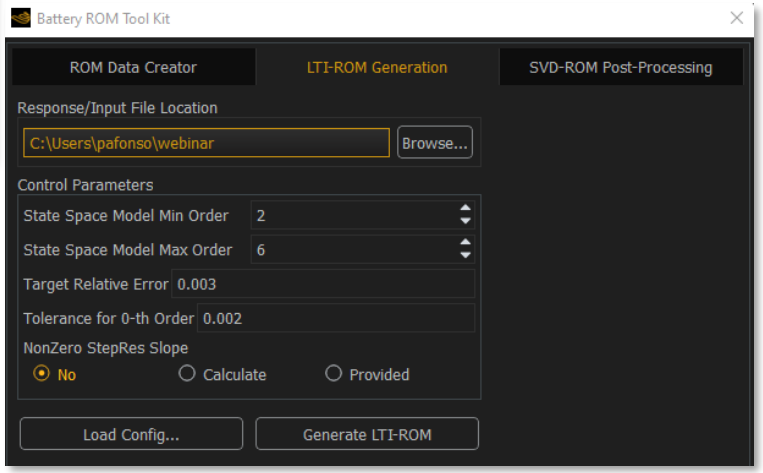
Battery Modeling

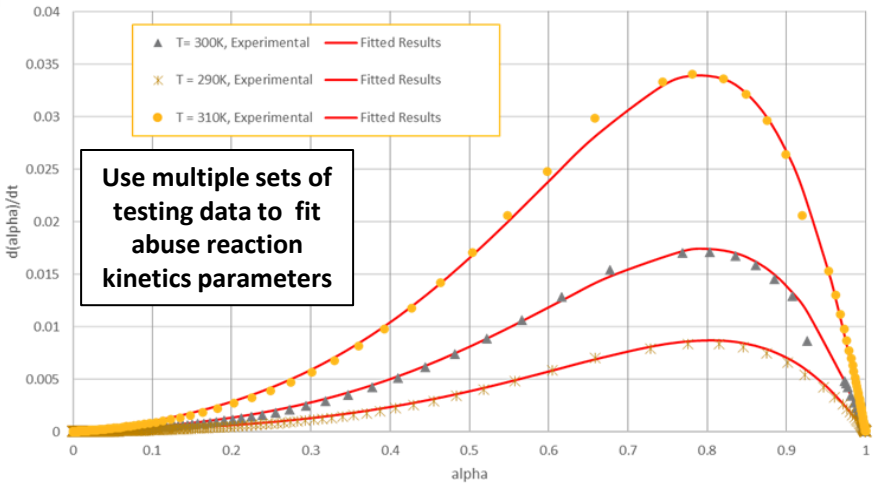
Comprehensive improvements on battery design workflow & functionalities

- Generalized pack-builder pattern
  - Improved shell conduction model support
- **LTI-ROM generation in Fluent**
- **Parameter Estimation Tool improvements**
  - Improved workflow for both charging and discharging curves
  - **Use multiple testing curves in fitting of abuse reaction kinetics**
- Extended internal short propagation model to the 1-eqn thermal abuse model
- Automatic V, I and T_max monitoring in battery simulation
- Provide cycle history profile for physics-based life model









Use multiple sets of testing data to fit abuse reaction kinetics parameters

Green Hydrogen Production and Fuel Cells

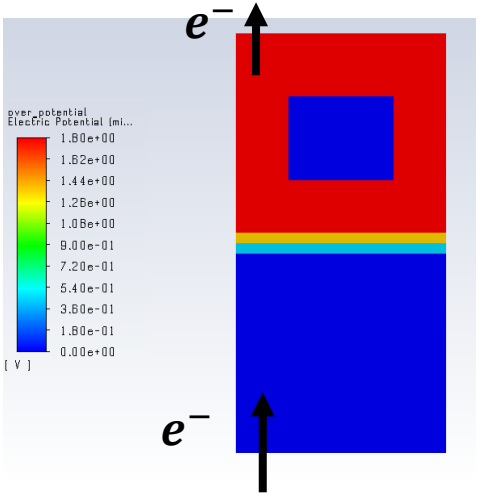
Simulate electrolysis for green hydrogen production

- 2023 R1 supports electrolysis with the Proton Exchange Membrane (PEM) model and the Alkaline Electrolysis model
  - Multiphase modeling integrated with electrochemistry solver
  - Modeling of porous electrodes in PEM or Alkaline electrolyzer

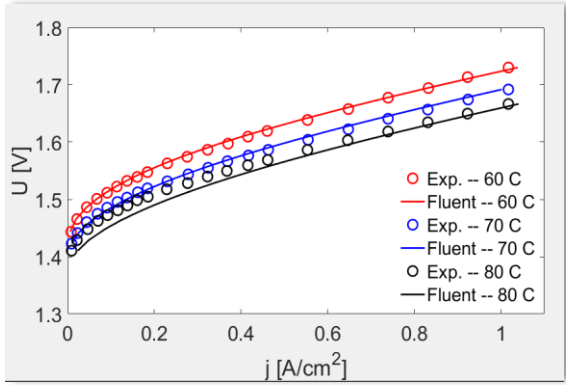
Extensions to Fuel Cell modelling

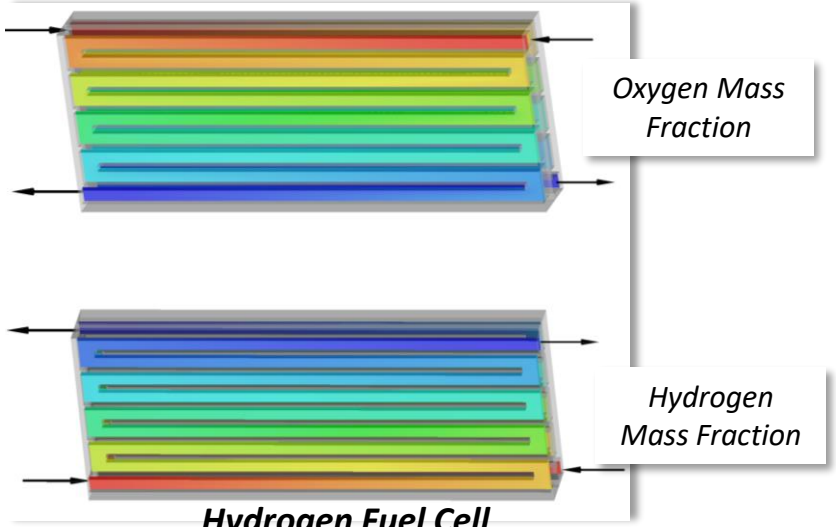
- Knudsen Diffusion for PEMFC in addition to SOFC
- Temperature-dependent exchange current density for SOFC
- Easier scripting by supporting thread names in setting up tortuosity, contact resistance, etc.

PEM Electrolysis



I-V curves at three temperatures



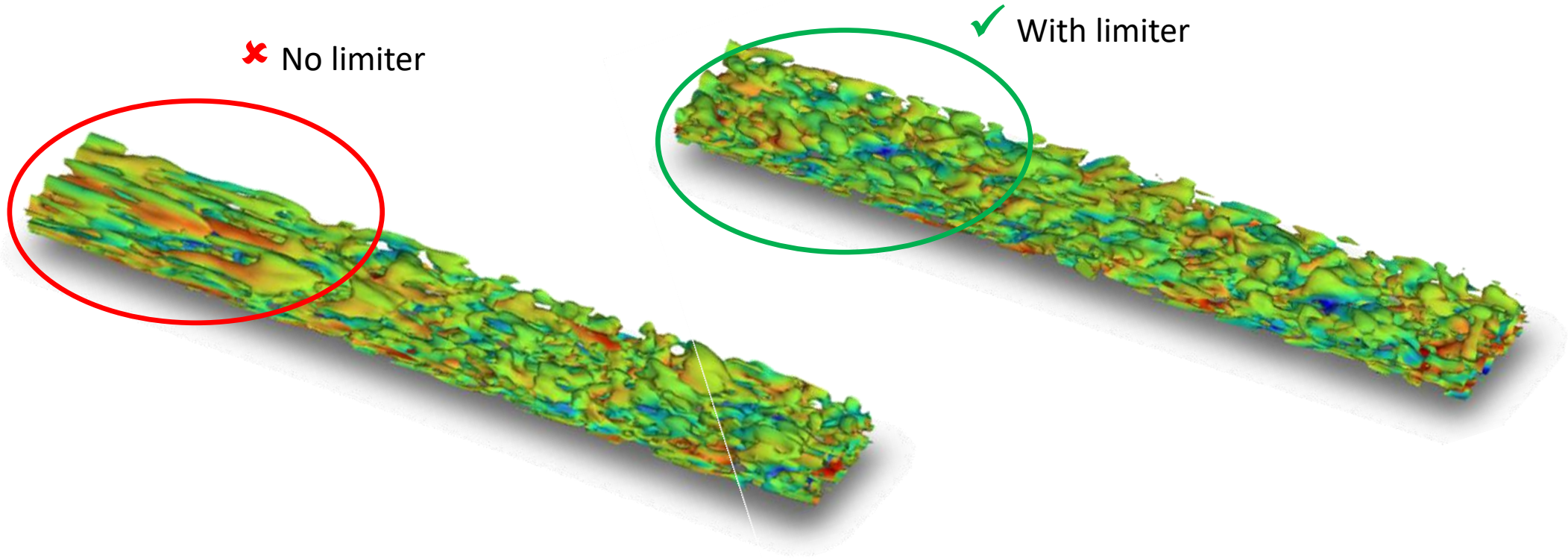


Hydrogen Fuel Cell

Turbulence

Synthetic turbulence generator

- New limiters prevent free stream affecting the calculated turbulence scales

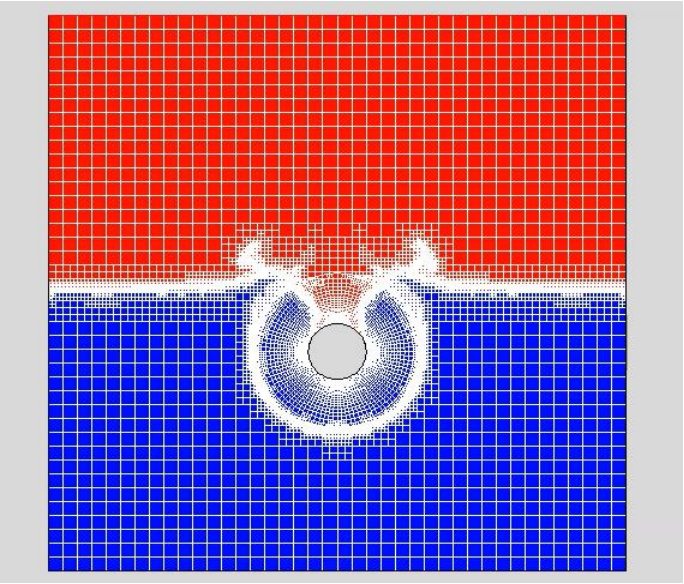


Meshing Adaption and Remeshing

Extended adaption capabilities to Overset meshes & support for Poly Remeshing

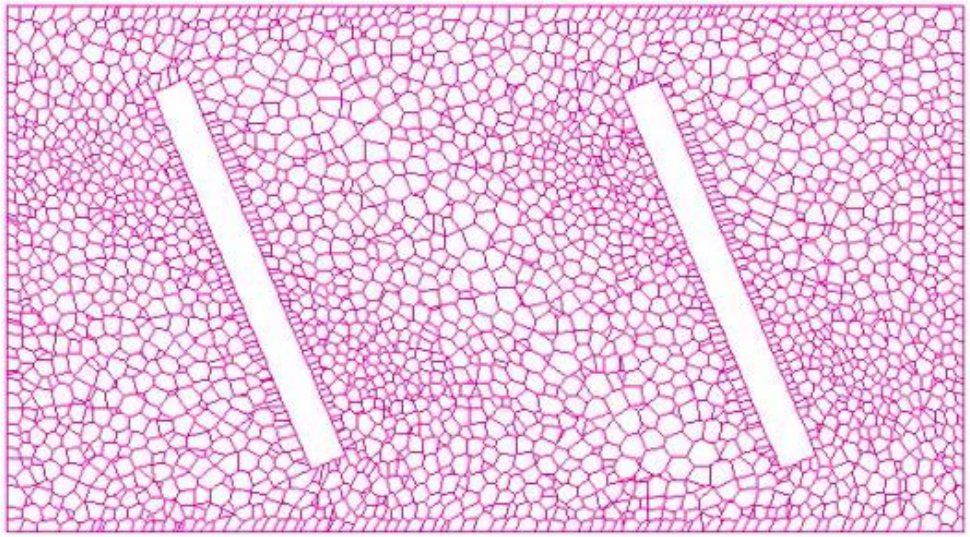
Mesh Adaption

- Consistent normal prism refinement
- Anisotropic refinement respects mesh quality limits
- **Overset adaption as pre-defined criterion**
  - Including gap and anisotropic adaption



Remeshing

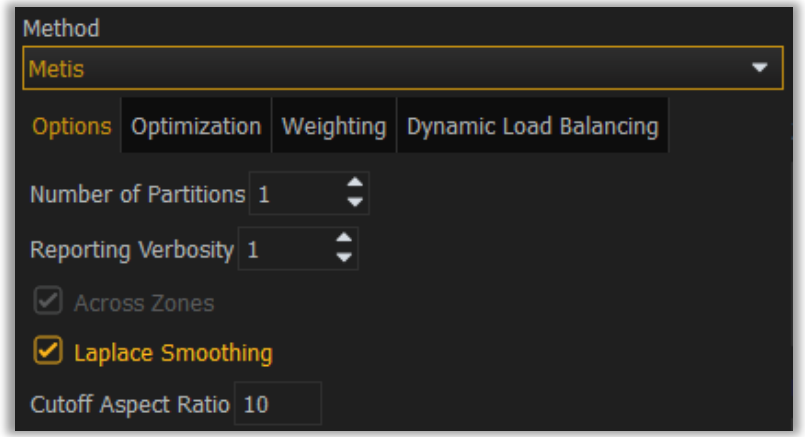
- Prism and size controls are now imported from Fluent Meshing
- Polyhedra remeshing (beta)



Mixing blades with polyhedra remeshing

Additional Solver Performance, Physics and Numerics Improvements

- **Laplace Smoothing is now on by default for mesh partitioning**
  - Reduces convergence dependencies on the number of cores used and the platform used
  - Reduces repartitioning memory usage
  - When opening cases from prior releases, Laplace Smoothing will be used once you have saved your case in 2023 R1
- **Enhanced skewness correction**
  - Improved robustness for the skewness correction option
- **The Floating Operating Pressure options now supports real gases**
  - Removes the restrictive acoustic timescale limit in closed domains at low Mach number
- **Local time stepping**
  - Convergence Acceleration for Stretched Meshes (CASM) is now available with the coupled pressure-based solver (beta)
- **Symmetry boundary conditions are now supported for Shell Conduction**
  - Gives accurate temperature fields when radiation is included
- **The Monte Carlo radiation model now supports non-conformal interfaces (β)**

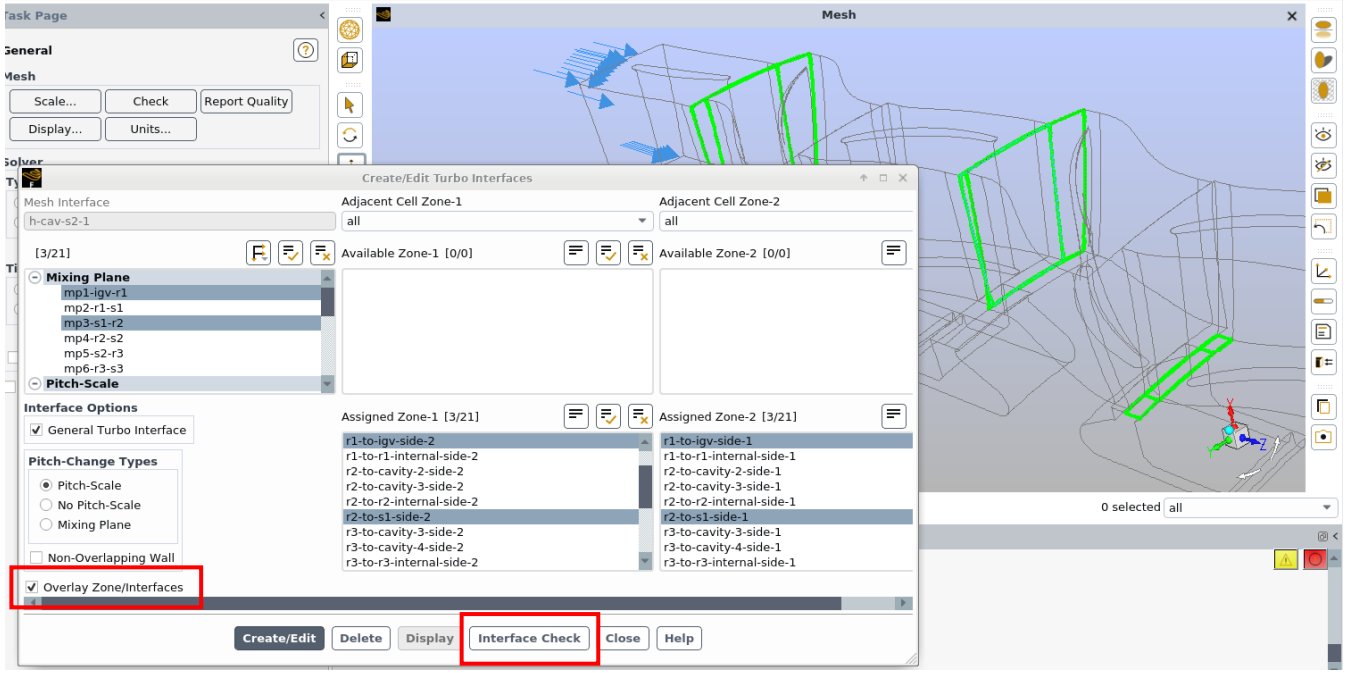


Turbomachinery

Fluent Turbo: Interfaces

General Turbo Interface (GTI) creation

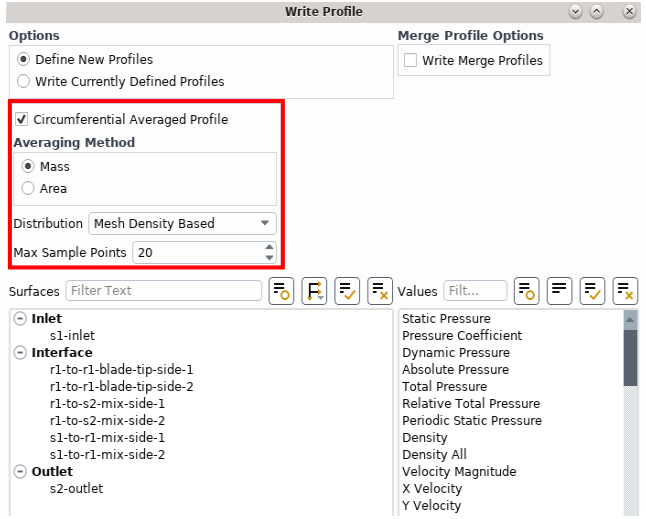
- **Overlay Zone/Interfaces** (beta): selected interface zones are highlighted and overlayed on full-geometry wireframe in the viewer, making zone selection much easier
- New **Interface Check** provides diagnostics



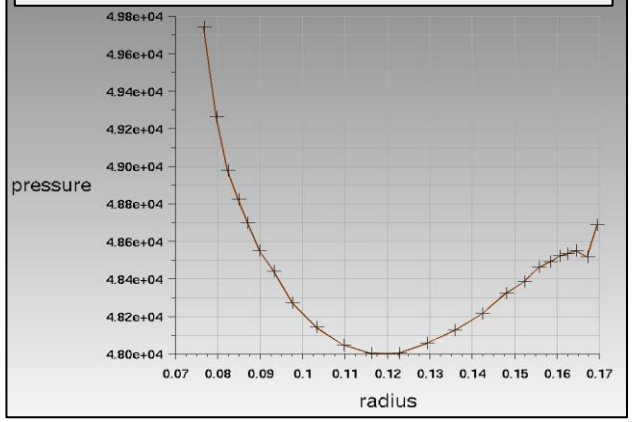
The Mixing Plane model now has improved band creation, leading to better conservation

Circumferential-averaged profile tool (beta)

- Export circumferentially-averaged profiles for solution variables, using inlet, outlet or interface surfaces



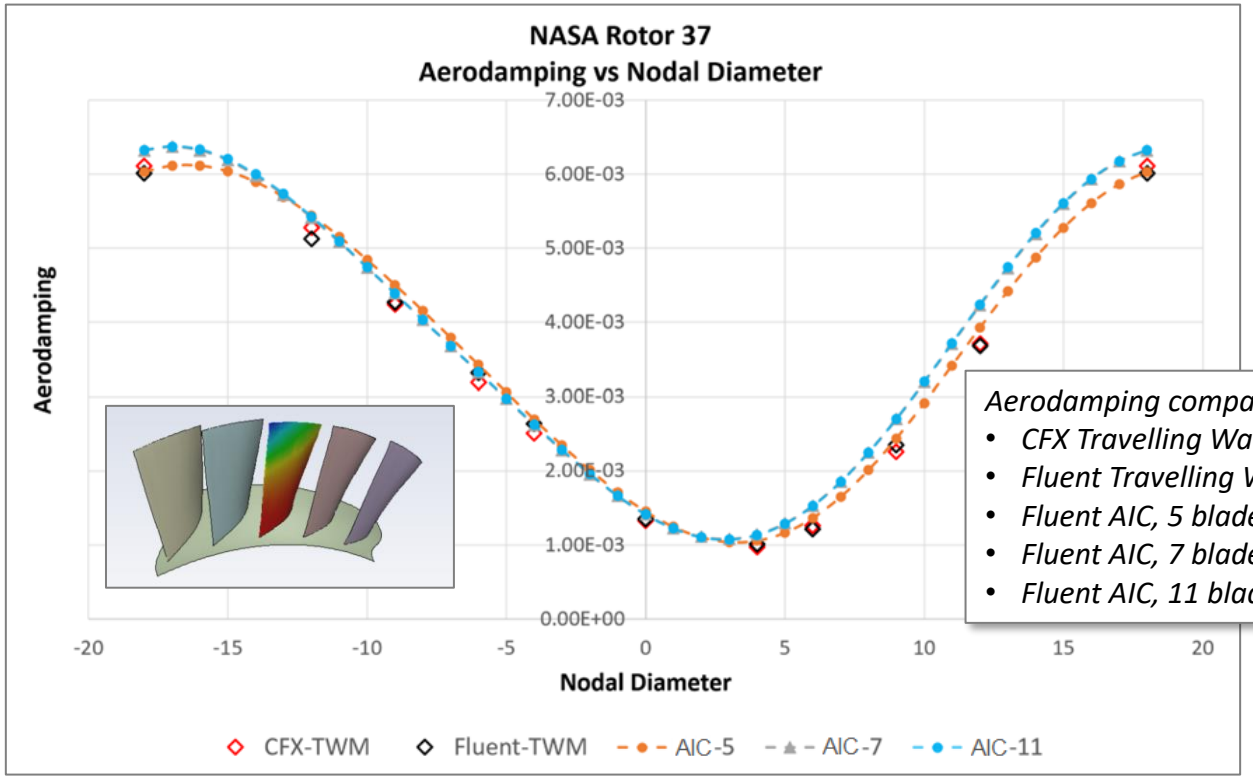
Circumferentially Averaged Pressure Profile



Fluent Turbo: Aeromechanics

Periodic Displacement and Aerodynamic Damping

- New Aerodynamic Influence Coefficient (AIC) method
  - For real mode shapes only
  - A single simulation provides results for the entire nodal diameter range
  - Results match Travelling Wave Method

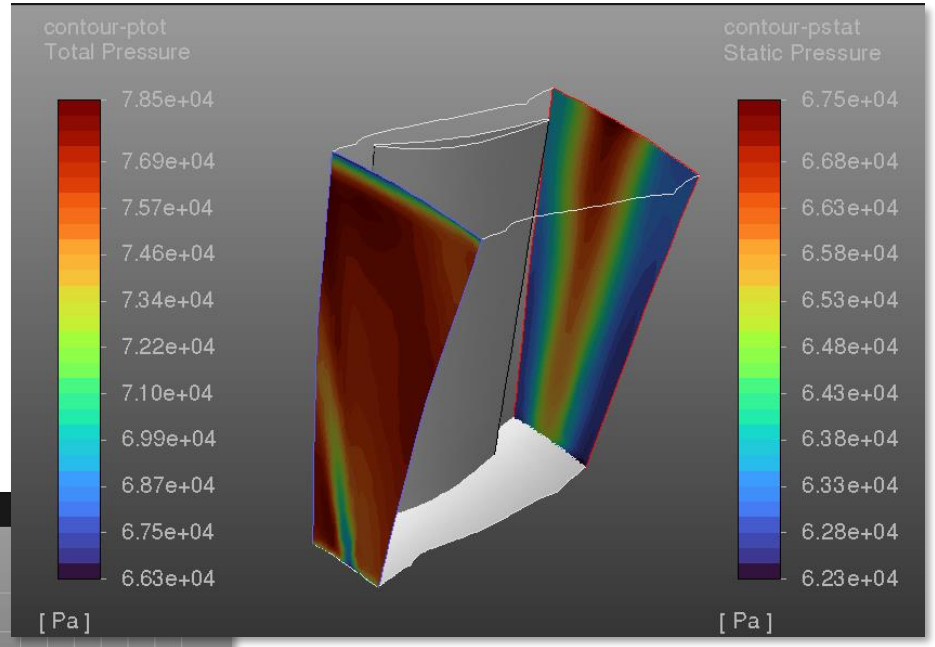


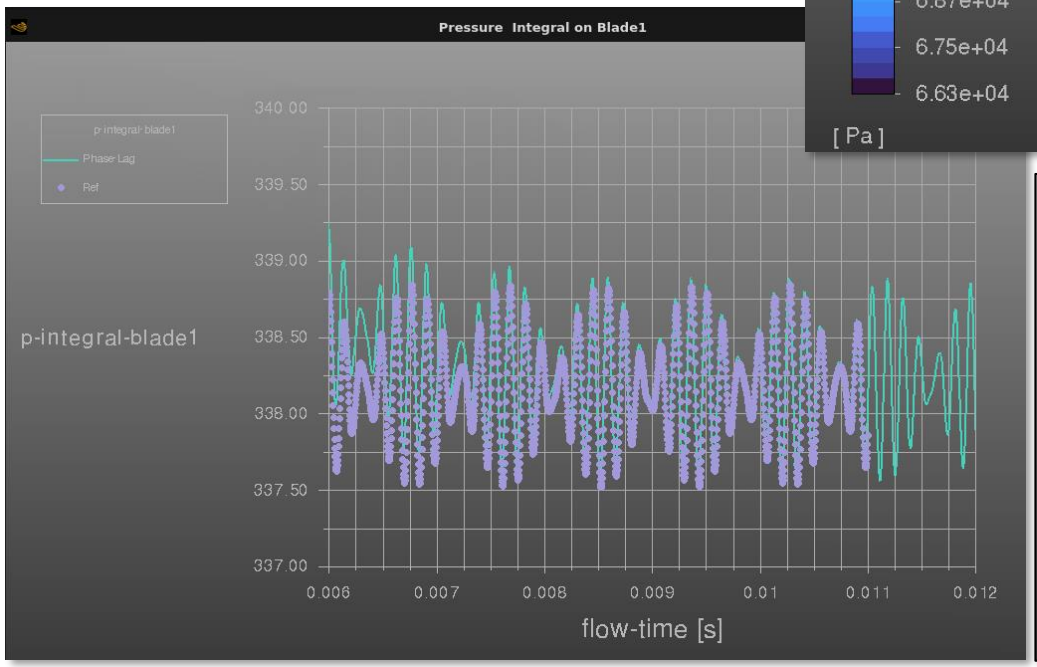
Aerodamping comparison showing:
- *CFX Travelling Wave Method*
- *Fluent Travelling Wave Method*
- *Fluent AIC, 5 blade passages*
- *Fluent AIC, 7 blade passages*
- *Fluent AIC, 11 blade passages*

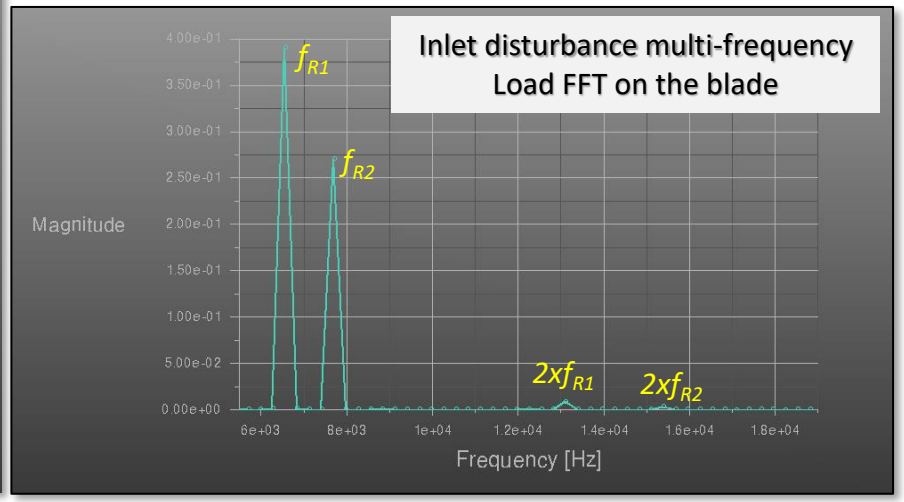
Fluent Turbo: Aeromechanics

Fourier-based Phase Lag

- Base model released
- Available use Cases:
  - Inlet/outlet flow disturbance
  - Blade flutter



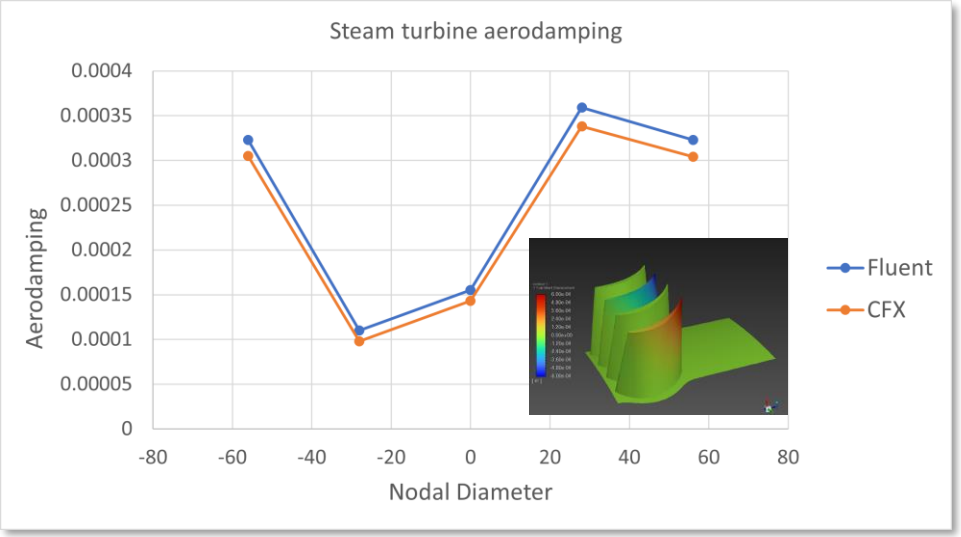




Fluent Turbo

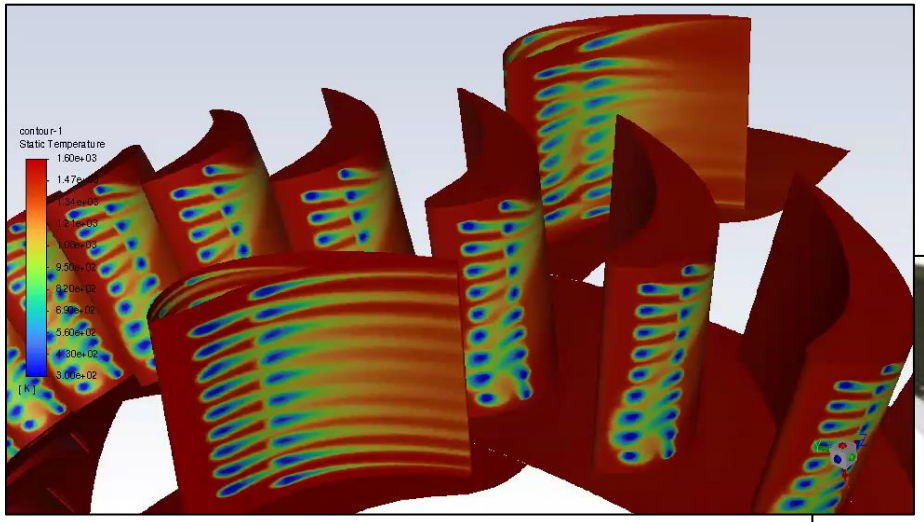
Wet Steam Non-Equilibrium Condensation (NEC) model for Steam Turbines

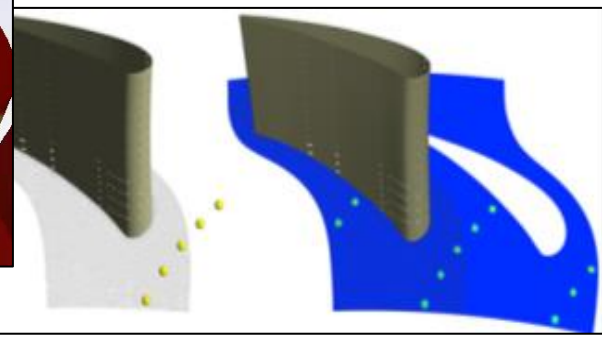
- Ability to activate or deactivate NEC per fluid zone
  - Run an initial solution without liquid formation and then easy activate the NEC model
- Wet Steam NEC available with Travelling Wave Method to model blade flutter in steam turbines



Blade Film Cooling Model for Gas Turbine

- Available with TRS cases (moving mesh)
- Automatic periodic expansion of virtual cooling holes across all passages
- Mass flow specification for the injection holes can be uniform or a profile can be used across the hole





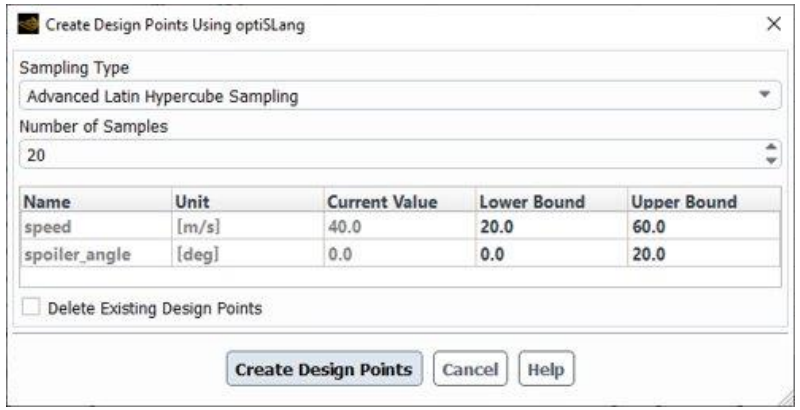
Injection locations automatically expanded

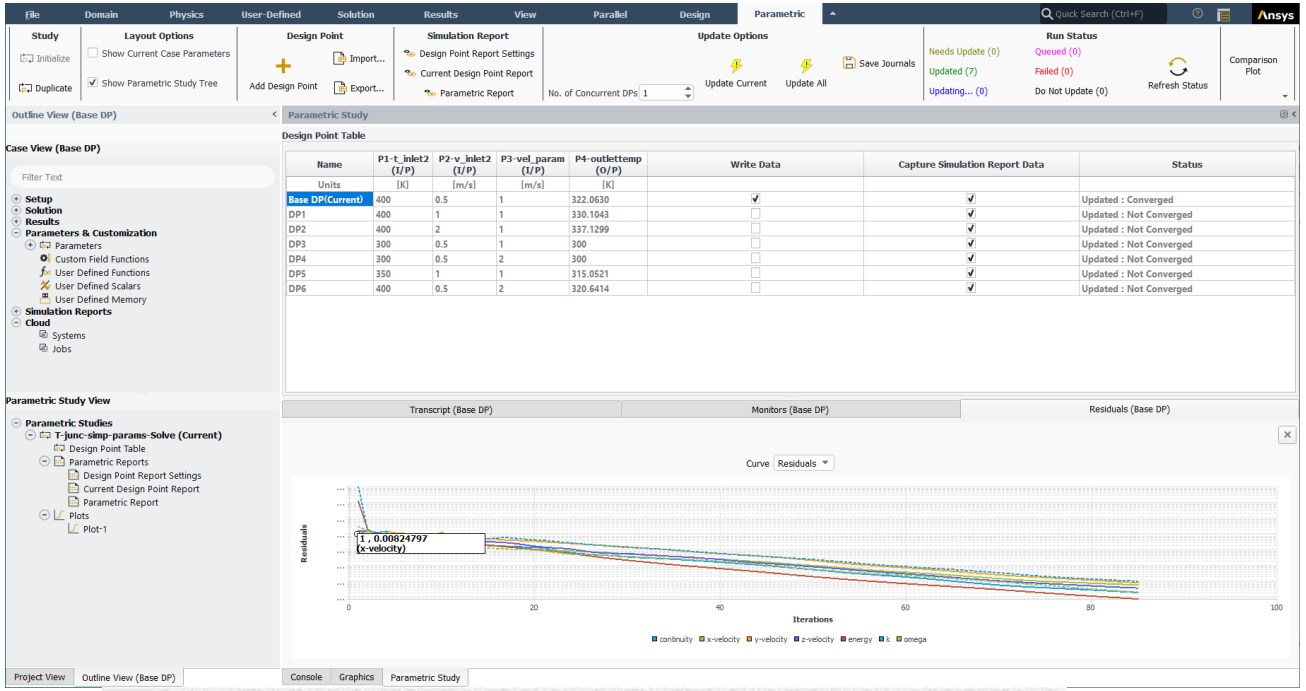
Workflows & UX

Design & Optimization

Parametric Solver Workflow embedded in Fluent

- **Parametric Workflow**
  - Distributed concurrent design point updates
  - Supports HPC-based and optiSLang-based parametric licensing
  - **optiSLang algorithms create the DoE in Fluent**
  - Conditional formatting in reports
  - “Open in optiSLang” option for detailed optimization
  - Demonstration of Parametric Morphing (next slide)



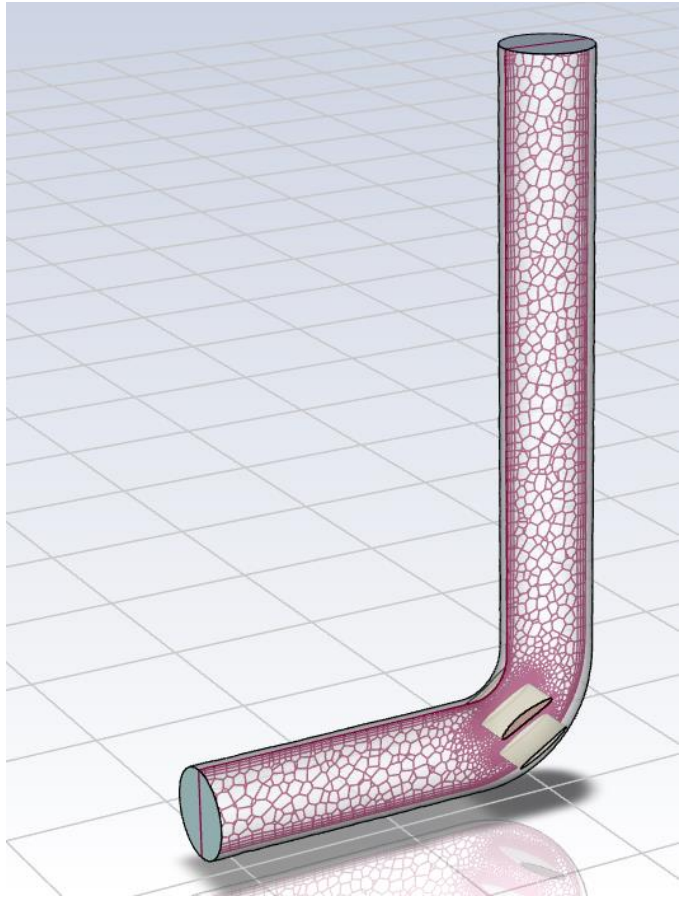


	spoiler_angle [deg]	speed [m/s]	cd-op	cl-op	lift_force [N]	drag_force [N]	axle_load_r [N]	axle_load_f [N]	Write Data	Status
Base DP	0	40	0.194686	0.068078	46.06951	201.4233	-244.089	197.7394	True	Updated
DP1	0.322885	21	0.214834	0.010839	-6.3107	61.81703	-49.3298	55.55459	False	Updated
DP2	0.130899	55	0.198069	0.052156	48.01741	389.9879	-414.524	365.9642	False	Updated
DP3	0.218166	39	0.198222	0.037436	7.633363	195.6364	-194.684	186.7793	False	Updated
DP4	0.165806	57	0.198785	0.047463	39.47194	420.8083	-432.212	392.1553	False	Updated
DP5	0.253072	29	0.203332	0.031182	0.582707	111.0209	-104.957	104.22	False	Updated
DP6	0.183259	45	0.198178	0.043446	19.15346	260.703	-266.192	246.6762	False	Updated
DP7	0.340339	58	0.203035	0.025982	-16.0231	462.5555	-402.447	417.8278	False	Updated
DP8	0.200712	47	0.198986	0.041406	17.13391	285.9157	-285.778	268.2467	False	Updated
DP9	0.113446	43	0.19673	0.053257	31.75311	235.8805	-258.581	226.5	False	Updated

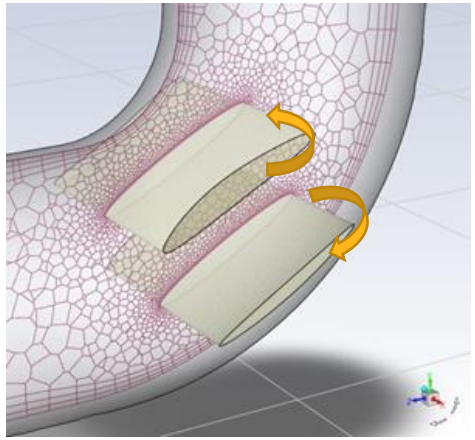
Design & Optimization

Morphing option added to Design Tool

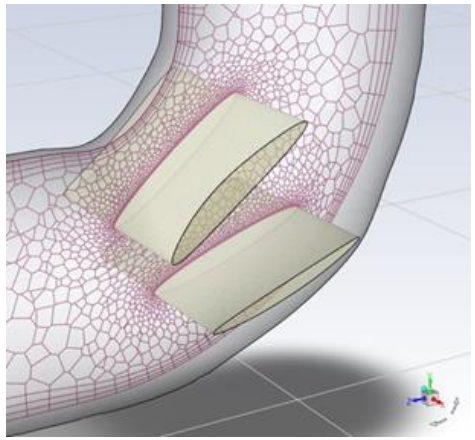
- **Design tool – Geometry exploration**
  - Streamlined design tool to easily morph the geometry and mesh
  - Enables the exploration of multiple designs without repeated mesh generation
  - Simply define the motion, morph, and re-compute the solution
- **Enhanced constrains in design tool**
  - Handle design conditions more efficiently and accurately
  - Simplified and improved user experience



Original



Morphed



Design & Optimization

Adjoint improved robustness and optimization criteria

- **Gradient-based Optimizer Improvements**
  - **Line search in optimizer:** automatically sets a step size for optimization, leading to a better optimal solution
  - **Enhanced execute command:** Deeper customization of gradient-based optimizer

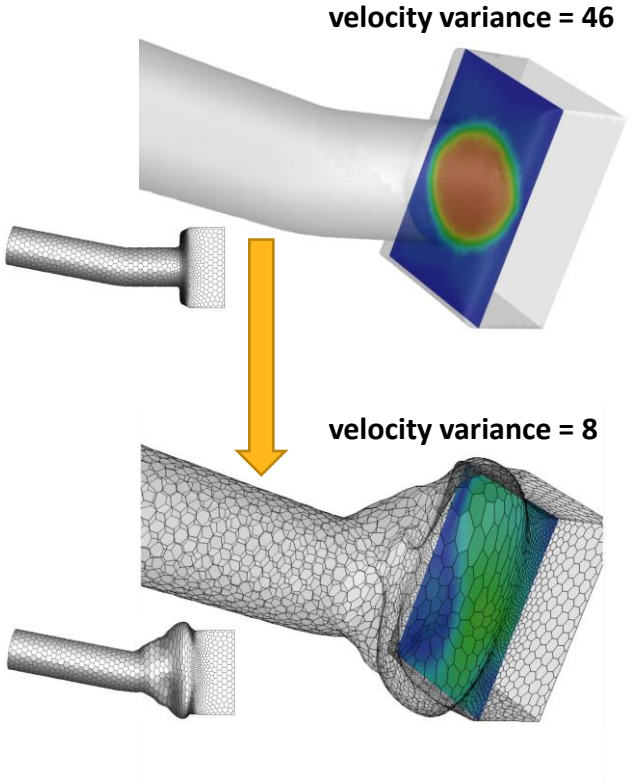
- **Physics**
  - Support **expression as the source terms** for all the equations

Name

noiseIntegral

Definition

```
VolumeInt(a*rho0*0.09*TurbulentKineticEnergyk*SpecificDissipationRateOmega*ratio*ratio*ratio*r
atio*ratio,['fluid'])
```

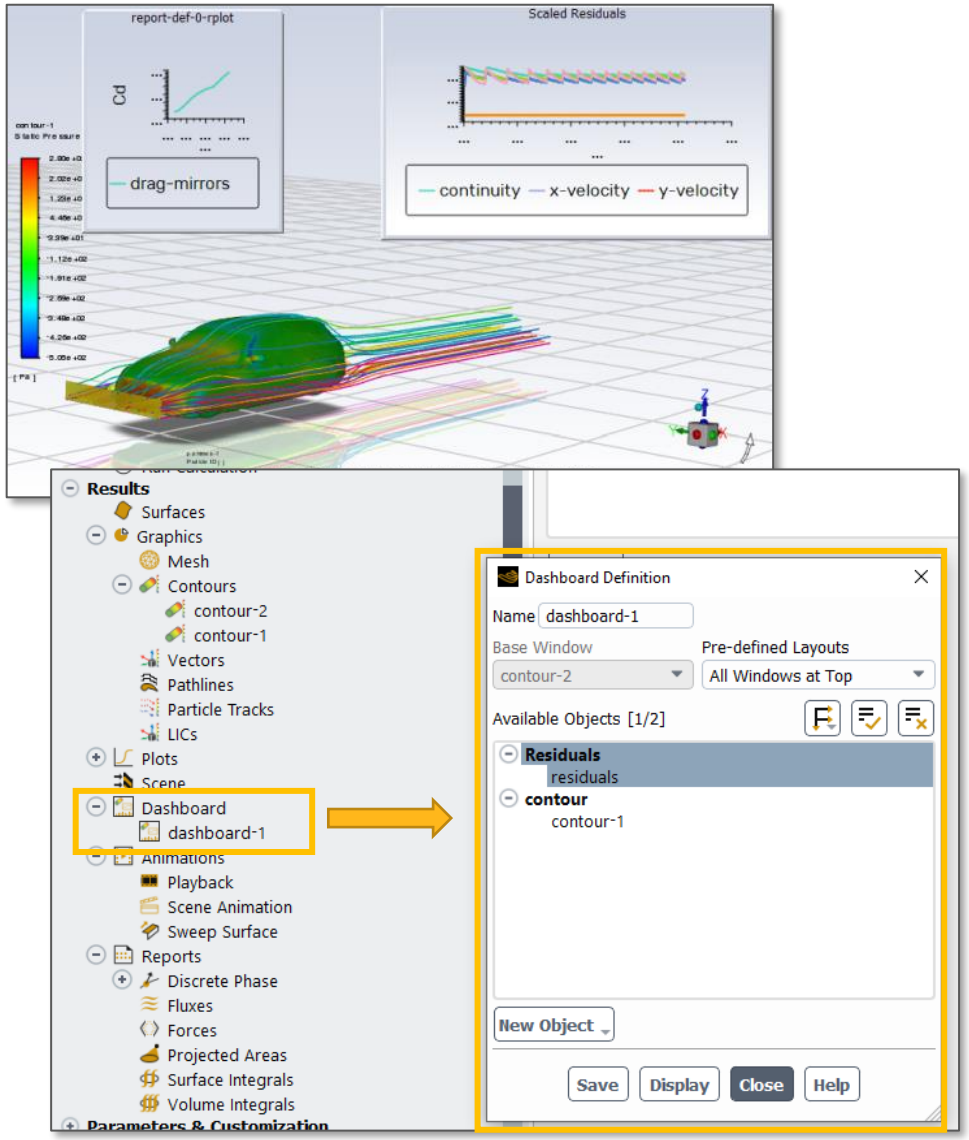


Line search in optimizer

- **Significant decrease in the observed variance using the Line Search method**
- **Using fixed or adaptive step size method only produced a slight reduction in the variance**

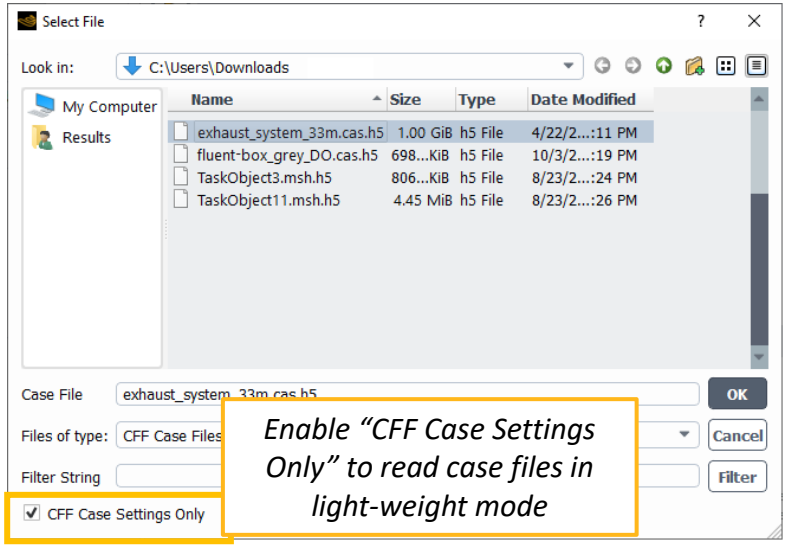
User Experience

- UI performance gains for cases with many zones
  - 2x – 10x faster for many UI operations when working with many zones, e.g. battery modelling
    - Open/edit panels, Zone generation, Copy/paste zones with shell conduction, etc …
  - Monitor point performance improved 10x for cases with many zones
- Dashboard manager for embedded graphics windows
  - Manage the layout of embedded graphics windows
  - Fluent saves the position of embedded windows with the case file
- Simulation Reports can now be exported to PowerPoint
- Transient statistics are now supported on arbitrary cut-planes

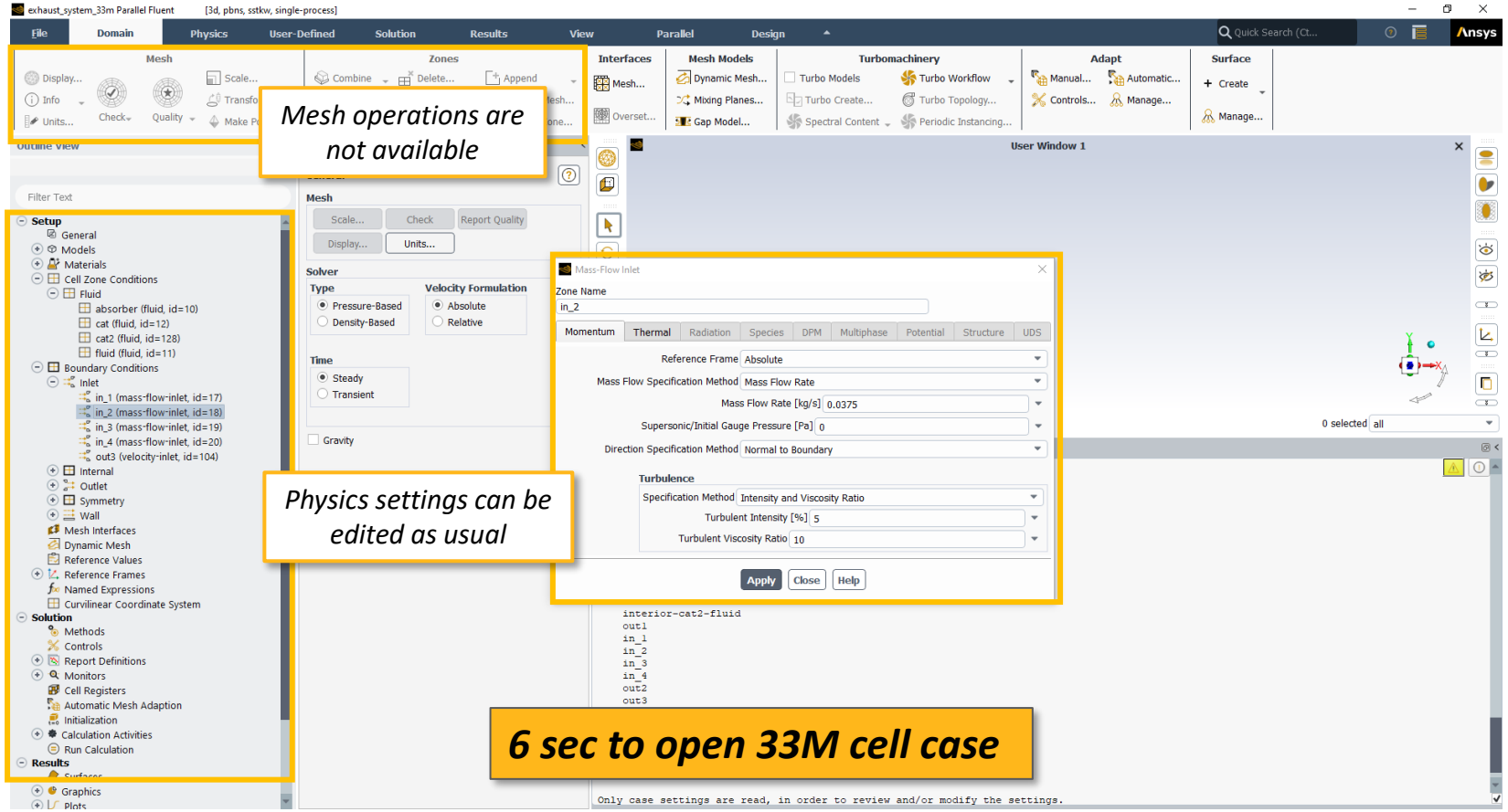


Light-weight Setup Mode (β)

- Quickly edit your setup without loading heavy-weight mesh and solution data
- In this mode, **read/write are fast and memory usage is minimal**



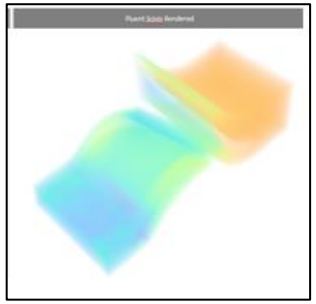
33M cells, serial	Case read time	Memory Usage
Light-weight mode	6 sec	1 GB
Standard mode	Minutes	18 GB

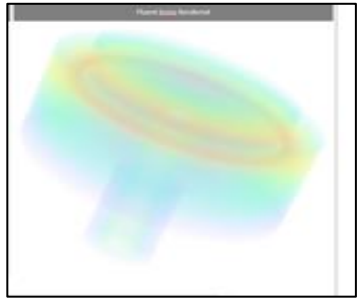


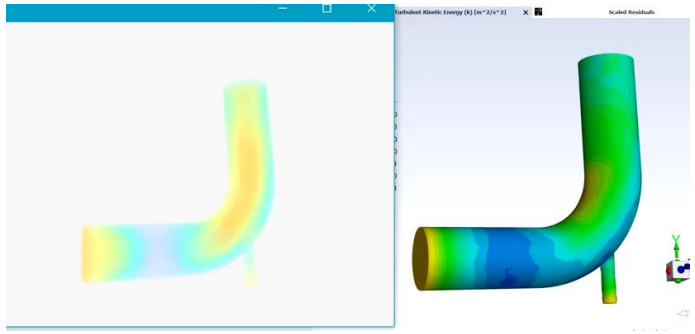
Graphics/Post

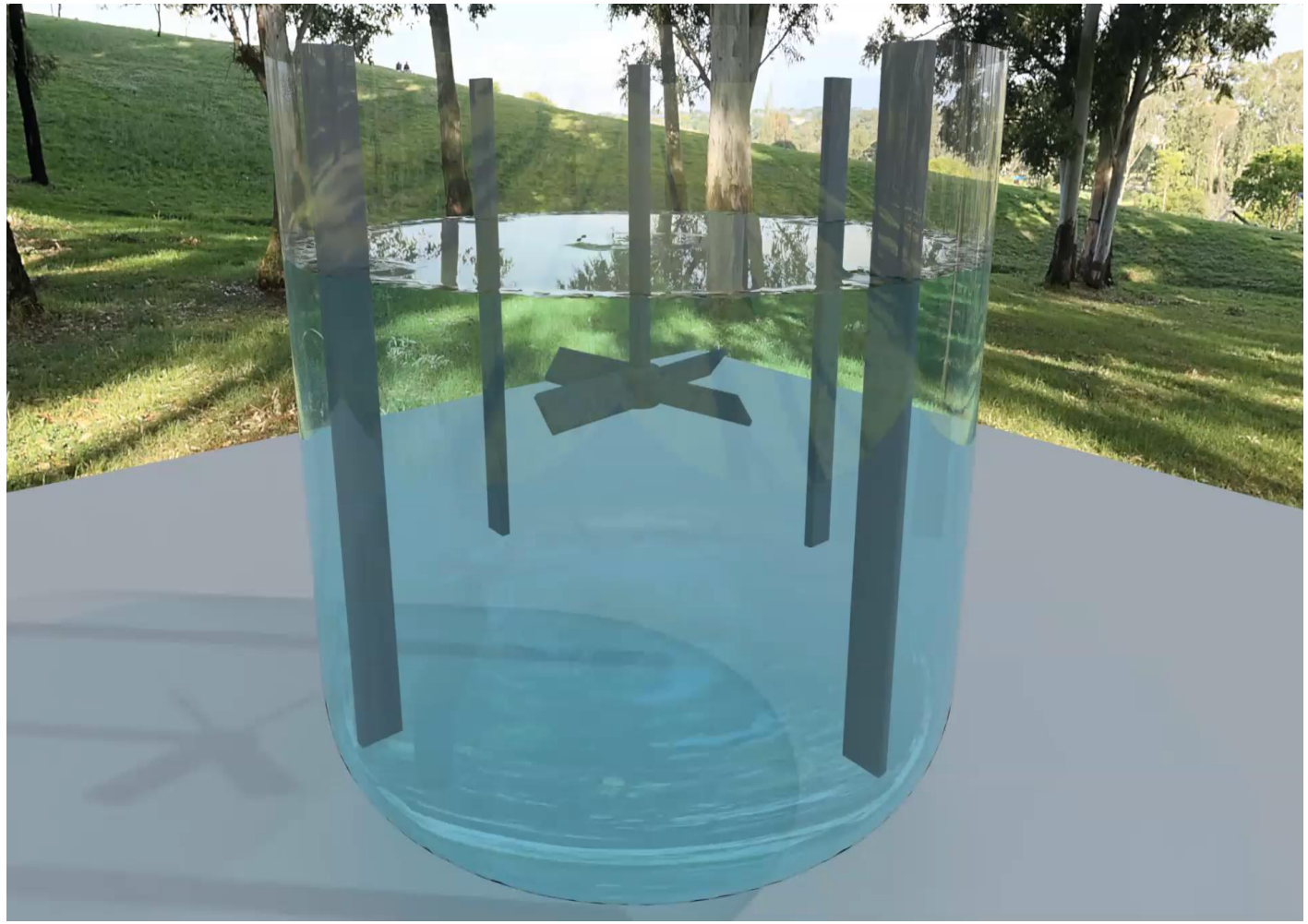
Photo-realistic rendering via OSPRay raytracing (beta)

- Supports embedded live preview, environments, ground plane, hardcopy and solution animation export
- Plus, Light presets, panoramic-360 camera, custom environments are supported



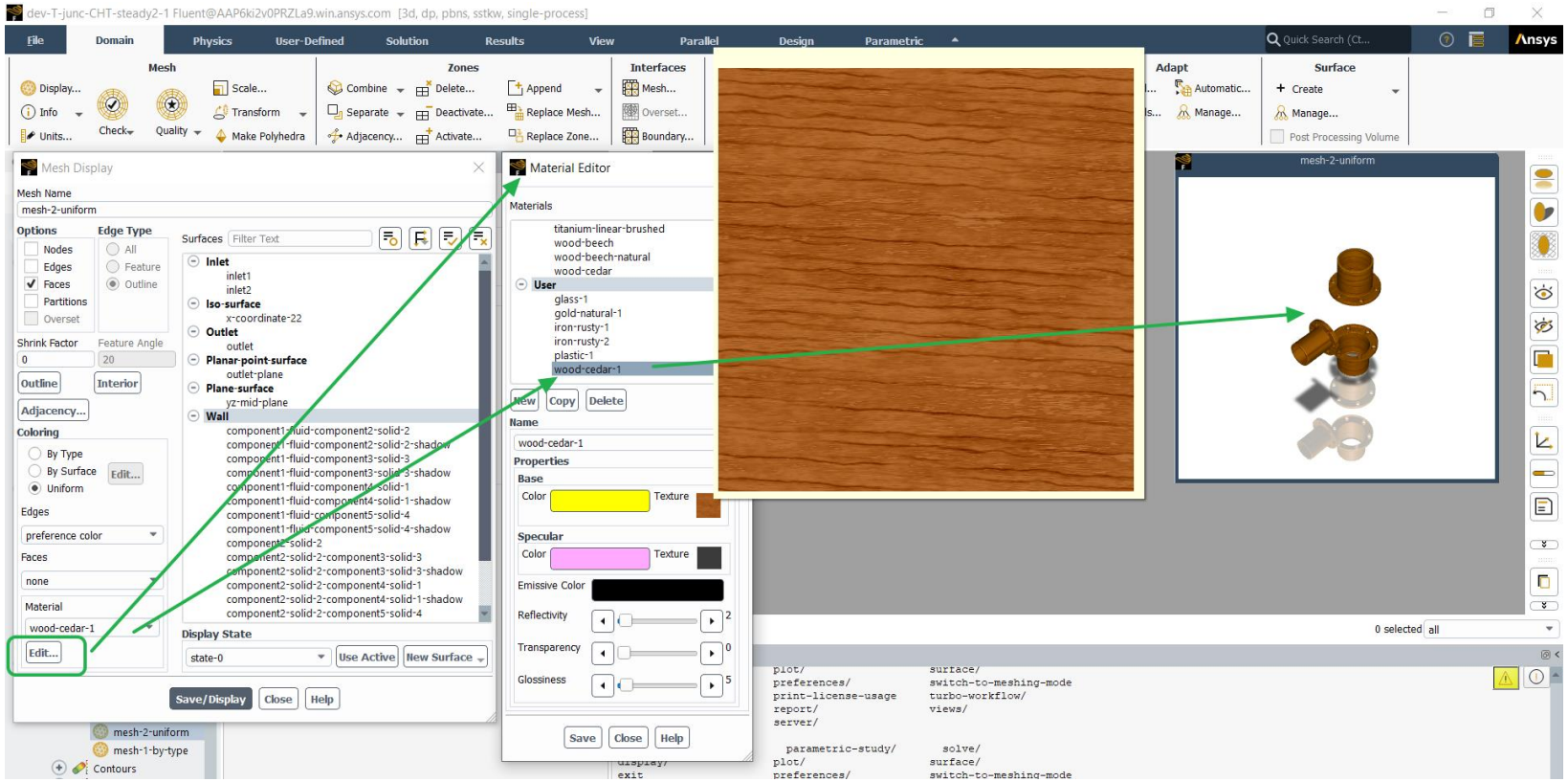






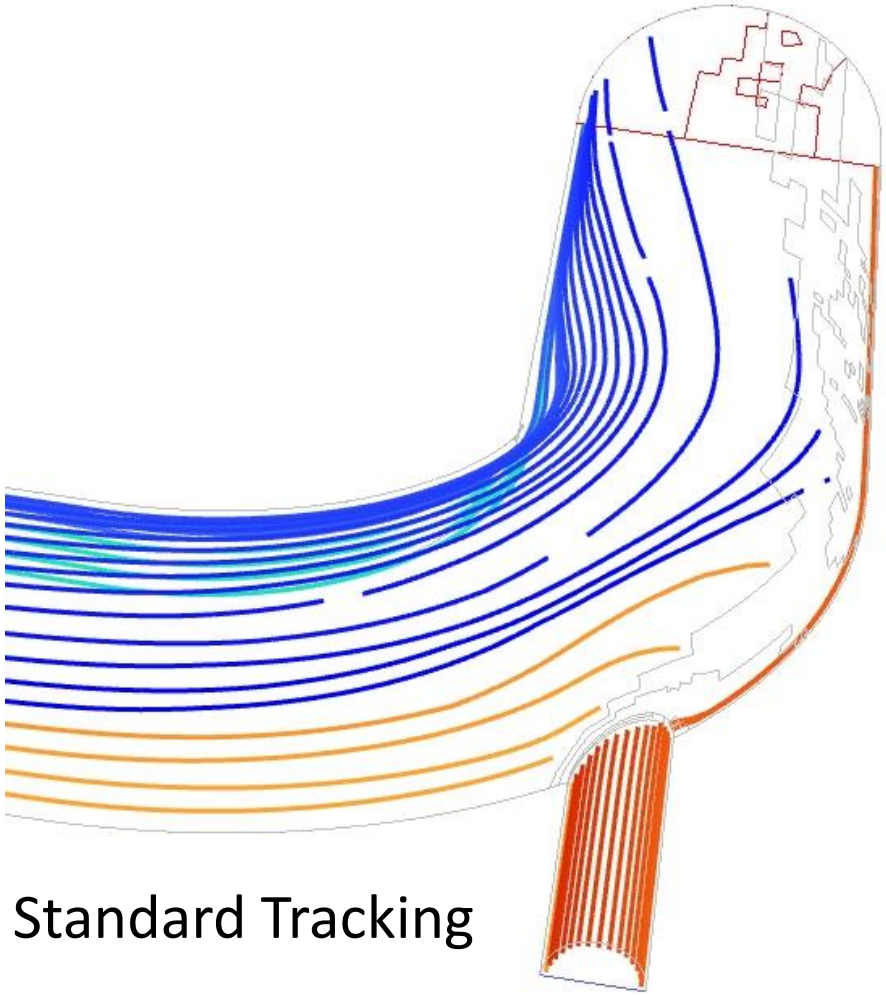
Material Editor provides realistic rendering

- The new Material Editor is used to define materials and their properties
- Preview how materials will look in the viewer
- Define, map and edit texture

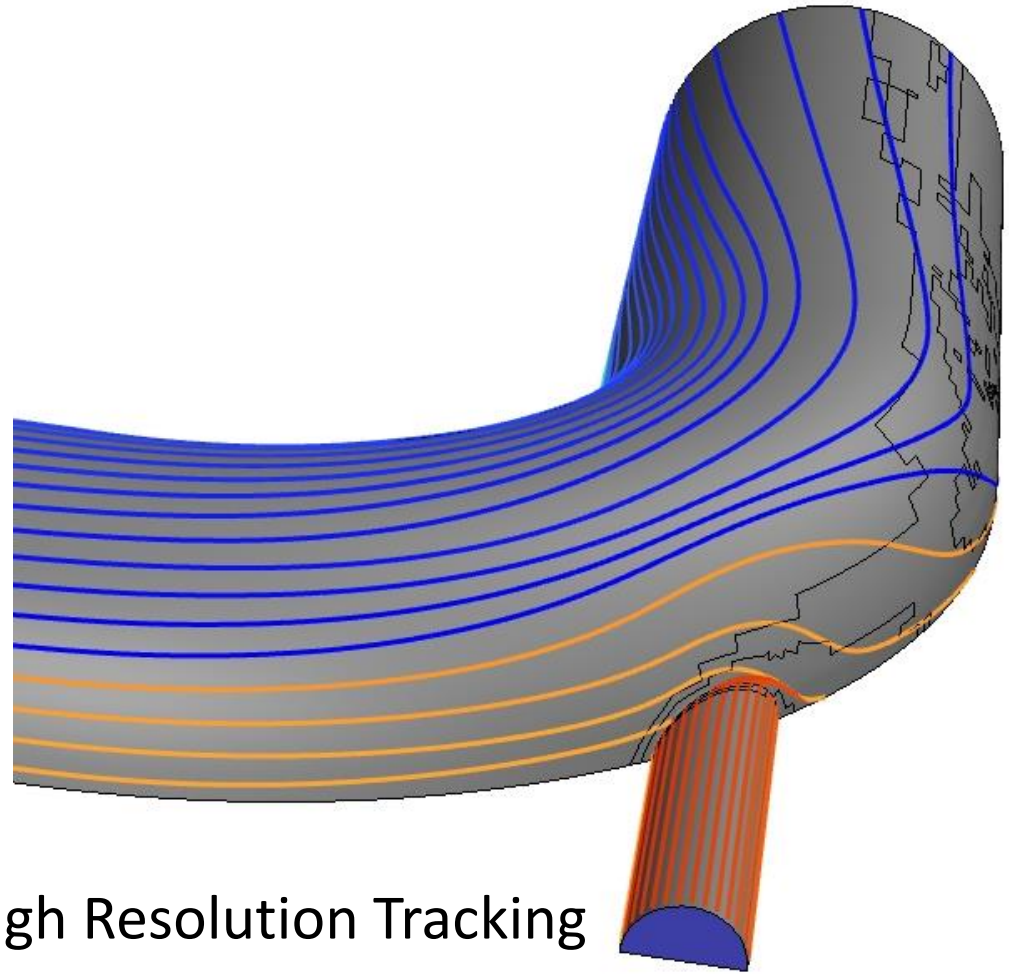


High Resolution Oil Flow Pathlines

Making use of HighRes to track pathlines directly on face zones



Standard Tracking



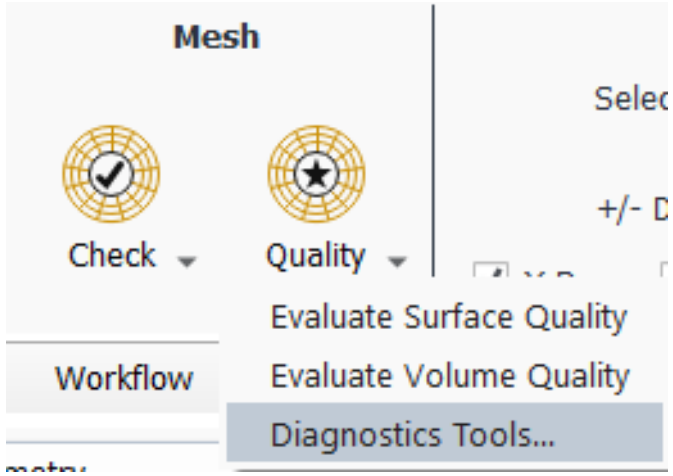
High Resolution Tracking

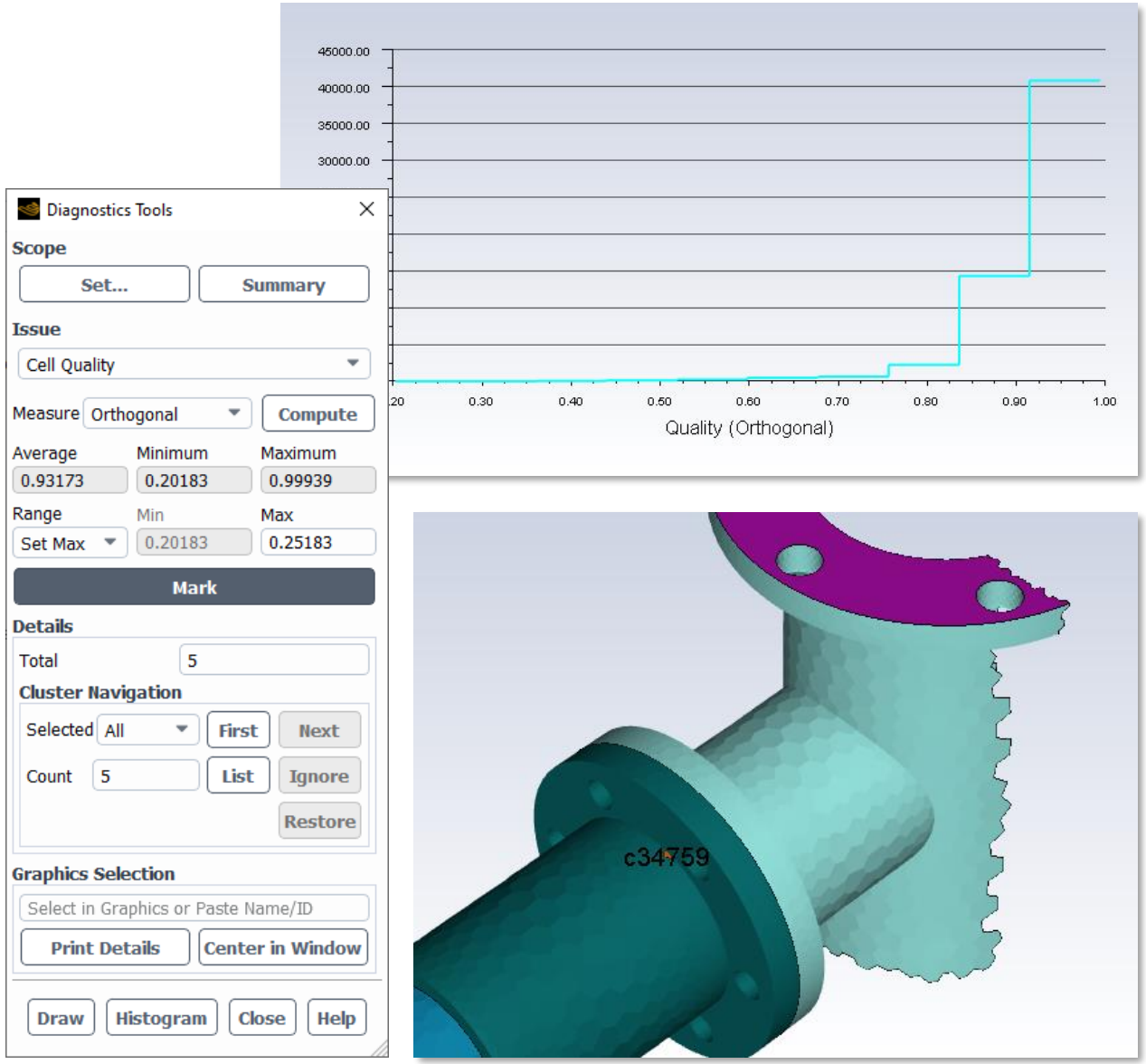
Watertight Meshing Workflow

Ansys

Mesh Diagnostics Tools

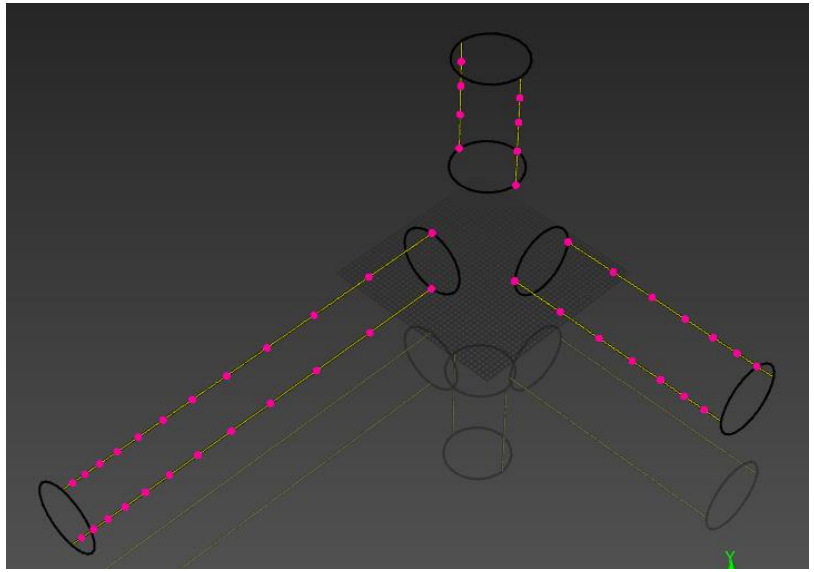
- New comprehensive mesh diagnostics tools with visualization
- Surface mesh and Volume mesh characteristics
- Can be used on entire mesh domain or scoped to particular zones/objects

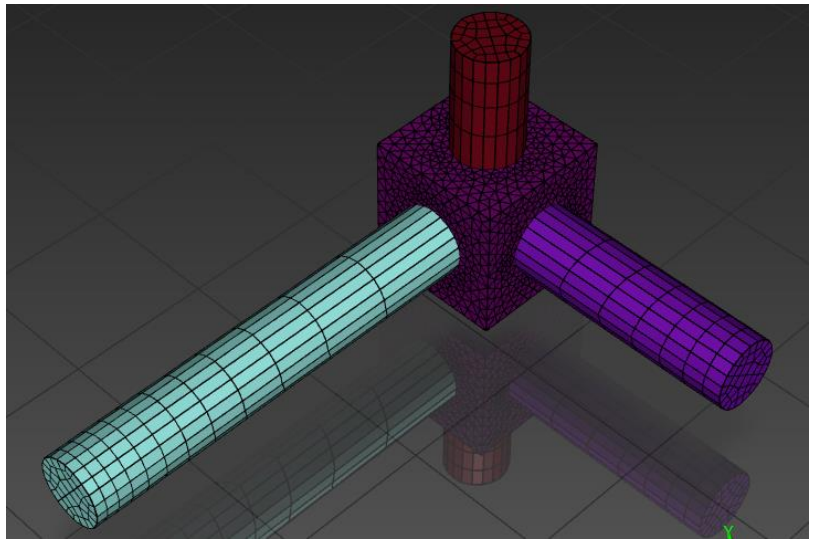


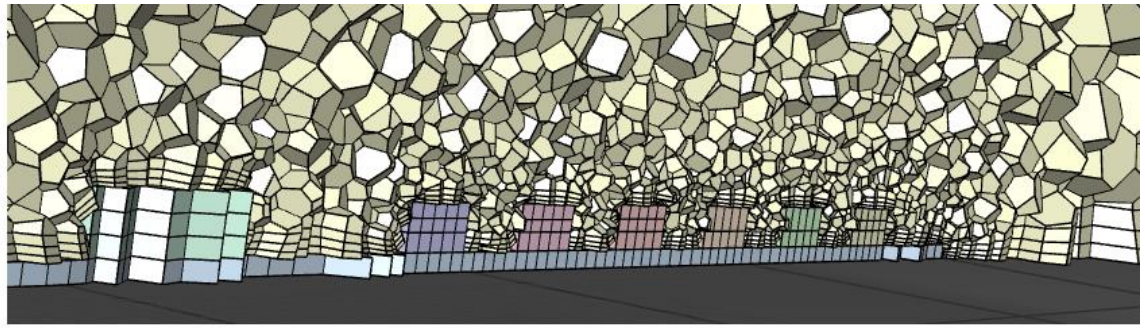


Multizone Meshing Enhancements

- New Edge-based sizing
  - Specify **Intervals**, **Size**, or **Smallest Height**
  - Single- and Bi-directional biasing
    - Specify **Growth Rate** or **Bias Factor**
  - Automatic edge-direction synchronization
  - Smart “Similar Edge” selection automatically selects all parallel edges of similar length
- “On-the-fly” choice of non-conformal connection to unstructured tet/hexcore
  - Can be used even if shared topology is present
  - Poly / poly-hexcore connection must be non-conformal

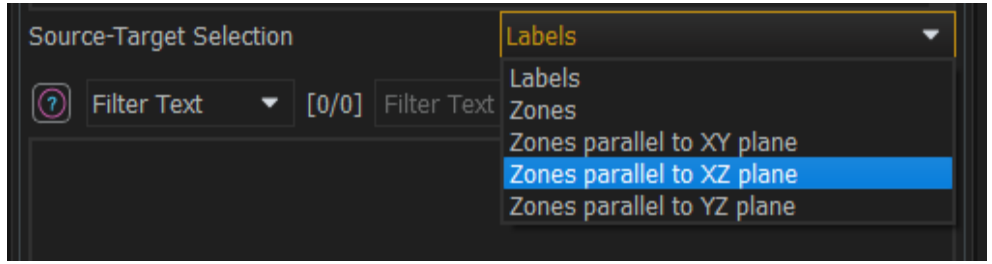




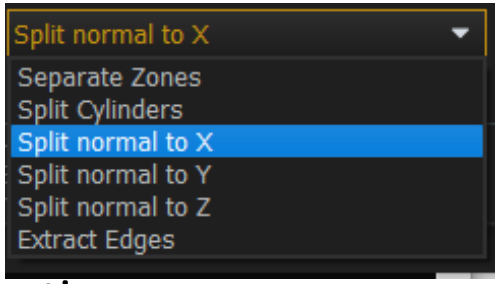


Additional Multizone selection features

- In Add Multizone Controls
  - Ability to select All source-target zones parallel to a global plane
    - Zones parallel to XY plane
    - Zones parallel to XZ plane
    - Zones parallel to YZ plane
  - Improved persistence with design changes



- In Manage Zones
  - Ability to split bodies normal to a global direction
    - Split normal to X
    - Split normal to Y
    - Split normal to Z
  - Split non-cylindrical shapes (e.g. heat exchanger tubes) in selected direction

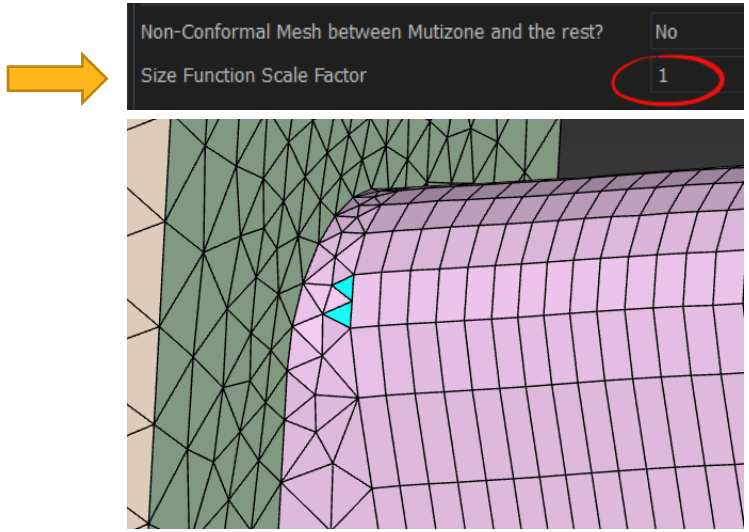


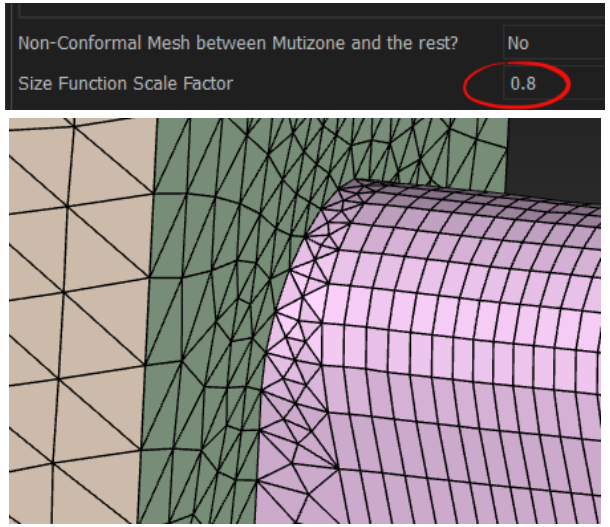
Multizone Size Function Scale Factor

- In some situations, the Multizone mesh is too coarse compared to the adjacent Surface mesh and the conformal connection is not completed, resulting in free faces

```
---- Warning--- The multizone regions (tube1 tube2) could not be conformally connected to the remaining surface mesh.
There are two work-arounds
---- Warning--- Either set the option 'Non-Conformal Mesh between Mutizone and the rest?' to 'yes', or reduce the
'MultiZone Size Function Scale Factor' to get a better size match
```

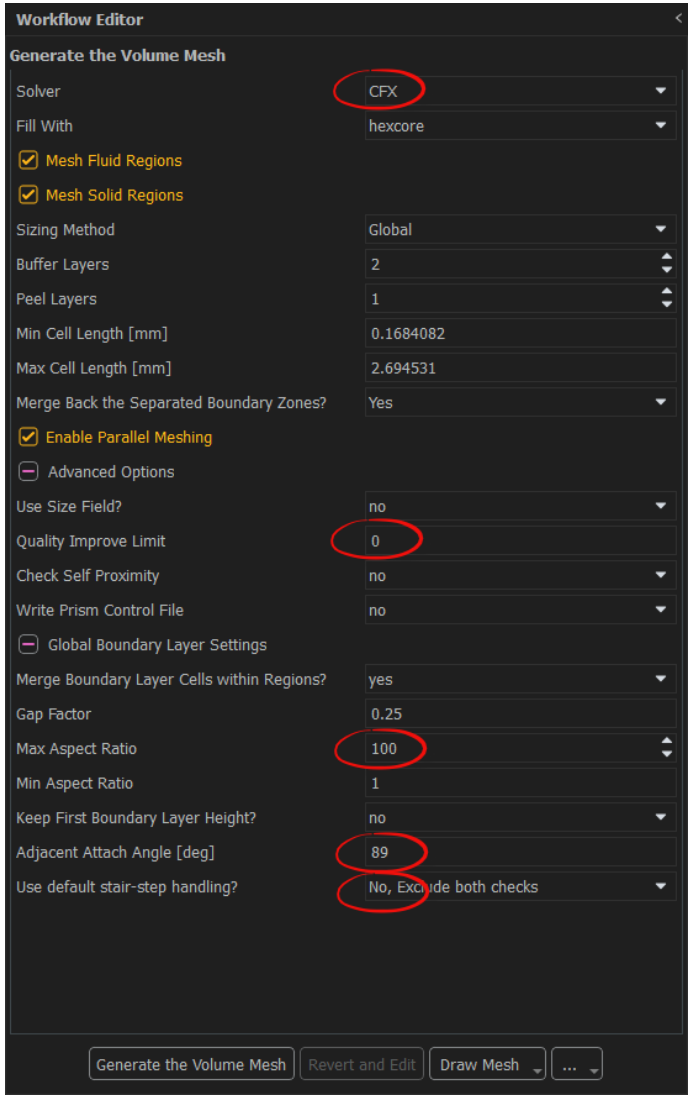
- A **Size Function Scale Factor** has been introduced that will scale the size field as seen by the multizone mesh
  - Located in the **Generate The Multizone Mesh** task





Solver Selection in Generate Volume Mesh

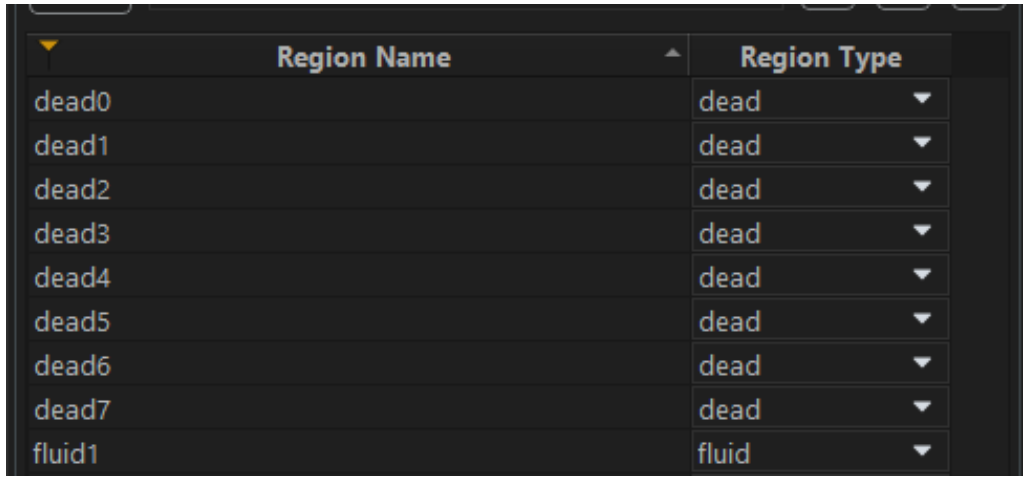
- **Ability to choose Fluent or CFX as target solver**
  - Avoids creating meshes that are incompatible and/or cause issues when read in CFX
    - Available volume fill types restricted:
      - Tetrahedral (default) and hexcore
    - Modifies **Global Boundary Layer Settings** defaults:
      - Max Aspect ratio 100
      - Adjacent angle 89
      - Exclude both stair-step checks
    - Advanced options hidden / modified
      - **Avoid 1:8 transition**
      - **Quality Limit** set to 0 – skips post-improve step that can cause negative control volumes in the CFX solver



WTM Usability Enhancements - Region and Volume Mesh Options

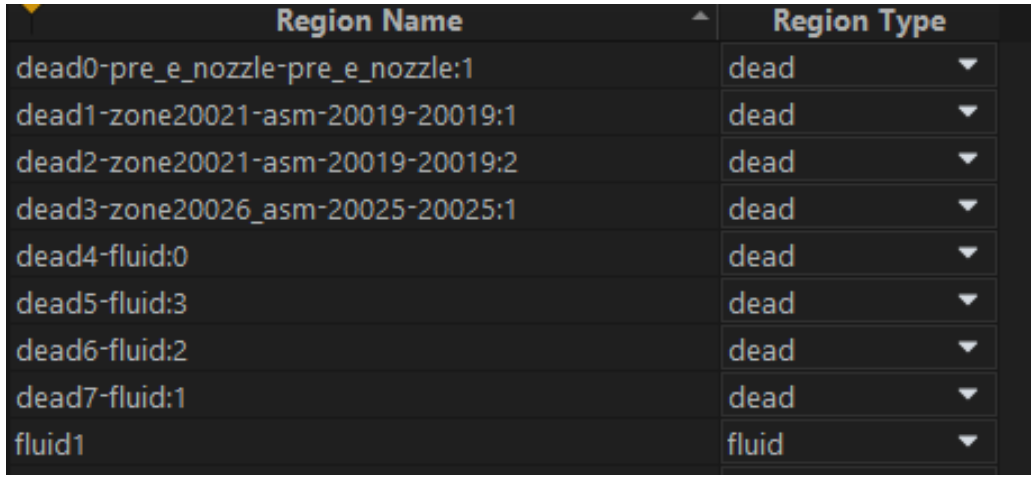
- Retain original name for dead regions
  - Voids or Dead regions are normally just named as "dead0", "dead1", etc.
  - But originally these regions are named after adjacent regions; fluid:1
  - New option in the Create Regions Task to retain the original names as suffixes

No - default



Region Name	Region Type
dead0	dead
dead1	dead
dead2	dead
dead3	dead
dead4	dead
dead5	dead
dead6	dead
dead7	dead
fluid1	fluid

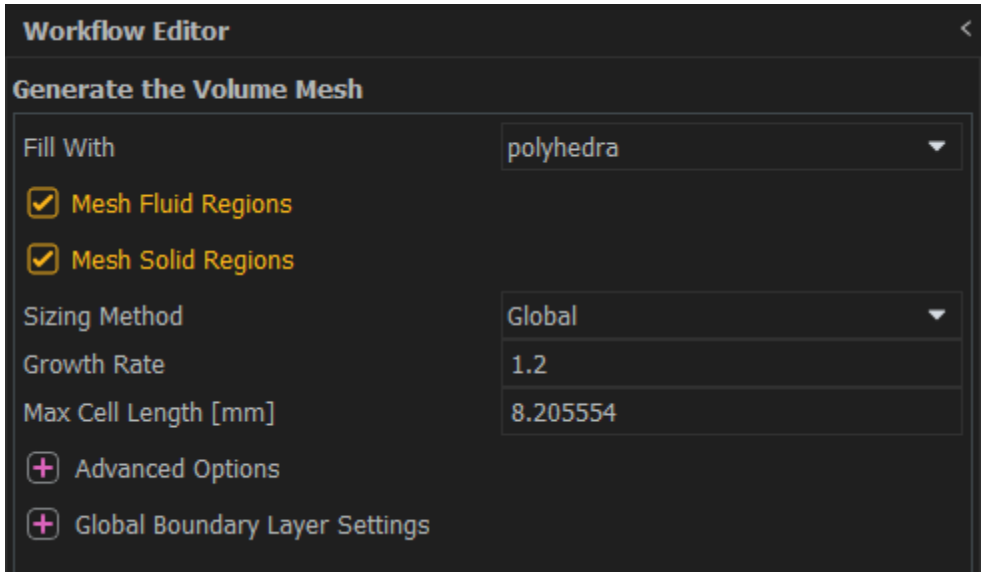
Yes



Region Name	Region Type
dead0-pre_e_nozzle-pre_e_nozzle:1	dead
dead1-zone20021-asm-20019-20019:1	dead
dead2-zone20021-asm-20019-20019:2	dead
dead3-zone20026_asm-20025-20025:1	dead
dead4-fluid:0	dead
dead5-fluid:3	dead
dead6-fluid:2	dead
dead7-fluid:1	dead
fluid1	fluid

WTM Usability enhancements - Region and Volume Mesh Options

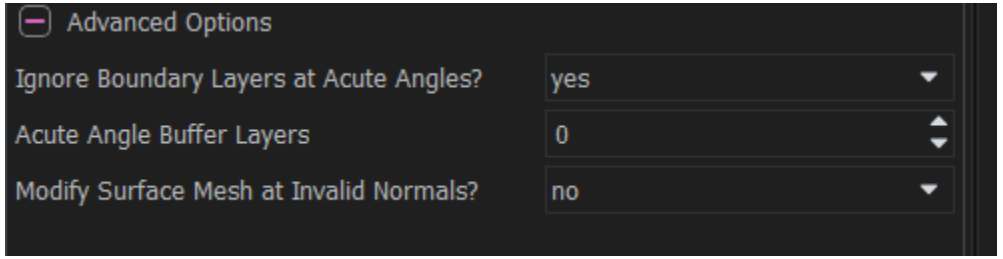
- Mesh Fluid or Solid Regions selectively
  - To complement the “Mesh Solid Regions” option in Generate the volume mesh task, a new option “Mesh Fluid Regions” has been added



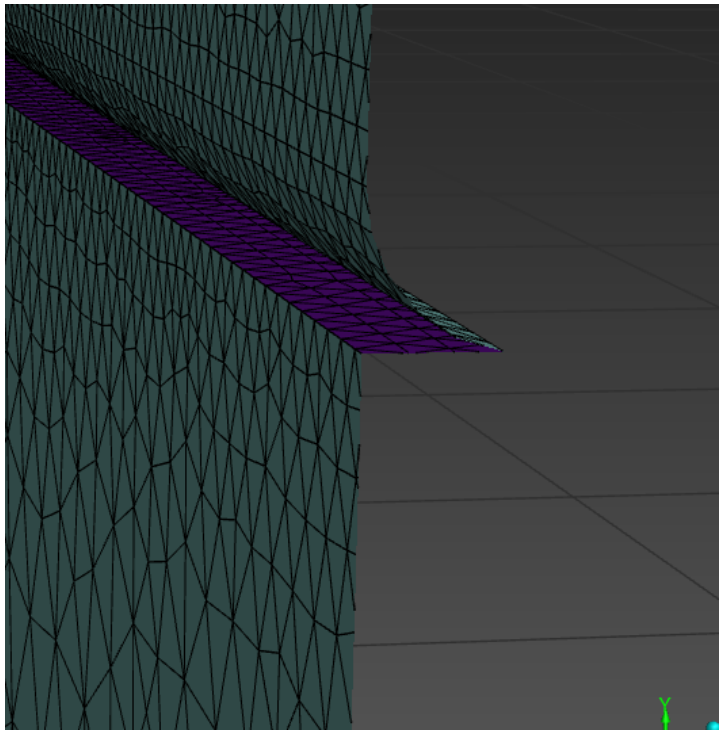
  - Allows independently creating fluid or solid regions of different types and save separately (FSI cases for example)
  - At least one must be enabled

WTM Usability enhancements - Improve Boundary Layer settings

- Add Buffer layers to boundary-layer-ignore at acute angles
  - A new advanced option has been added in Add boundary layer task

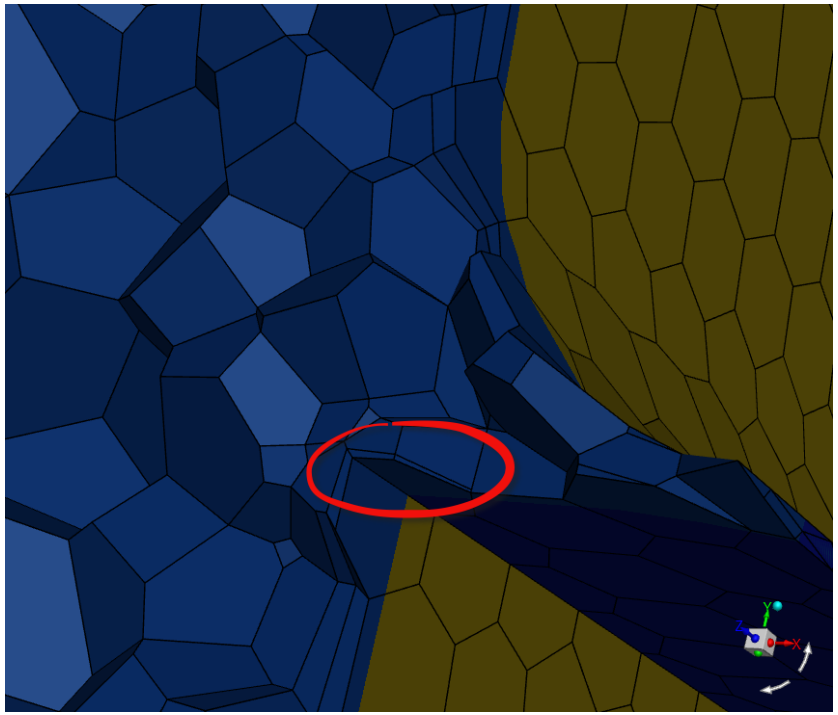


  - Increasing the Acute Angle Buffer Layers value increases the number of faces for which boundary layer will be ignored at acute angles

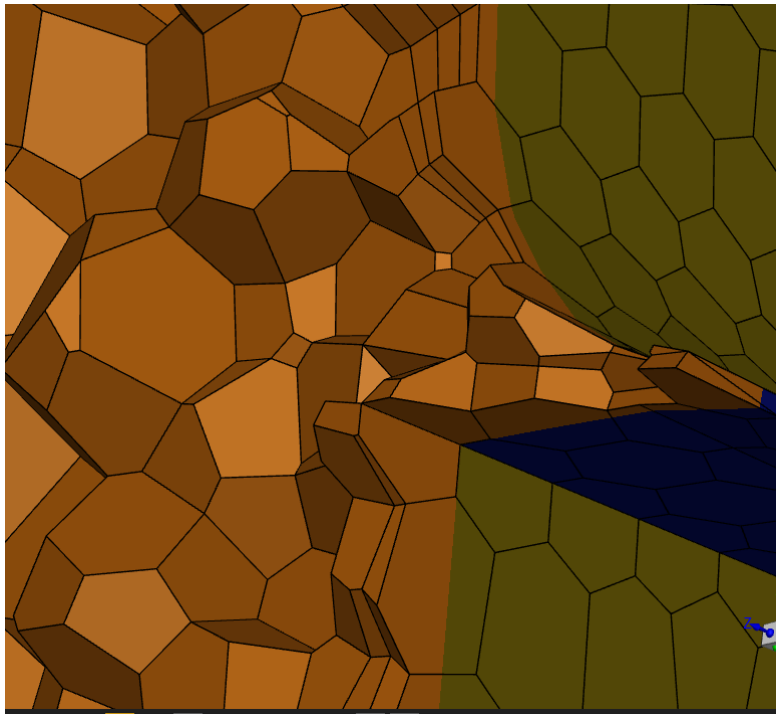


WTM Usability enhancements - Improve Boundary Layer settings

- Example
  - Avoid boundary layers in a fillet area



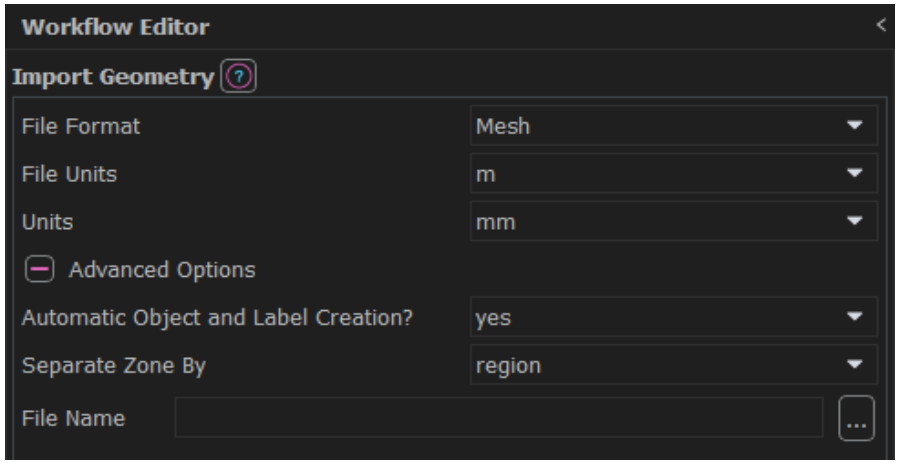
Acute Angle Buffer layer = 0



Acute Angle Buffer layer = 1

WTM Usability enhancements - Additional import Options

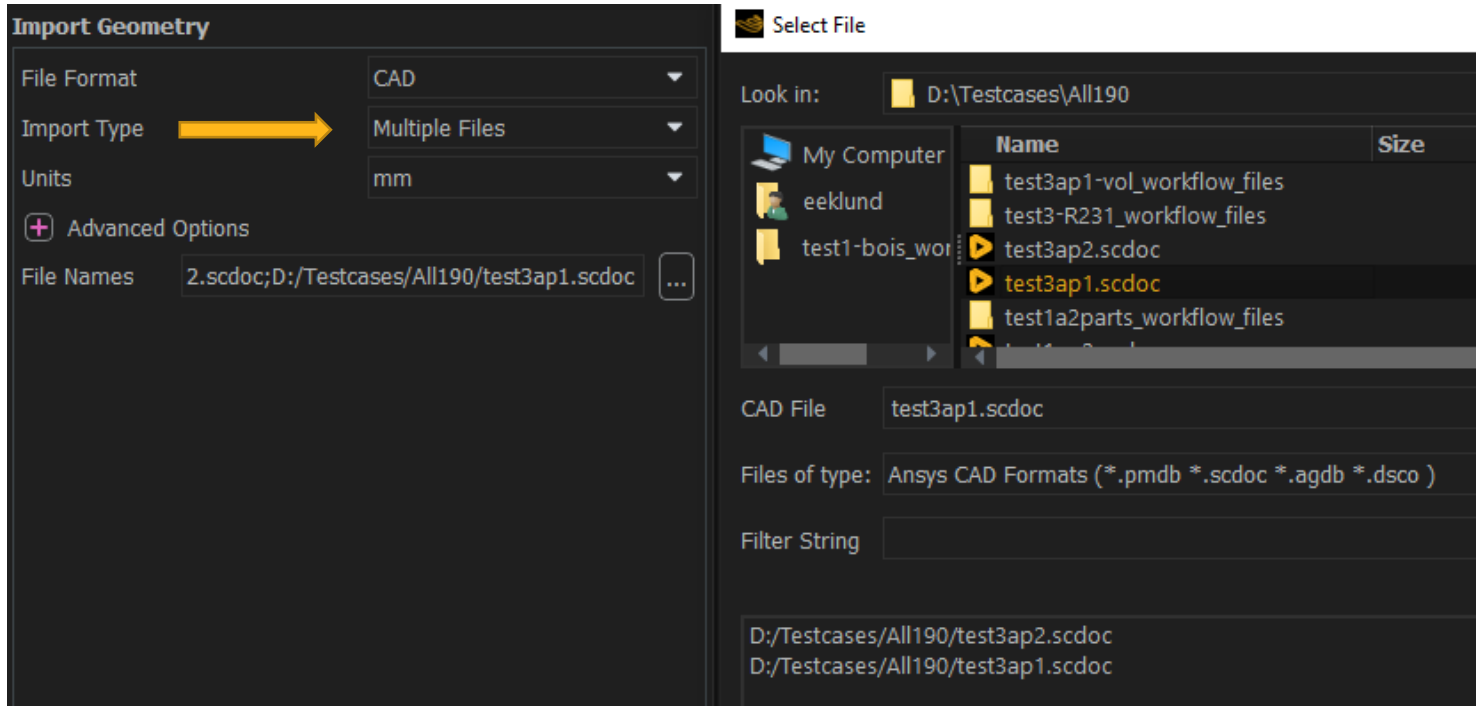
- Simplified Mesh import
  - A new advanced option; “Automatic Object and Label Creation?”, has been added in Import Geometry, when the File Format is set to Mesh



  - Default option is “yes”.
  - If set to “no”:
    - No Labels are created
    - A single mesh object is created using all zones
      - Exception: Zones including the string “*boi*” are excluded from the single mesh object creation, so that Bodies of Influence remain separate
  - For very large cases, this will dramatically increase the speed of mesh import

WTM Usability enhancements - Additional import Options

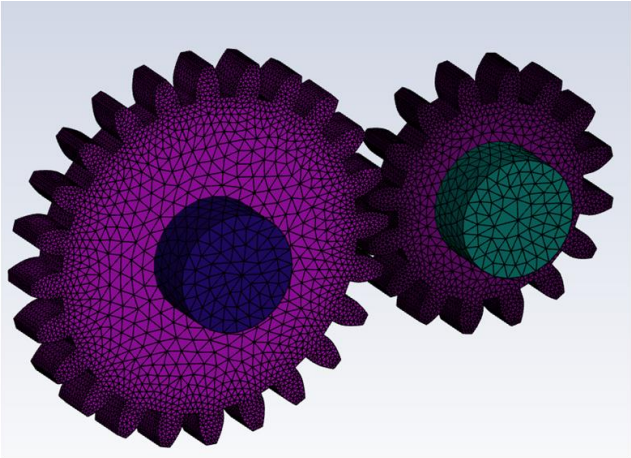
- Multiple CAD import
  - A new option; “Import Type”, has been added in Import Geometry, when the File Format is set to CAD
  - By default, the option is set to “Single File”
  - If this option is set to “Multiple Files”:
    - The File selection panel changes to multiple selections
    - Files selected must be
      - In the same Directory
      - Of the same CAD format

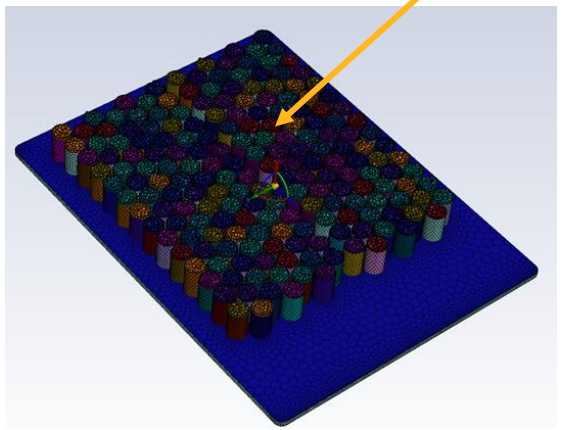


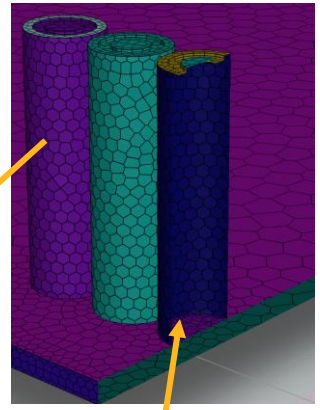
Fault-Tolerant Meshing

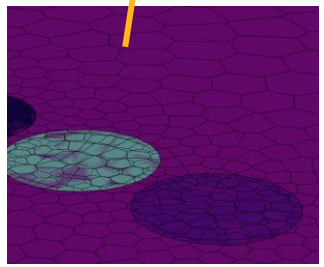
FTM: CFD Surface Mesh / Conformal Mesh

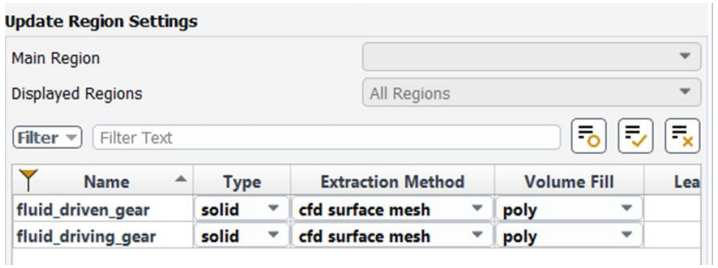
- New extraction option in Update Region Settings: "cfd surface mesh"
  - Uses new surface meshing technology offering higher performance /robustness on some challenging cases
- Optional geometry projection during extraction; option in Generate Surface Mesh
  - Project on Geometry of the CFD Surface Mesh Objects [Disabled by default]











Non-conformal meshes generated where overlapping faces are not shared

Conformal meshes created where faces are shared

Fill Without Size Field

- When volume meshing in parallel using size fields, total memory is increased by duplication of size fields
- **Use Size Field** = **No** reduces memory requirements by using specified parameters to control the interior mesh size:
  - **Octree Peel Layers**
  - **Octree/Boundary Size Ratio**
  - **Buffer Layers**
  - **Tet/Poly Growth Rate**

Small battery (no solids)
20 cores on single node (windows)

Generate the Volume Mesh

Quality Improve Limit	0.04
☑ Enable Parallel Meshing for Fluids	
☐ Save Mesh	
☐ Enable Region Settings	
⊖ Advanced Options	
Avoid 1/8th Octree Transition in Hexcore Region	No
Octree Peel Layers	1
Use Size Field	Yes
Conformal Prism Split	No

Total cell count: 1417957
14.7 minutes

```
-----------------------------
| Virtual Mem Usage (GB)    |
| Current       Peak        |
-----------------------------
| 44.7191       51.3932     |
-----------------------------
```

Generate the Volume Mesh

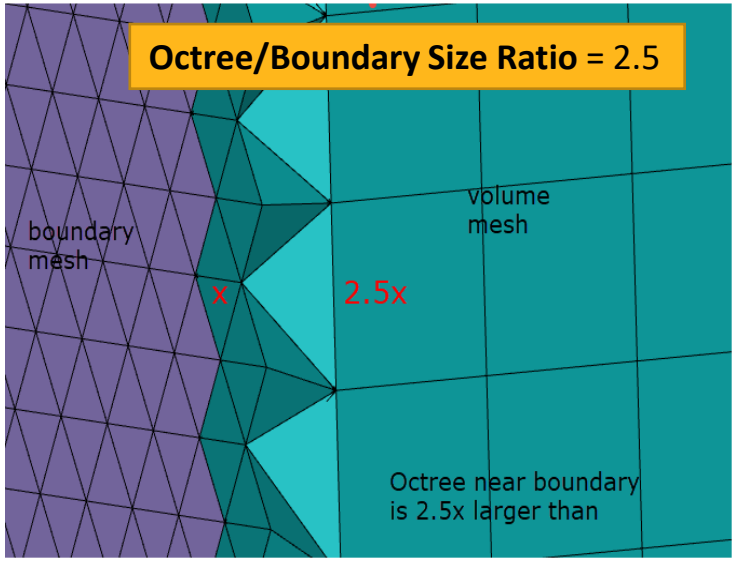
Quality Improve Limit	0.04
☑ Enable Parallel Meshing for Fluids	
☐ Save Mesh	
☐ Enable Region Settings	
⊖ Advanced Options	
Avoid 1/8th Octree Transition in Hexcore Region	No
Octree Peel Layers	1
Use Size Field	No
Octree/Boundary Size Ratio	2.5
Buffer Layers	2
Tet/Poly Growth Rate	1.4
Conformal Prism Split	No

Total cell count: 1494872
11.5 minutes

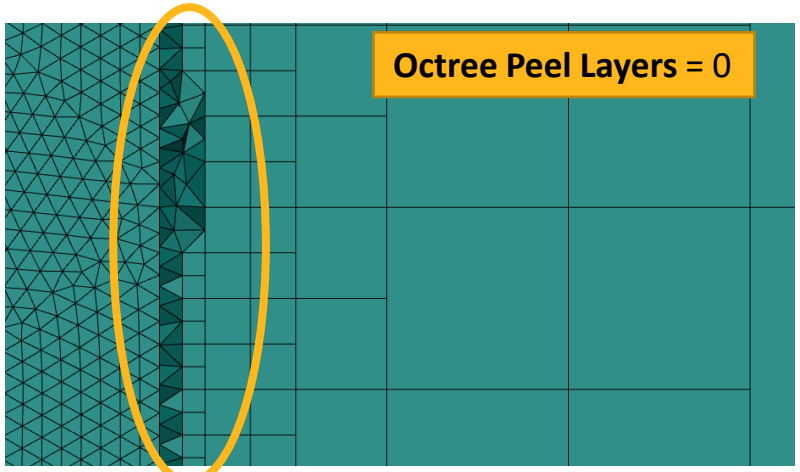
```
-----------------------------
| Virtual Mem Usage (GB)    |
| Current       Peak        |
-----------------------------
| 13.1558       19.2074     |
-----------------------------
```

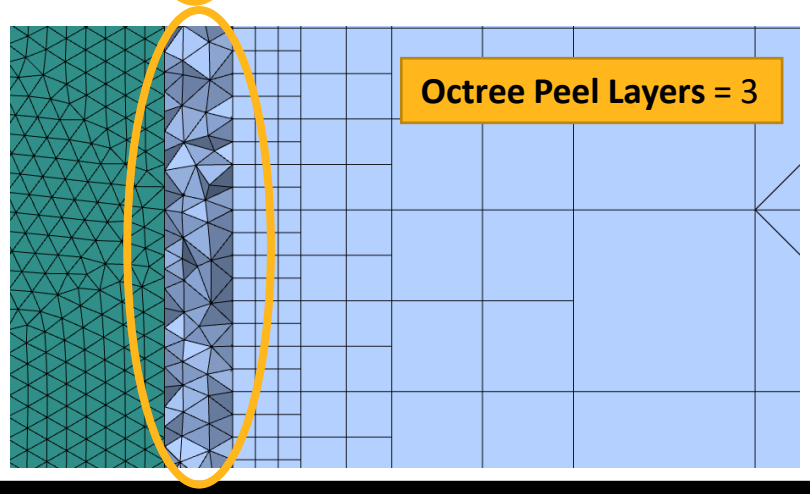
Fill Without Size Field Parameters

Octree/Boundary Size Ratio



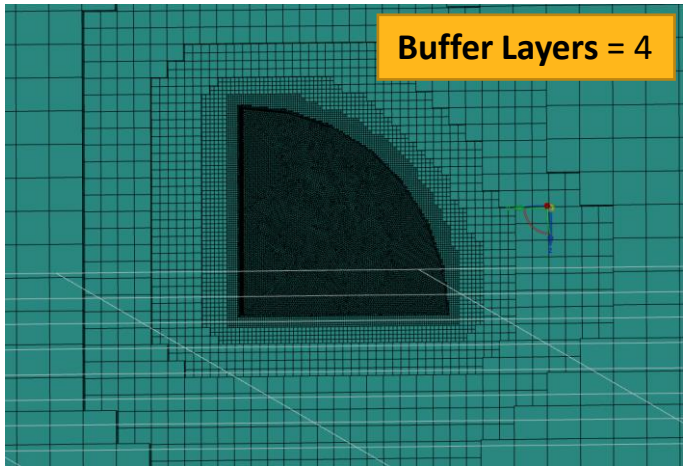
Octree Peel Layers

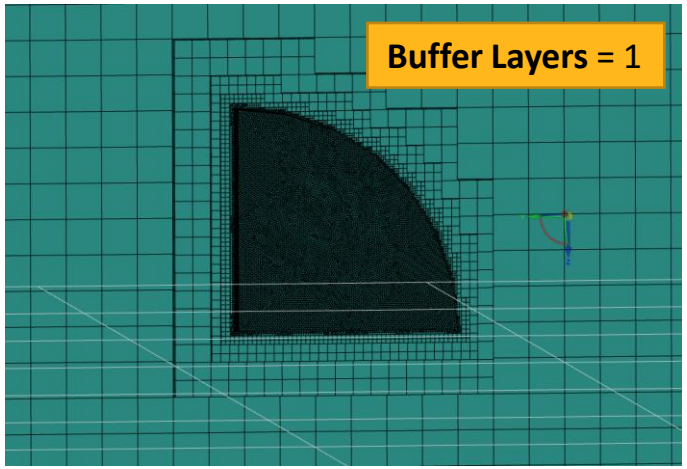




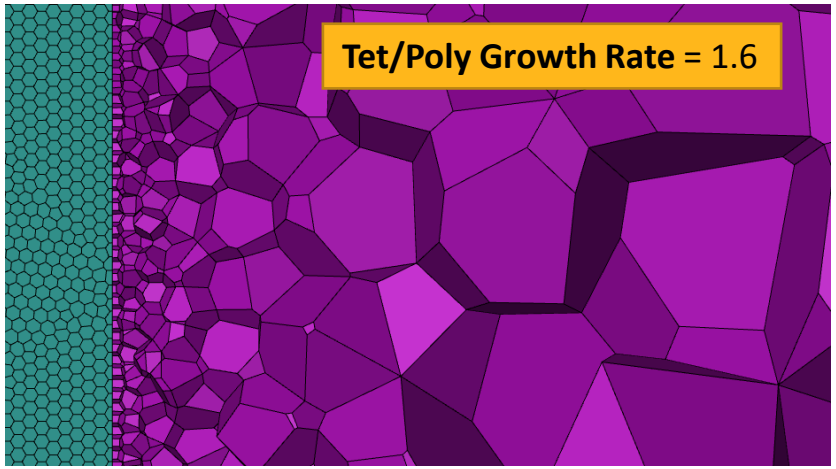
Fill Without Size Field Parameters

Buffer Layers





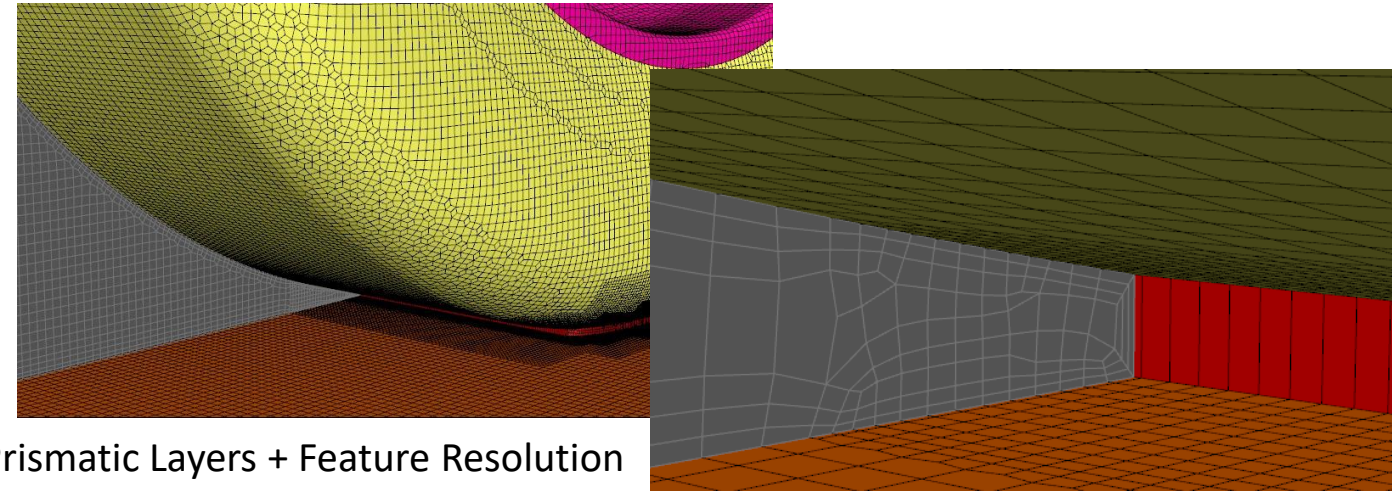
Tet/Poly Growth Rate



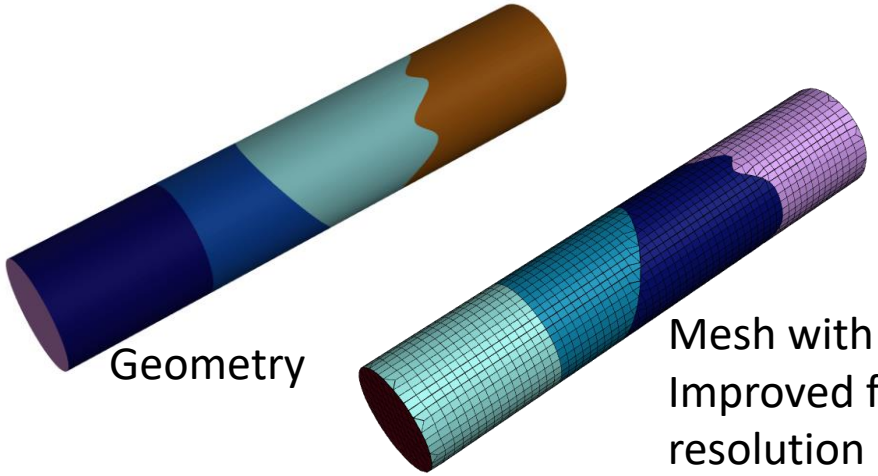
Rapid Octree Meshing

Rapid Octree Meshing Improvements

- Core Functionality
  - *Projection angle correction for Prismatic Layers*
    - Significantly reduces bad cell count
    - Improved splitting parameter computation
    - Additional prism bunching modes (e.g. First Layer Aspect Ratio)
  - *Improved Geometry Resolution*
    - Improved Mesh Optimization Scheme



Prismatic Layers + Feature Resolution



Rapid Octree: Projection Angle Correction For Prisms

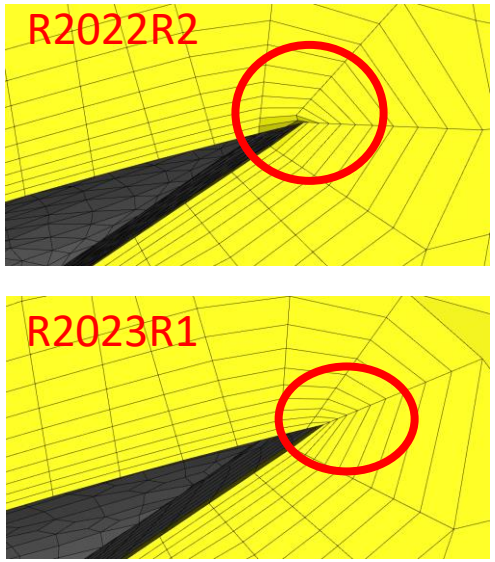
Example benchmark case

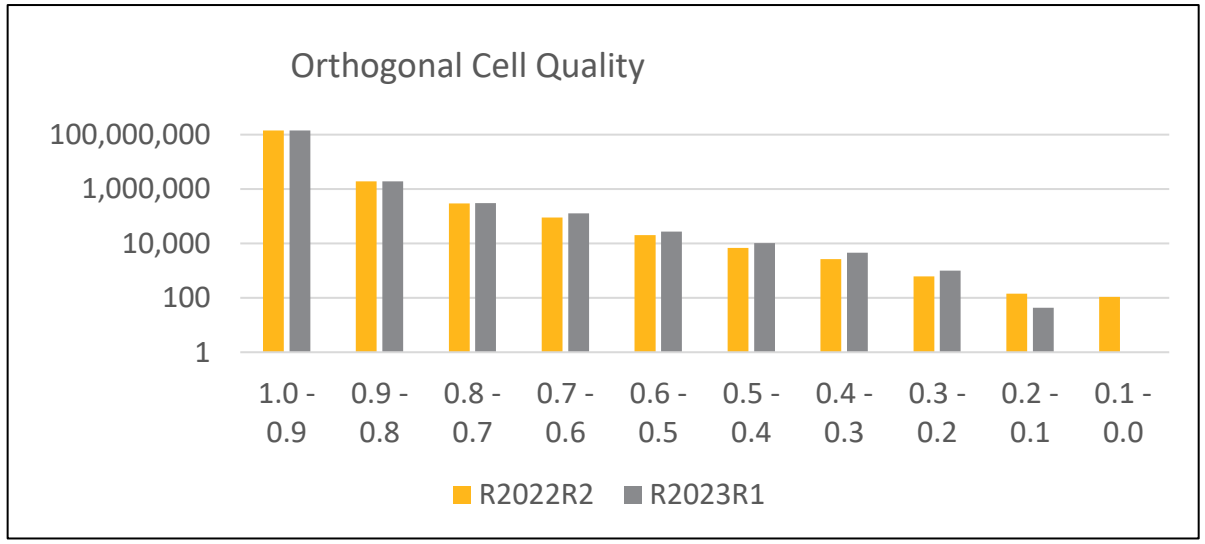
- 3 Prismatic Layers
- Non-Uniform surface sizing + Refinement Regions
- Improved Geometry Resolution Active
- 142M Cells

- 2022 R2
  - Minimum Orthogonal Quality 2.52e-02
  - After RO, 7 bad quality cells (< 0.01) needed post-fix with Auto Node Move
- 2023 R1
  - Minimum Orthogonal Quality 1.18e-01
  - No post-fix by Auto Node Move required

Significant improvement in mesh quality in 2023 R1

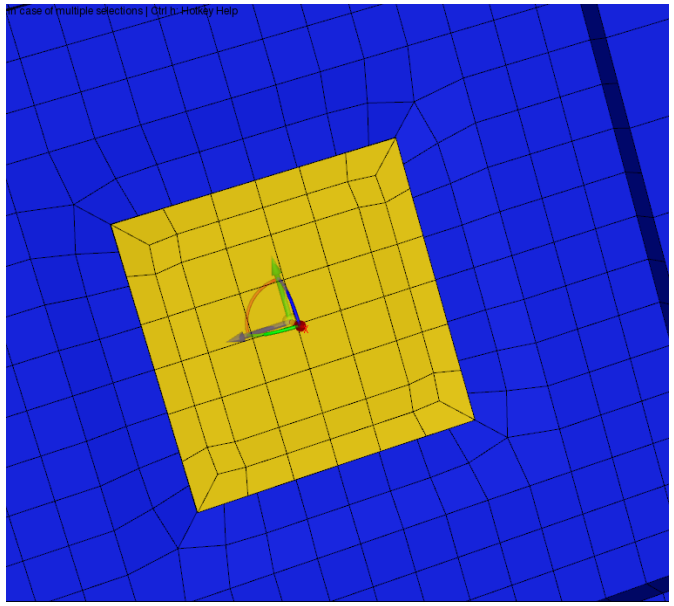
Angle Correction in Prismatic Layers



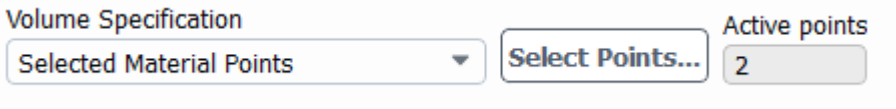


Rapid Octree – Generic Material Point Option

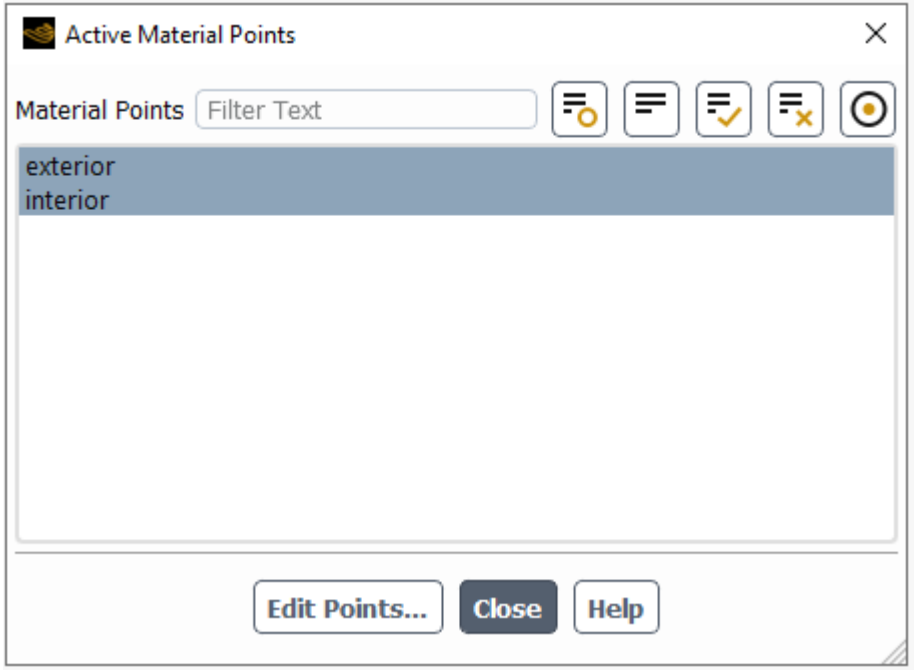
- Specify Material Points to enable multivolume meshing of dirty CAD input
- Requires large leakages to be closed before meshing
- Results in non-conformal multivolume



New Volume Option

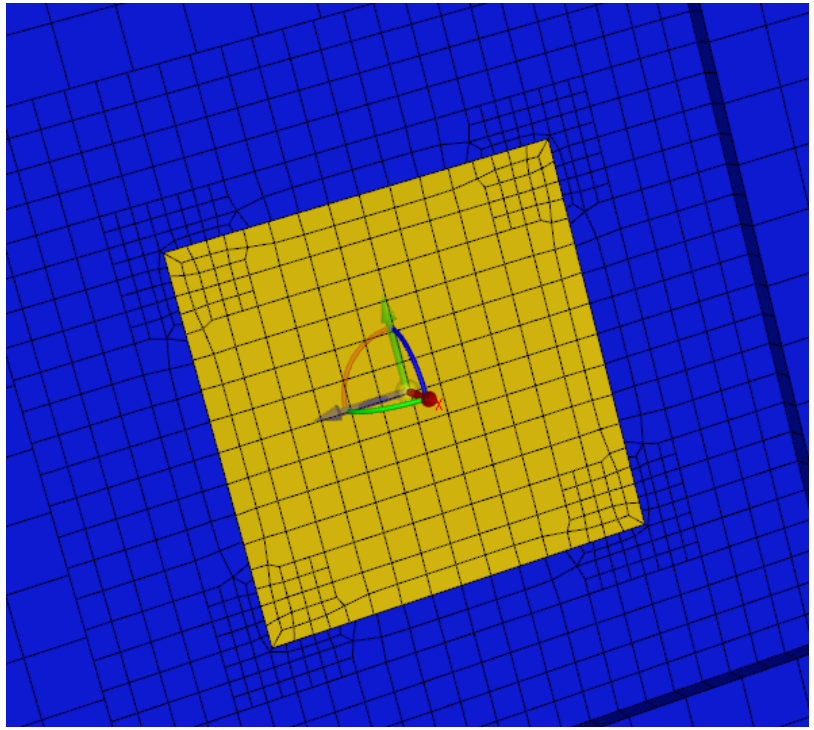


Select MPT Panel

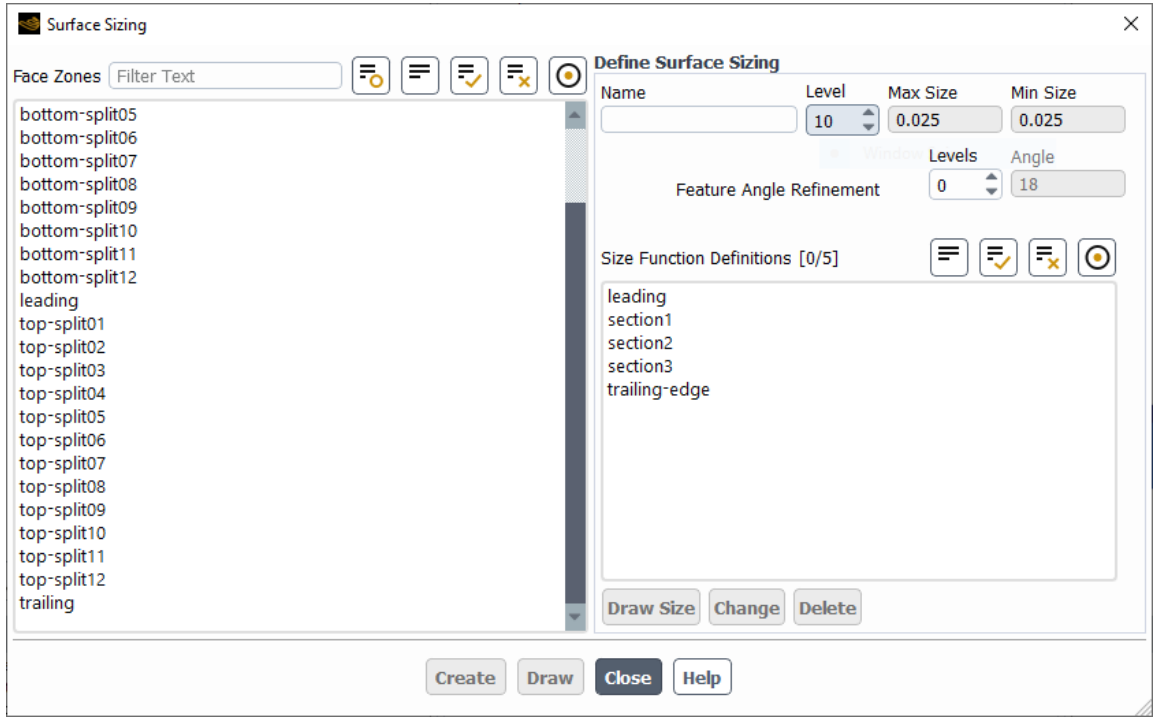


Rapid Octree – Angular Refinements for Boundary Sizing

- Angular refinements can now be set for each boundary (previously limited to a single global value)
- Enables finer adjustments of the mesh density



Extended Boundary sizing Panel



Ansys