DRD TECHNOLOGY

Ansys Fluids Update 2025 R2

Adam Kroll – Manager of Fluids Engineering

August 21, 2025

Fluent

Explanation of Symbols

Icons in the right top corner indicate in which solver or user interface the described capability is introduced or updated:

CPU solver 

GPU solver 

PyFluent 

Web UI 

Desktop UI 

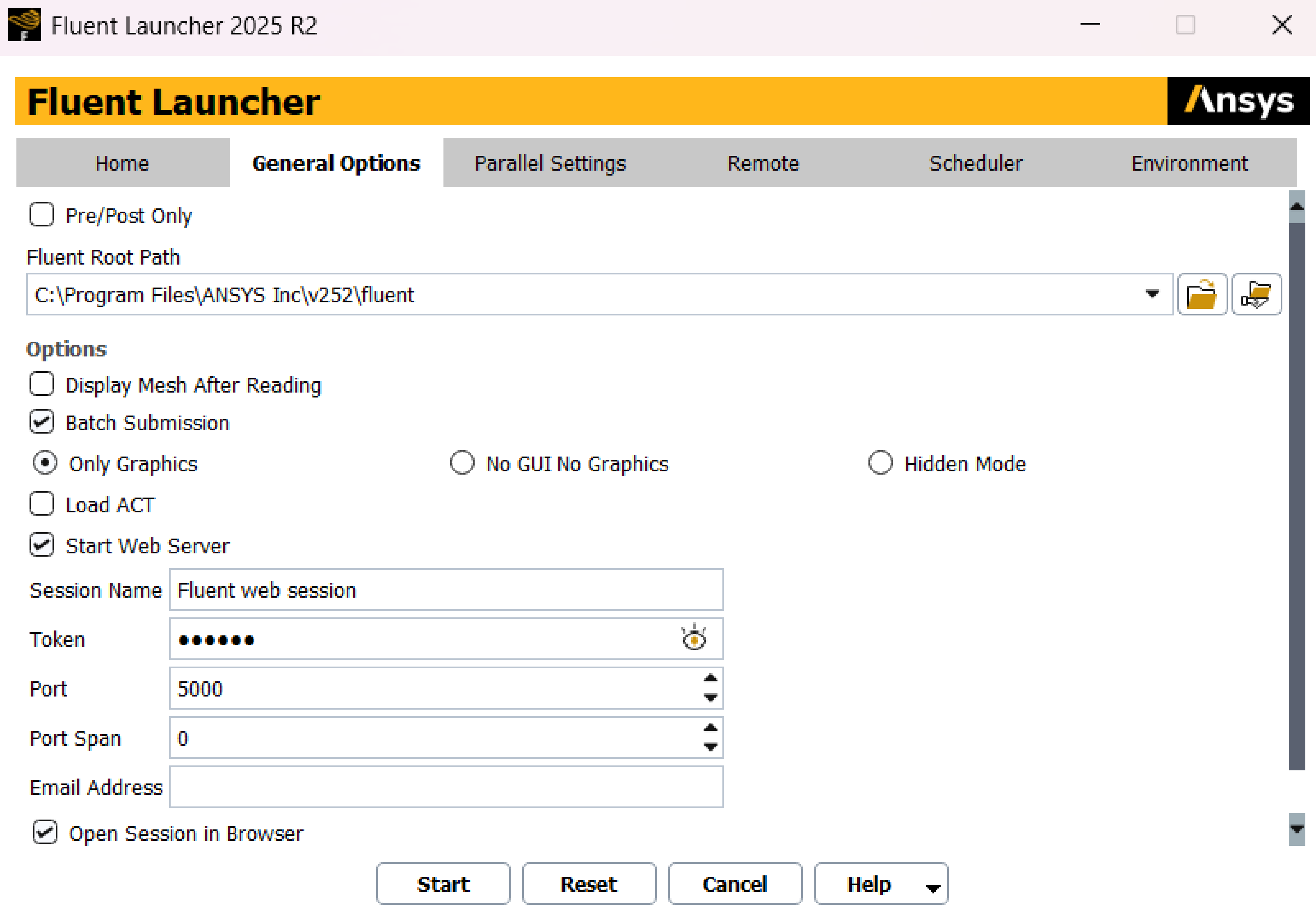
 Grey icons indicate the feature already existed in the CPU solver prior to this update

CUDA Requirements

- On Nvidia hardware, **CUDA 12.8** must be explicitly installed at the system level to use **Expressions** or **Python-Based UDFs [β]** in Fluent 2025 R2
  - Installing the GPU driver by itself is not enough
  - Refer to the Nvidia website for download and installation instructions
  - Only GPUs that are compatible with CUDA 12.8 can be used
    - Maxwell (2014) microarchitecture and newer
  - Can enter **nvidia-smi** in Windows Command Prompt to check driver and CUDA versions
- No additional installation needed for AMD GPUs on Linux

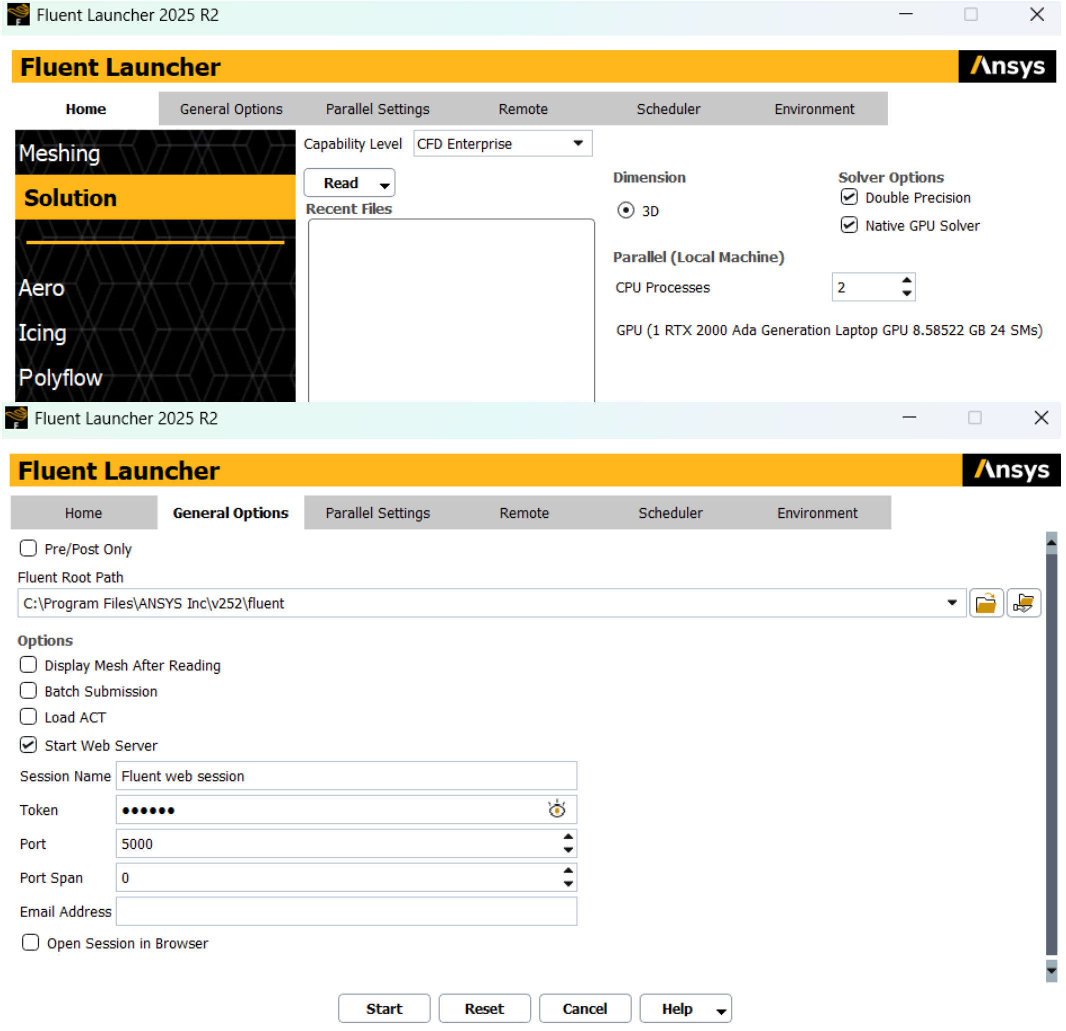
Batch Job Submission Option – Launcher Updates

- Submit batch jobs from launcher
  - Only Graphics (-gu)
  - No GUI No Graphics (-g)
  - Hidden Mode (-hidden)
    - Full GUI in “minimized” mode
- “Open Session in Browser” option to launch Fluent in browser on local machine



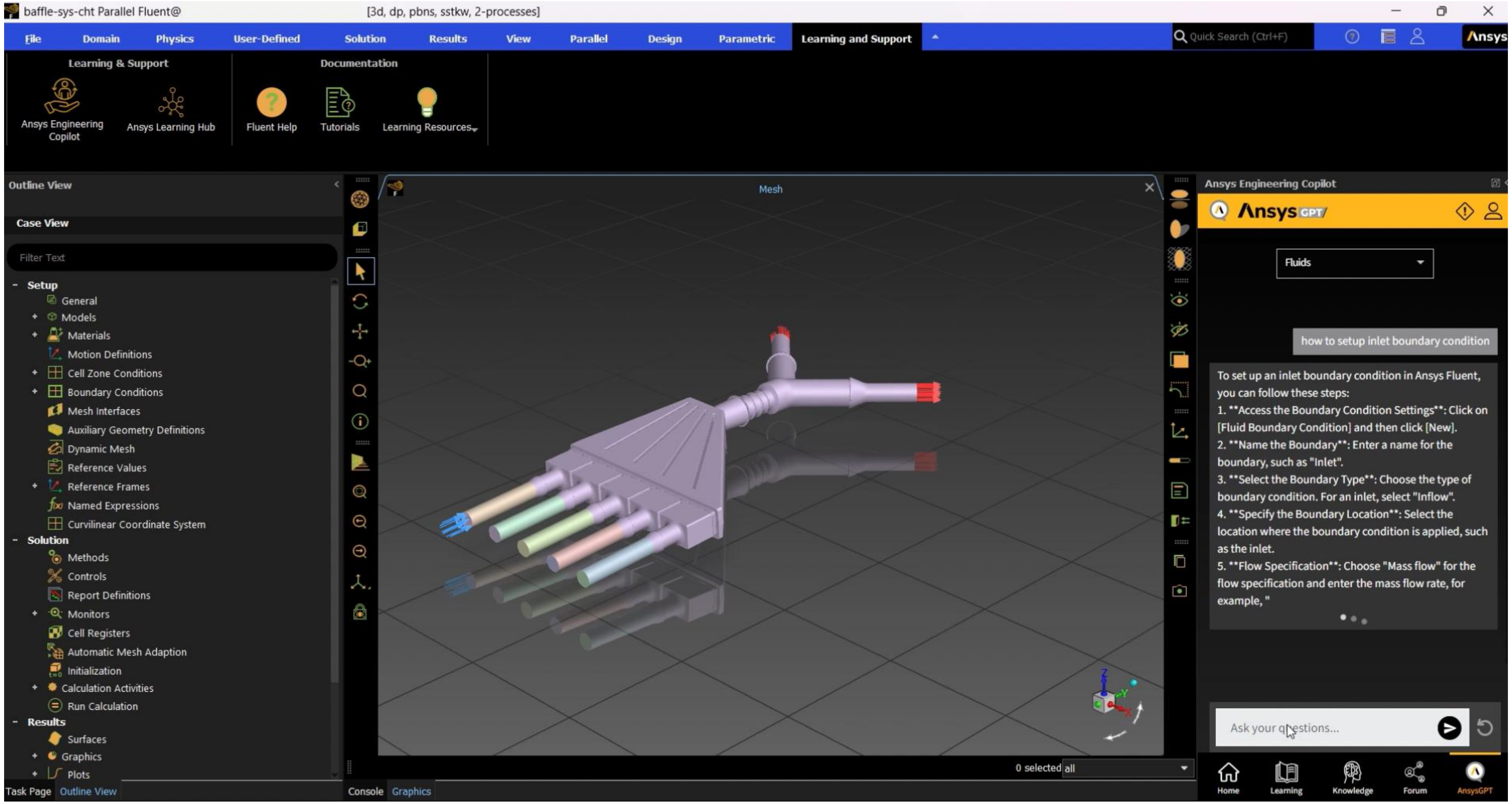
Web User Interface for GPU Solver

- Server can be started in GUI, TUI, PyFluent, and startup command line argument
- Access to all settings supported by the GPU solver when calculation is stopped
  - Cannot edit settings during calculation
- Post-processing during calculation is possible but may be slow
- Direct GPU post-processing (lite mode) has no effect on Web User Interface
  - Data must be transferred from GPU to CPU over the network to the client for visualization
  - Main speed limitations:
    - Bandwidth
    - Visualization speed of the client browser



Ansys Engineering Copilot* Integration

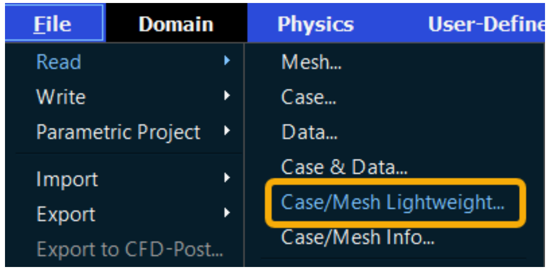
- Side panel with direct access to:
  - Ansys Learning Hub
  - Knowledge Materials
  - AIS Forum
  - AnsysGPT
- Limited to Windows in 2025 R2
- Requires login with Ansys Account



*Ansys AI+ license required for this feature. The license is free for user and can be downloaded from Ansys license center

Visualization of Surface Mesh in Lightweight Case

- Reading case or mesh in Lightweight mode loads only the settings by default

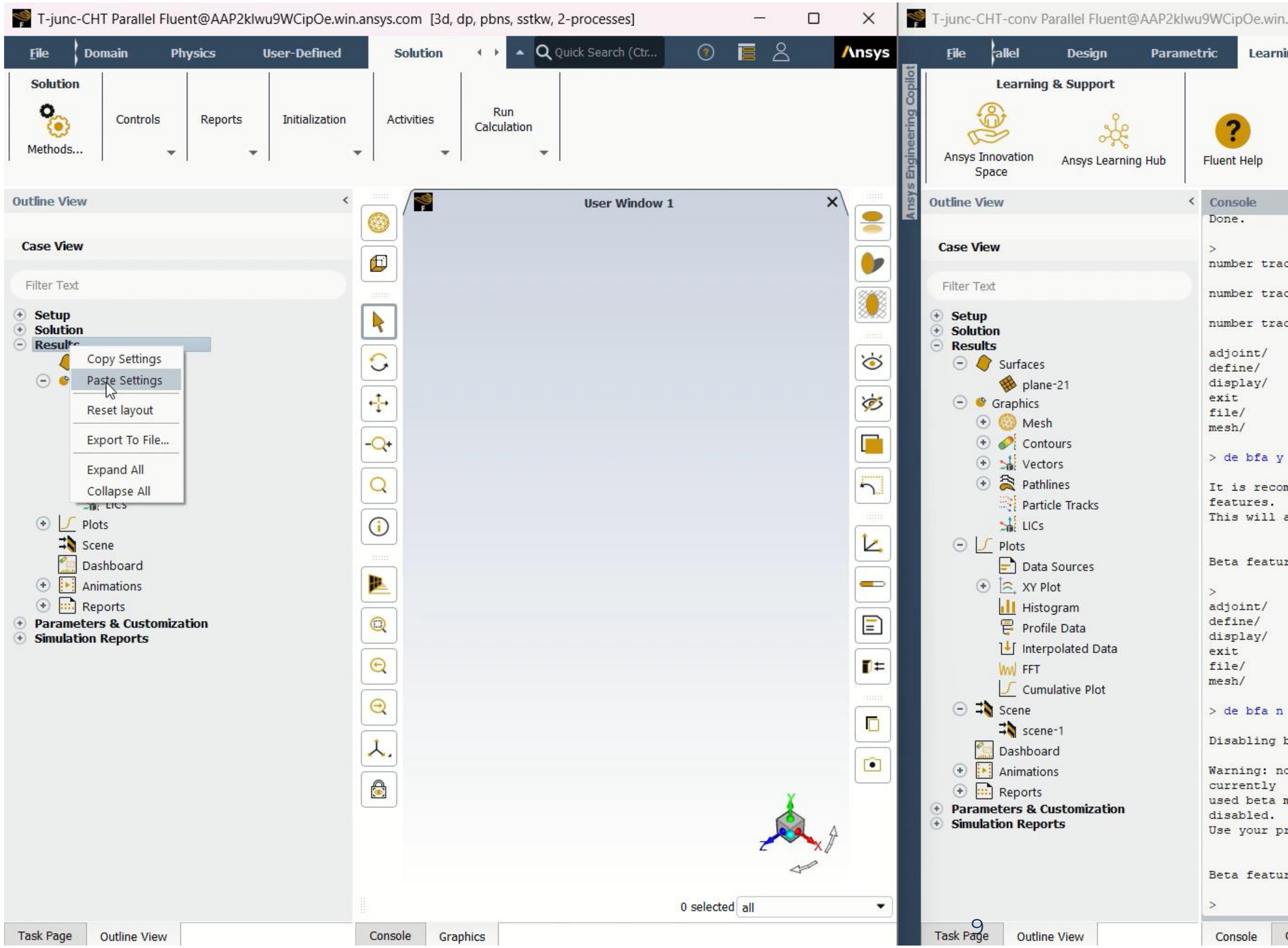


- Only when displaying the mesh, Fluent reads the surface mesh
  - Useful for interactive setup of a case with graphics support
  - Needs the usual memory for the surface mesh, only the volume mesh is not loaded
    - Best for cases with a very low number of surface elements compared to volume elements

Transfer Case Settings

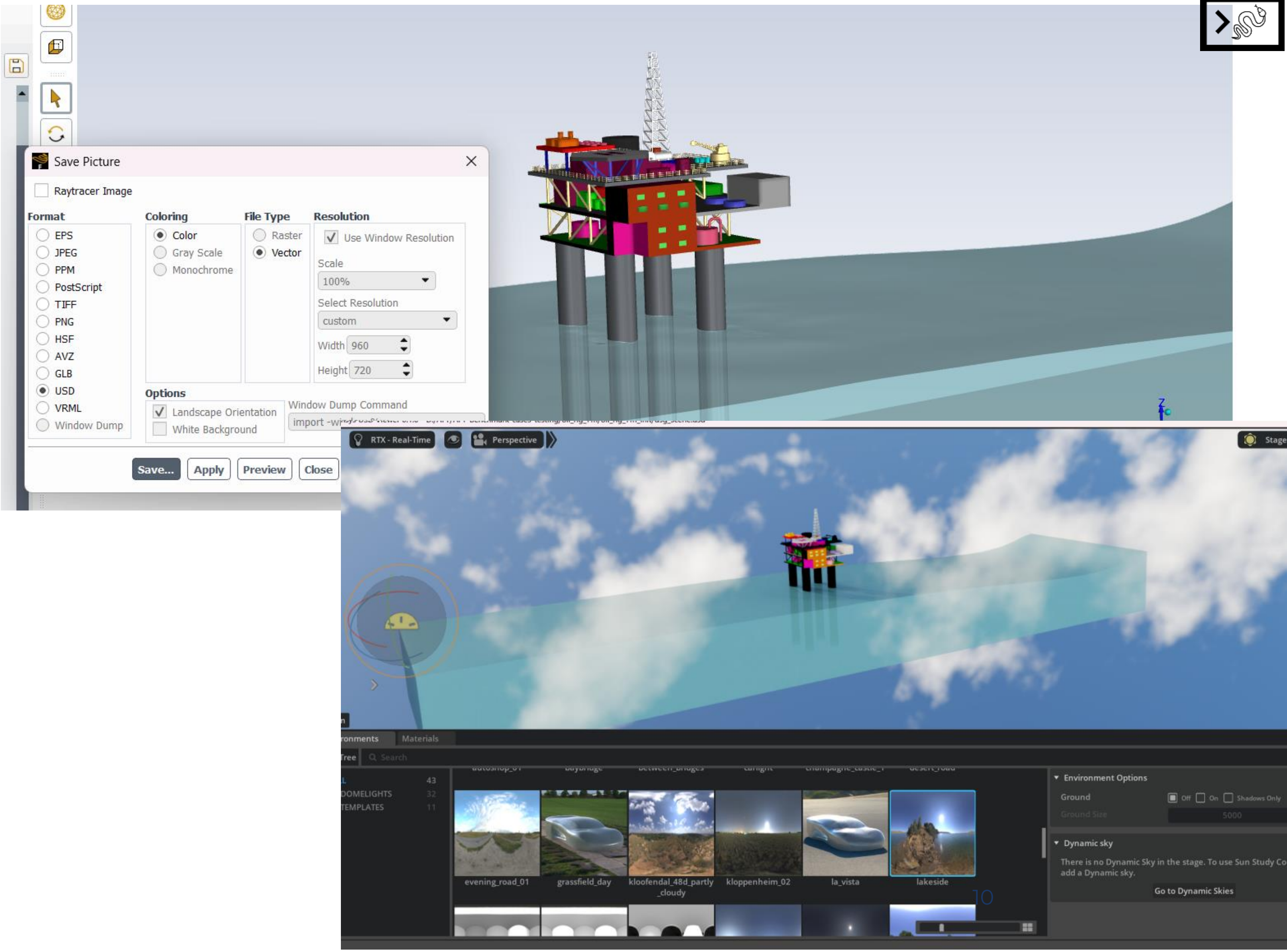
- RMB context menu on Results Outline node to copy and paste settings
  - Works on all hierarchy levels
- Drag from one Fluent session into another Fluent session

	Drag-Drop	Copy-Paste
Setup Node	n/a yet	Beta 2025 R2
Solution Node	Release 2024 R2	Beta 2025 R2
Results Node	Release 2024 R2	Release 2025R2



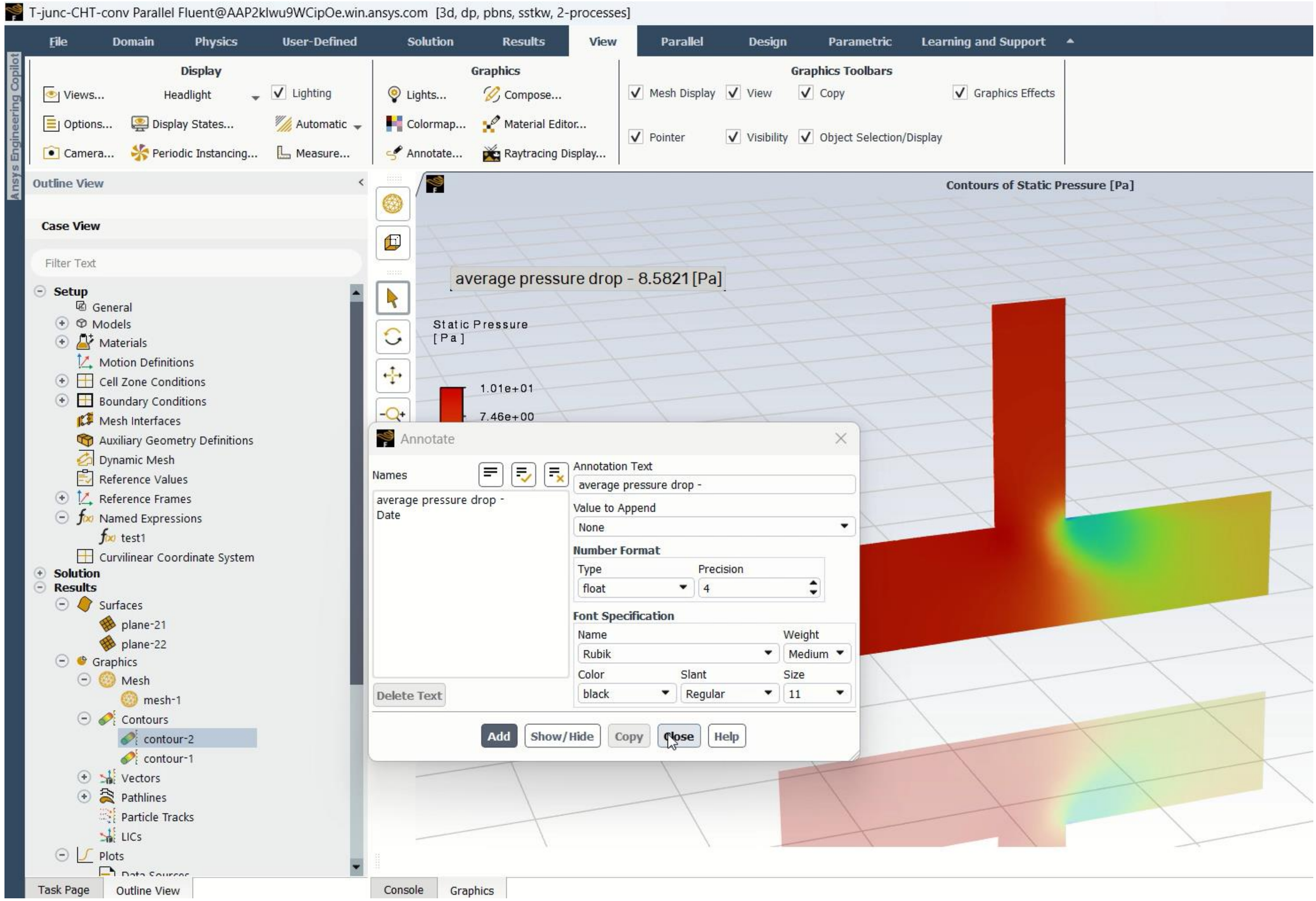
Universal Scene Description (USD) Export

- Export in USD format from Save Picture window
  - Creates a directory structure and a *.usd file
- USD is aimed at visualization and can be opened in many third-party applications like the Omniverse Viewer or Blender
- No transient export available



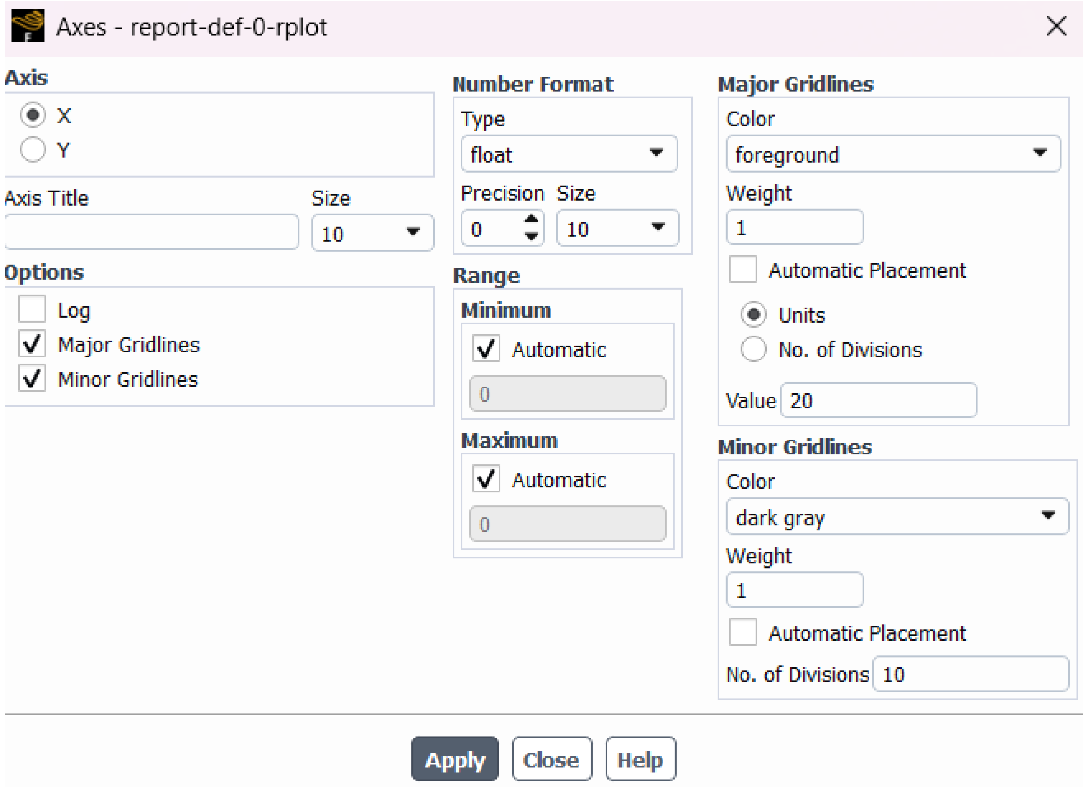
Annotations

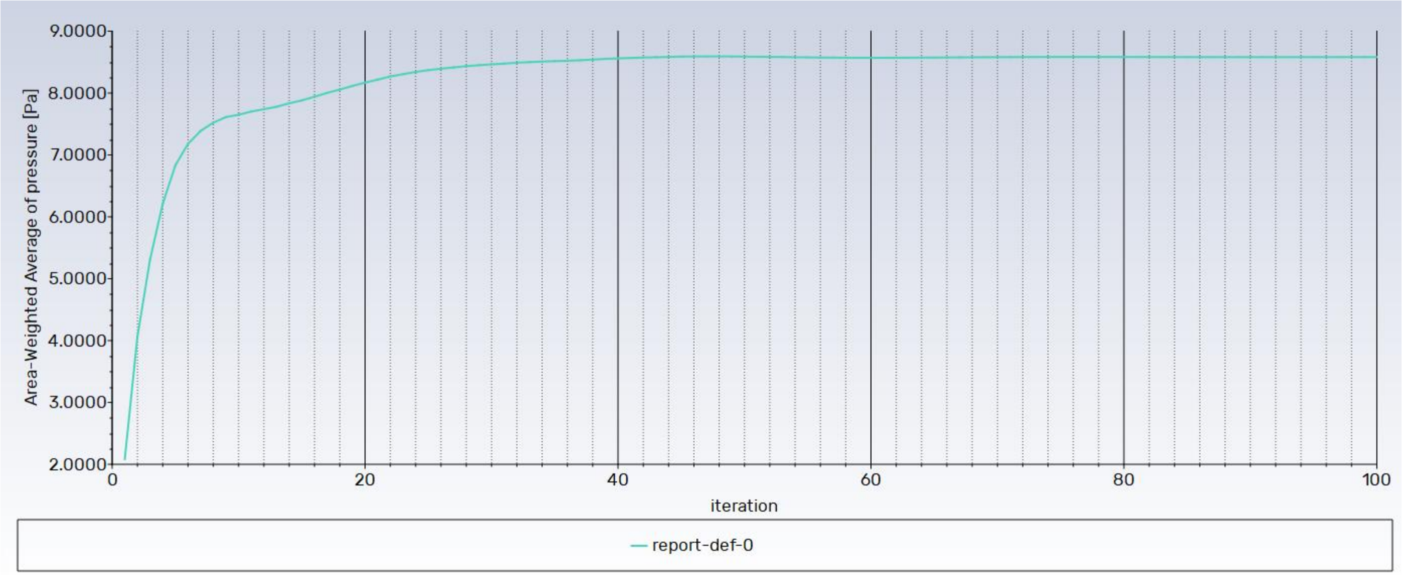
- Interactive and user-friendly annotations.
- Append values of certain quantities to the text: Date, Time, Version, Iterations, Timesteps, Flowtime, Named Expressions
- Save Annotations as Objects with graphics objects (and in case/data files)
- Use expressions in annotations during animation
- Annotation can be saved with animations of any type



Qt Plot Settings

- Separate Auto setting for the Minimum and Maximum axis values
- Option for distance between gridlines

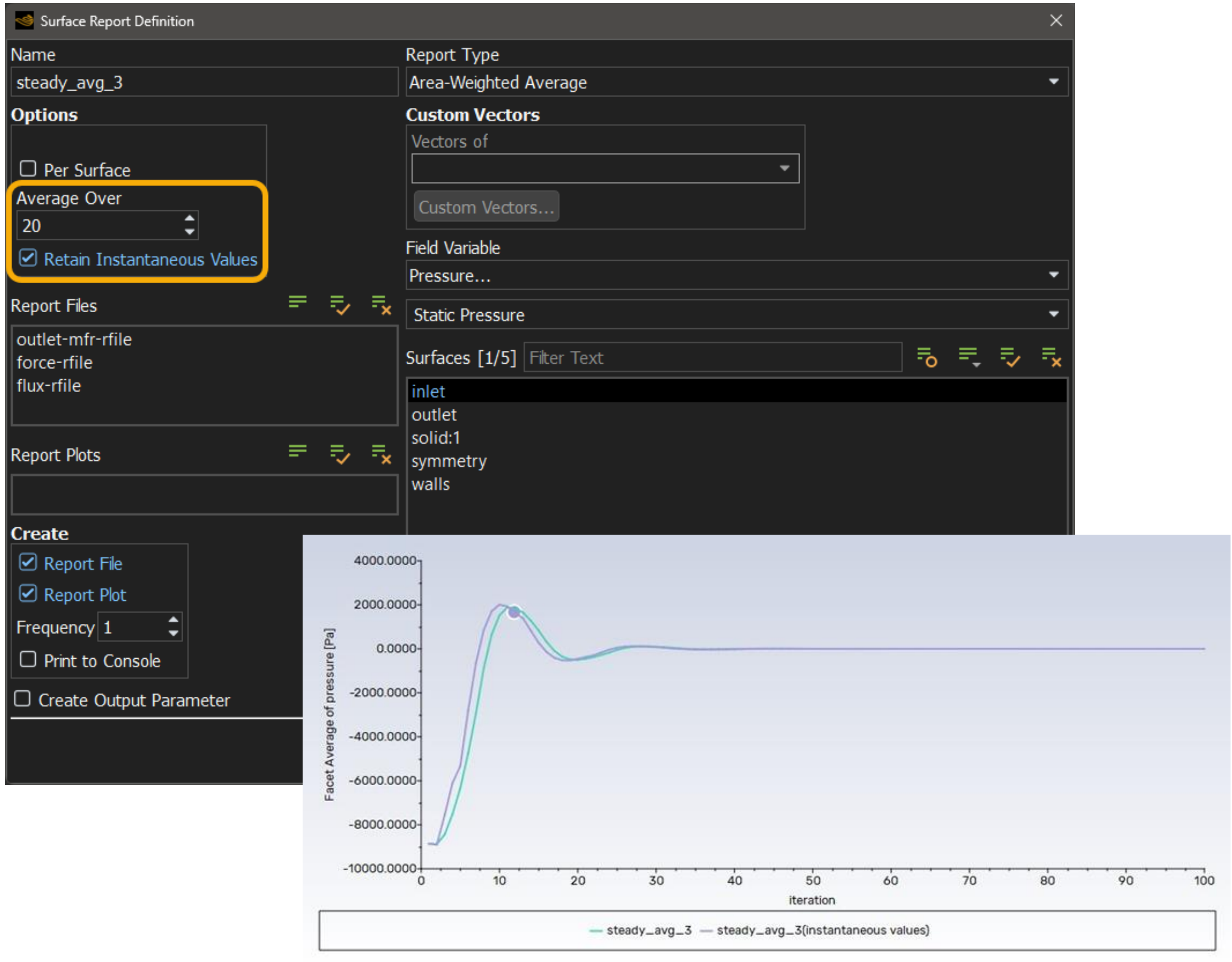




GPU Post Processing

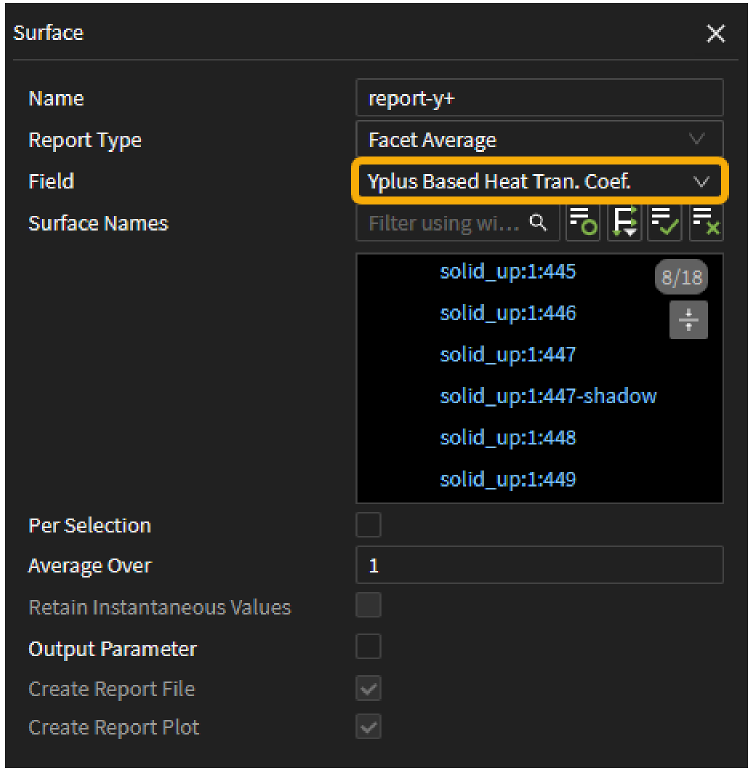
Average Over Monitors

- Write, plot, and print the moving average of report definitions



Y⁺-Based HTC Monitors

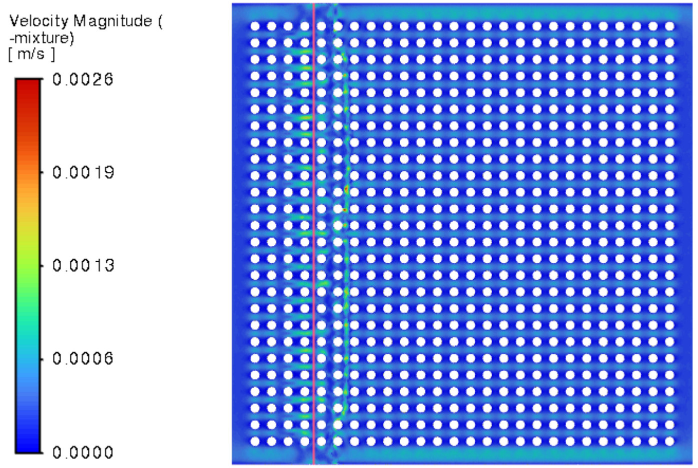
- Write, plot, and print Y⁺-based heat transfer coefficient

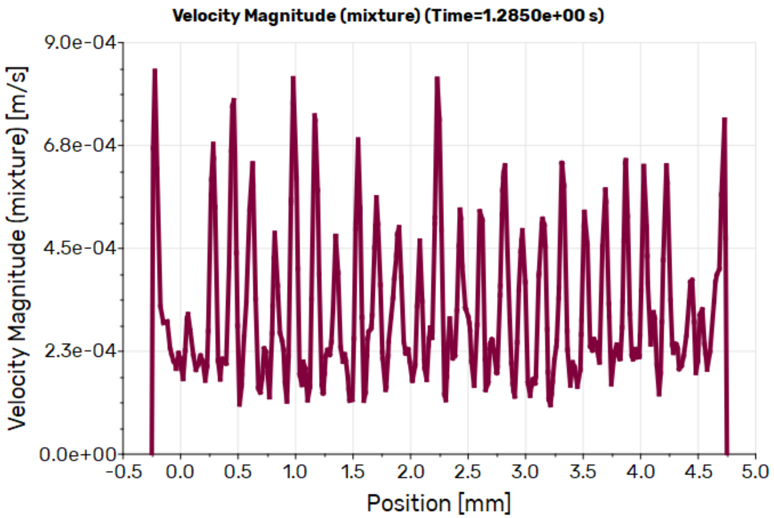


GPU Direct Post Processing [β]

Line Surfaces and XY Plots

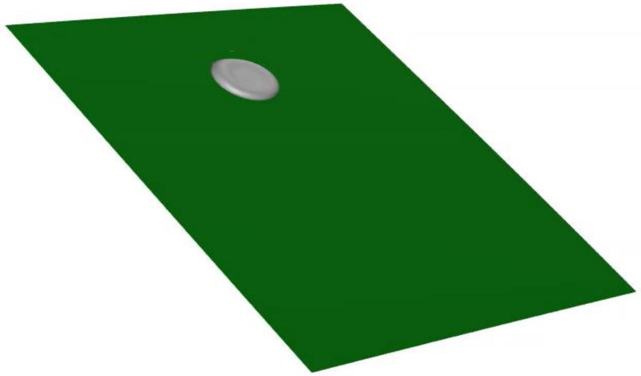
- Create, display, and evaluate results on surfaces of the type line in lite mode

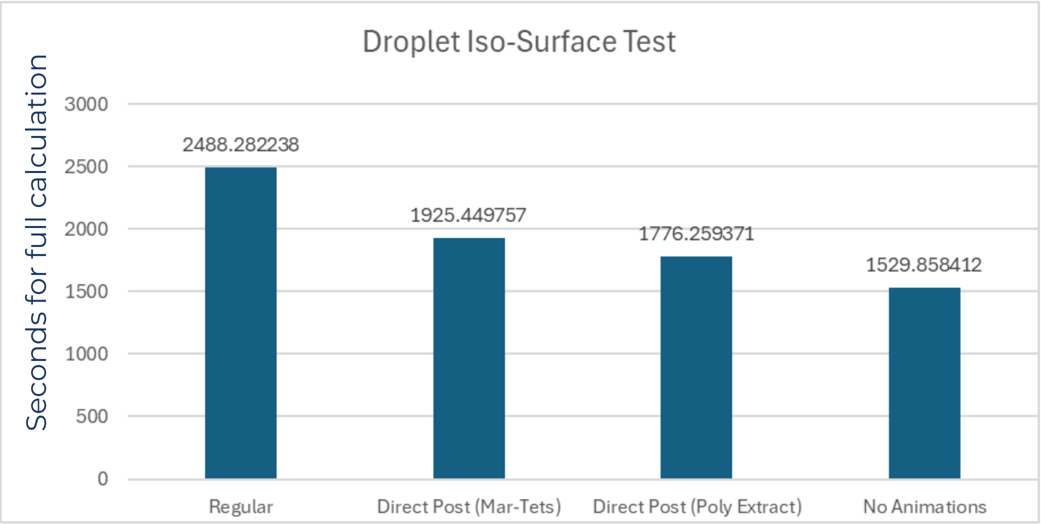




Iso-surfaces

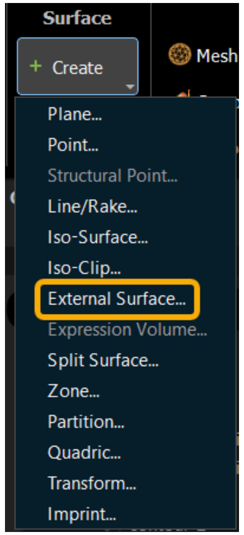
- Animations with iso-surfaces possible with GPU calculations

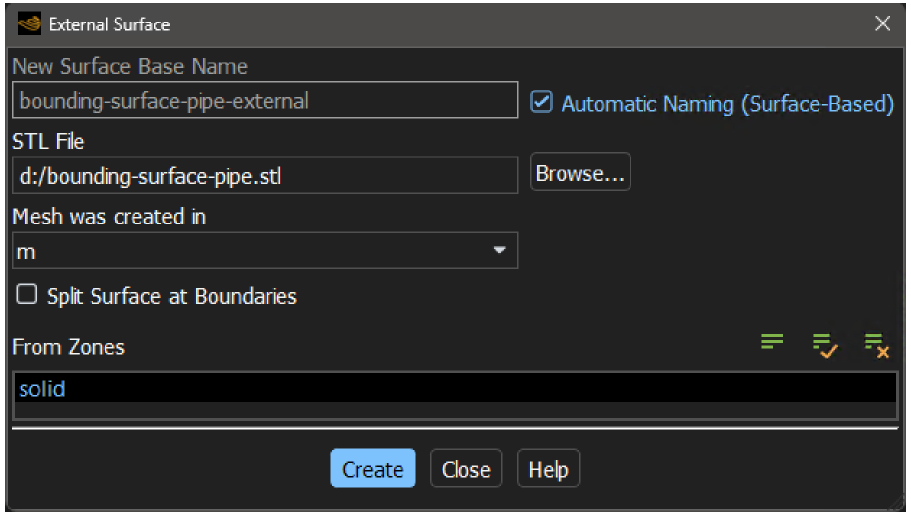




External Surfaces [β]

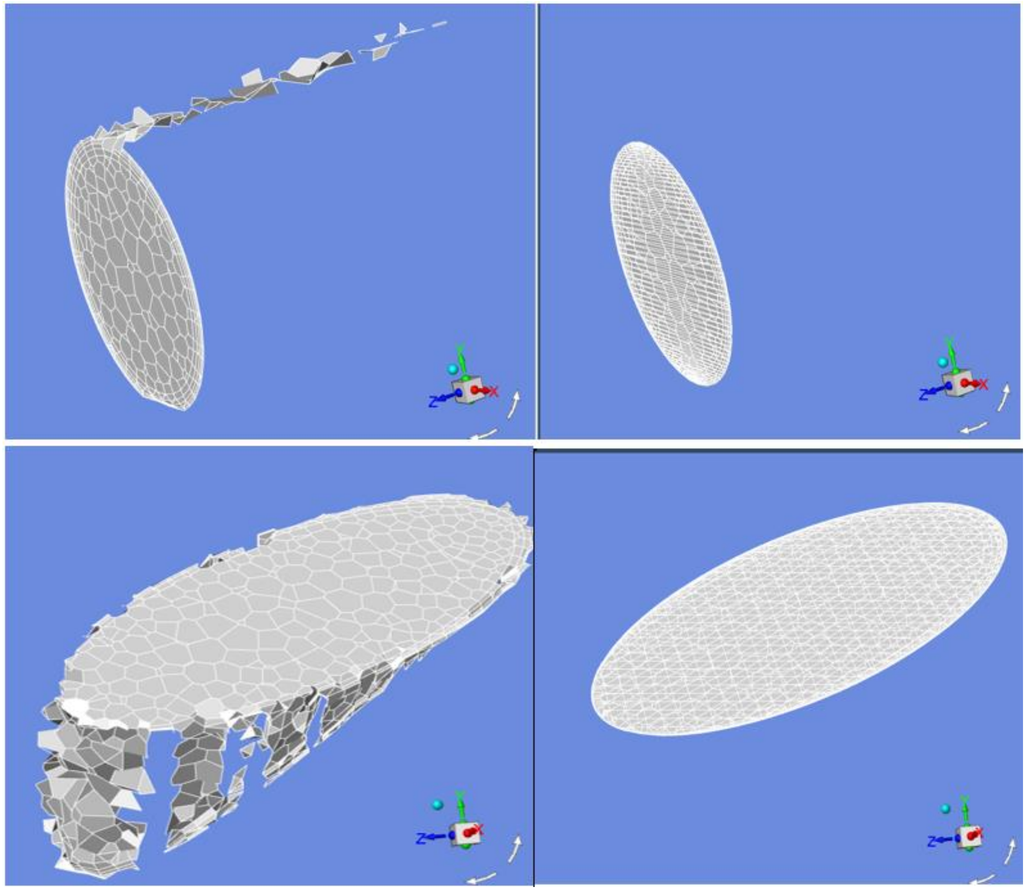
- New method to import surfaces
  - Useful for importing post-processing surfaces
  - Supports only STL format
  - Must intersect the volume mesh
    - For external reference geometry or mesh morphing use surface type Imprint





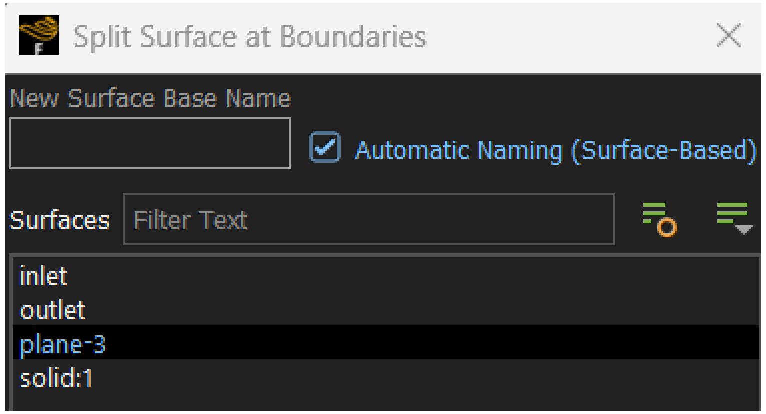
Old Imprint functionality

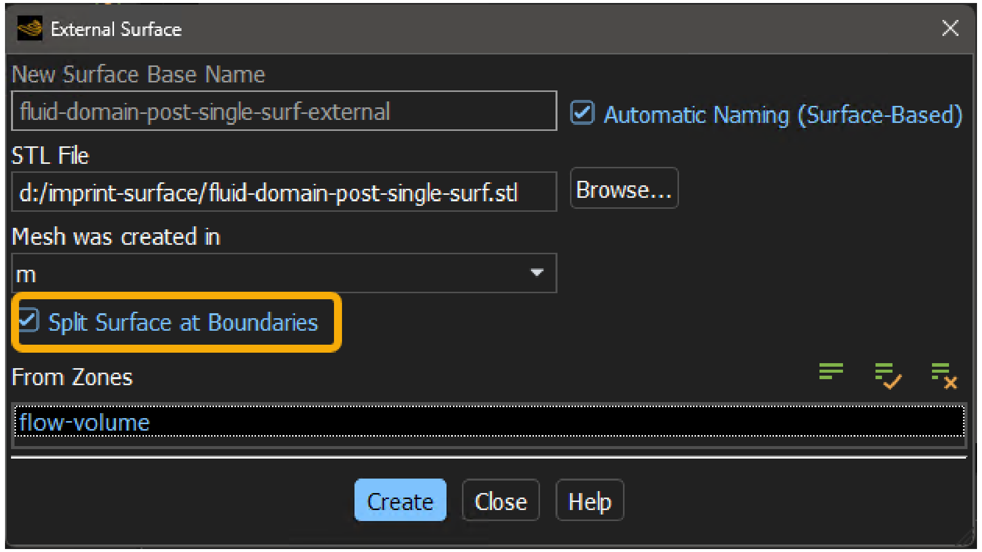
New External surface functionality

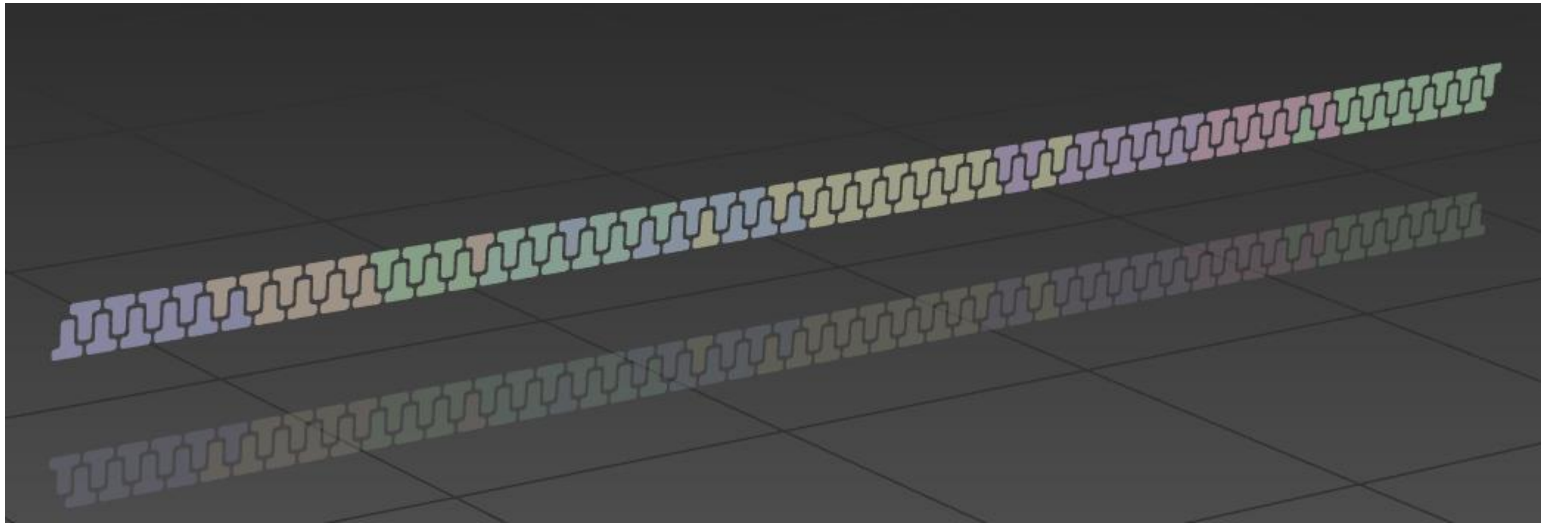


Split External Surface [β]

- External surface can be split into multiple entities for separate processing
  - Not available in lite mode

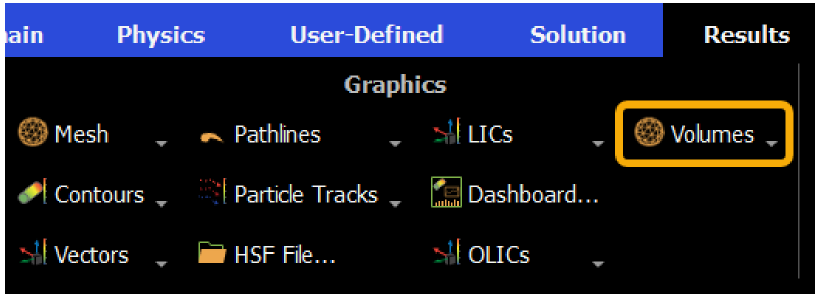


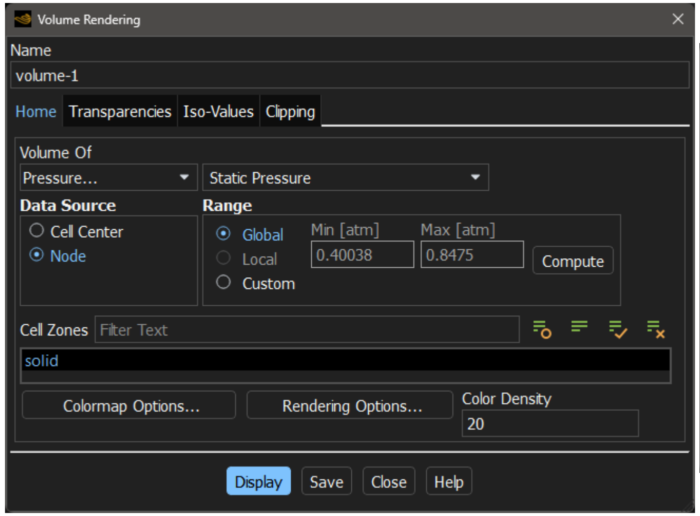


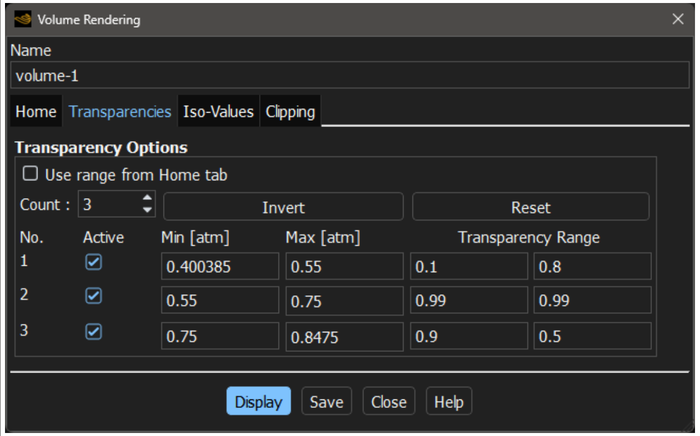


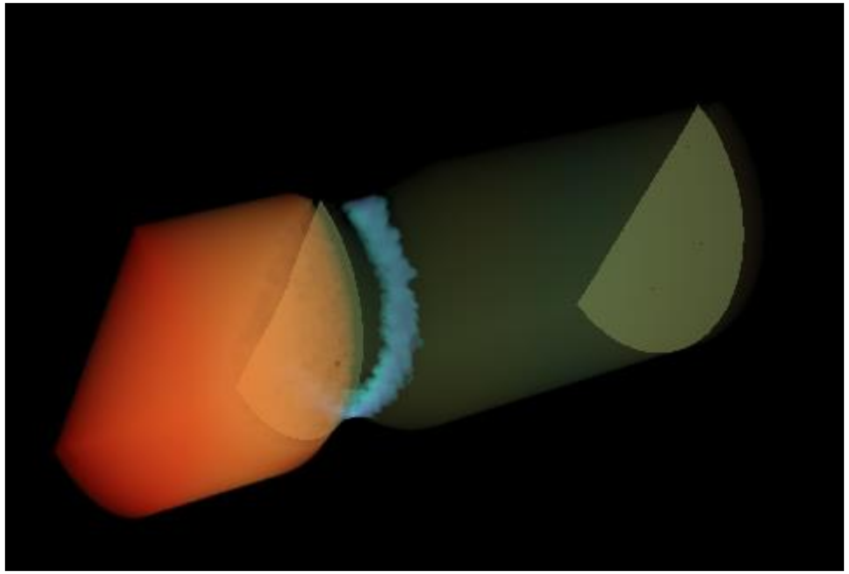
Volume Rendering [β]

- Only available from ribbon Results > Graphics > Volumes
  - Uses Raytracing workflow for volume rendering
- Setup steps:
  - Home: select variable and zone
  - Transparencies: select ranges
  - Iso-Values: optionally add iso-surfaces
  - Clipping: reduce region size
  - Home/Transparencies: fine-tune color density and transparency values



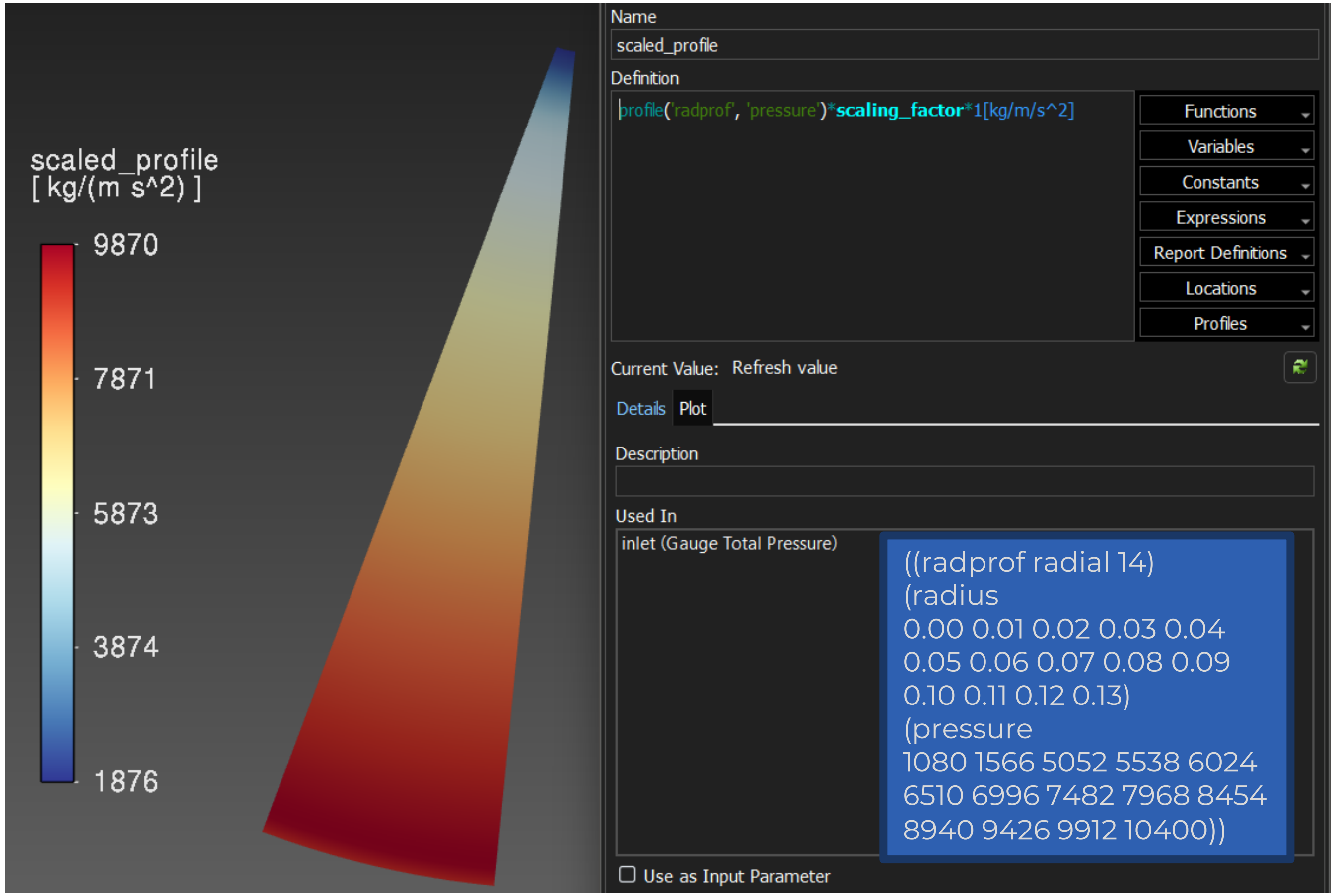






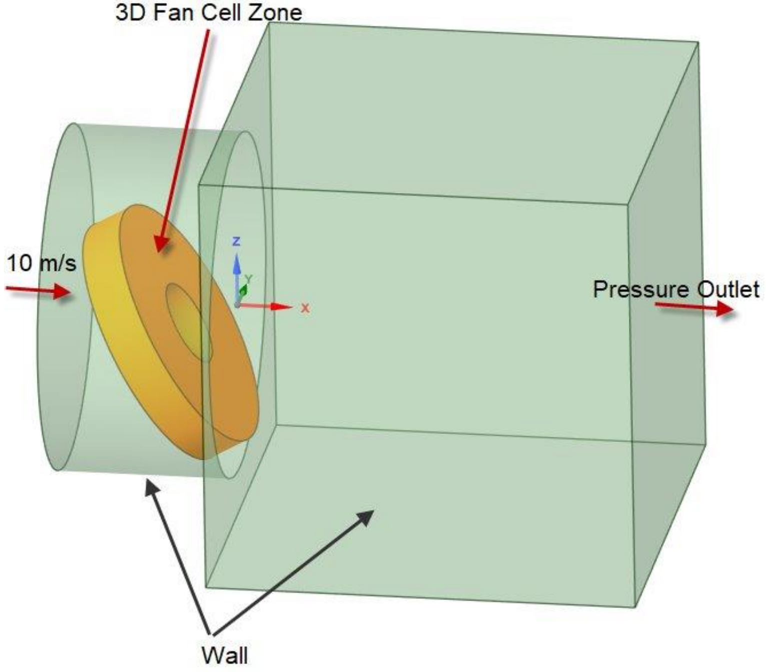
Expressions

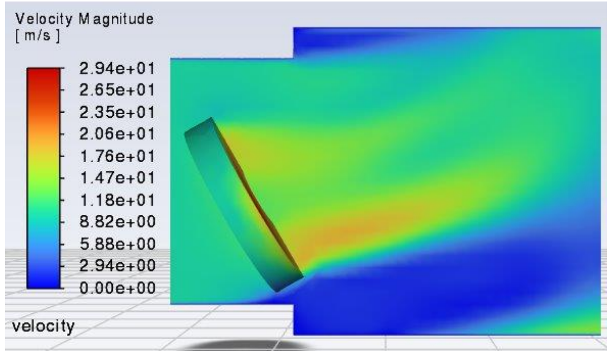
- Improved speed for import and load operations
- Expressions can now be renamed
  - All dependencies update automatically
- Radial profiles can be used in expressions
- New operators available for GPU solver:
  - `abs`
    - `AreaAve(abs(T-296[K]),['outlet'])`
  - `CountIf` and `SumIf`
    - `CountIf(P>200[Pa],['outlet'])`
  - `MassFlowAveAbs`
    - `MassFlowAveAbs(SpecificEnthalpy,['outlet'])`
  - `PressureForce`
    - `PressureForce(['wall'])`
  - `Weighted Sum`
    - `Sum(u*x+v*y+w*y,['outlet'],Weight='MassFlowRate')`



3D Fan

- 3D fan condition available on GPU
  - Same capabilities as on CPU solver
  - Applies axial, radial, and tangential sources to model all 3D effects of a fan without resolving the geometry





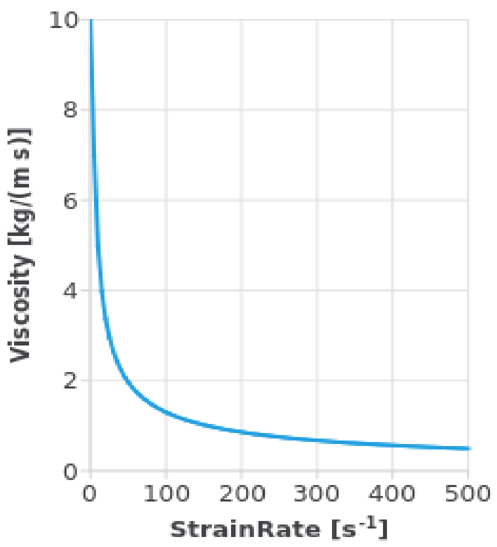
Non-Newtonian Flow

- GPU solver supports the following Non-Newtonian viscosity models:
  - Power-Law
    - n=1: Newtonian fluid
    - n<1: dilatant fluid
    - n>1: pseudo-plastic

$$\eta(\dot{\gamma}) = H(T)K(\dot{\gamma})^{n-1}$$

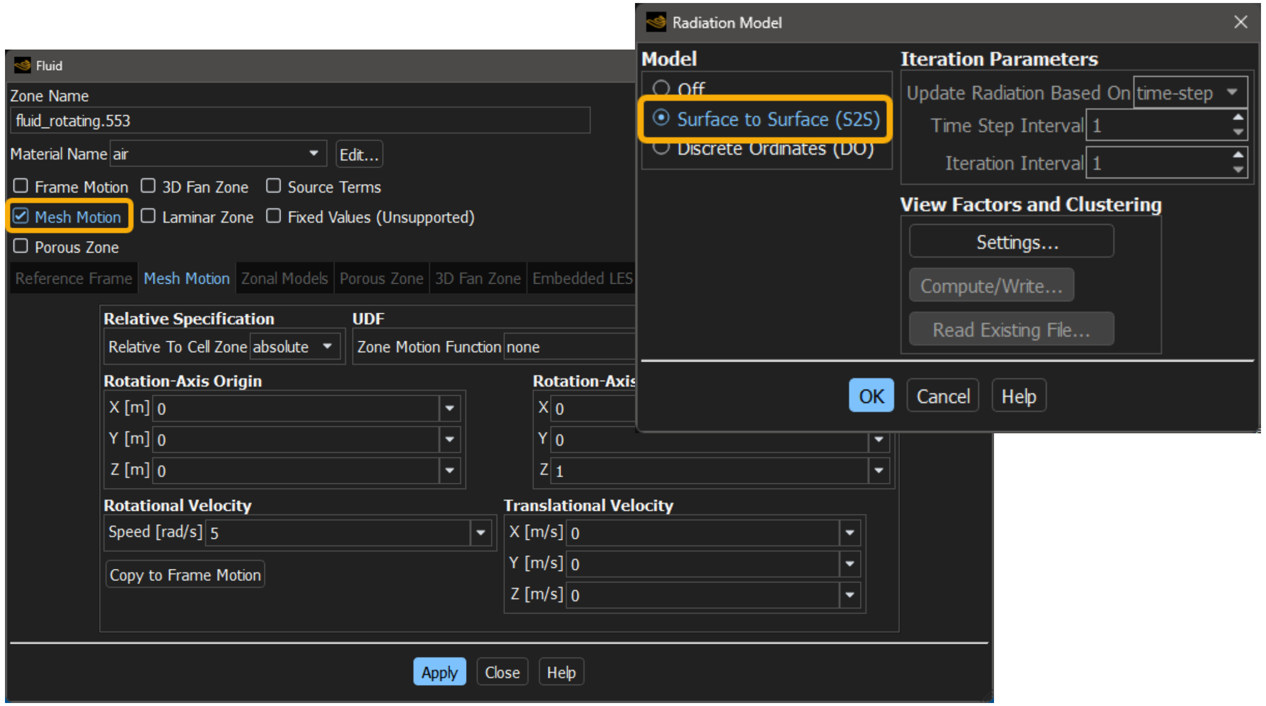
  - Carreau Law
    - Yasuda variant which is a generalized form of the Carreau Law
    - a being the shape factor (a =2 → Default Carreau)
    - New adjusted Law is also implemented in Fluent CPU

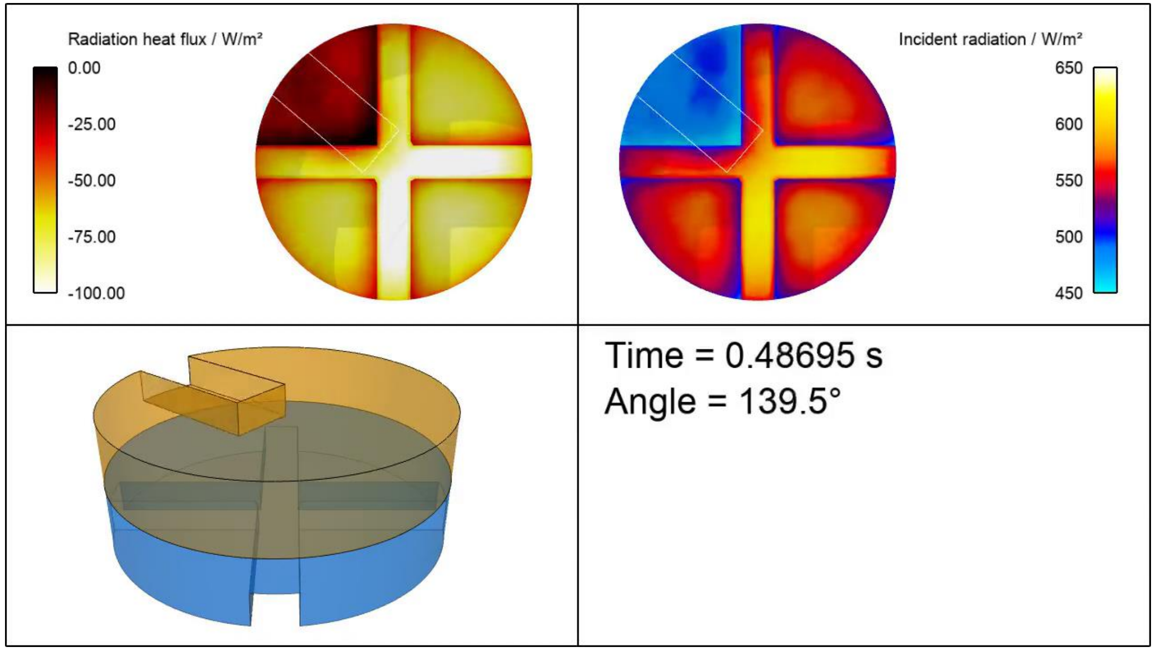
$$\eta(\dot{\gamma}) = H(T)\left(\eta_\infty + (\eta_0 - \eta_\infty)(1 + (\lambda\dot{\gamma})^{a})^{\frac{n-1}{a}}\right)$$



Surface-to-Surface Radiation with Sliding Mesh

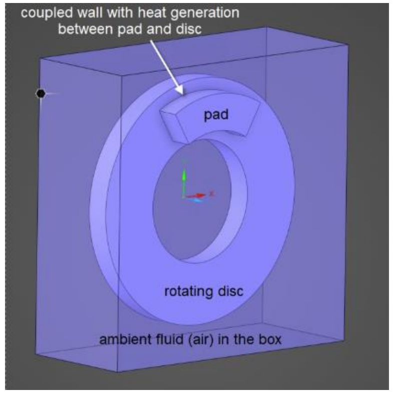
- Combine S2S with Mesh Motion for sliding mesh configurations
  - View factors are recomputed automatically every time the mesh updates
    - Calculation within the GPU solver is a lot faster than for the CPU solver, making this approach possible for many cases

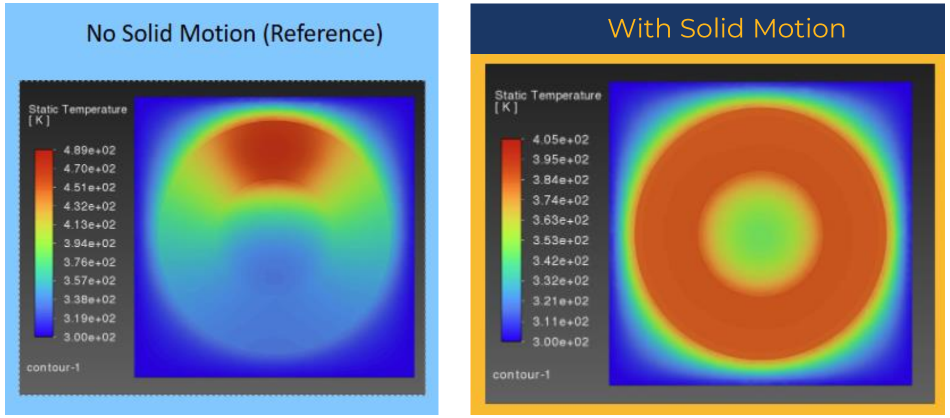


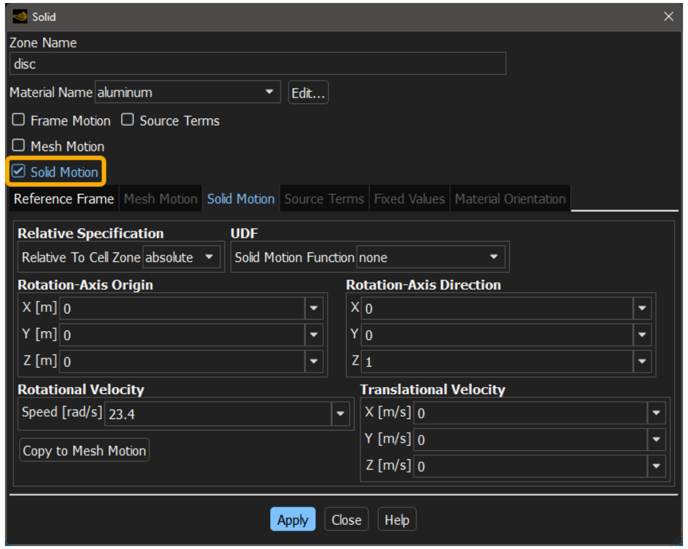


Solid Motion [β]

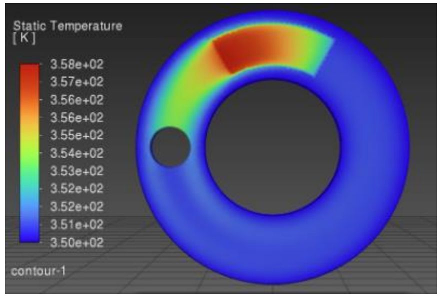
- Model energy transport due to solid motion via a convection term (no moving mesh)
- Efficient way to achieve steady state solution (neglecting holes in solid disc)
- Supports rotation, translation, boundary advection, fluid-solid contact, solid-solid contact, periodicity







If there is a hole, the true solution is inherently transient, but this model allows us to obtain a 'steady state' solution for a particular orientation. Solution is very similar with and without 'Boundary Advection'.



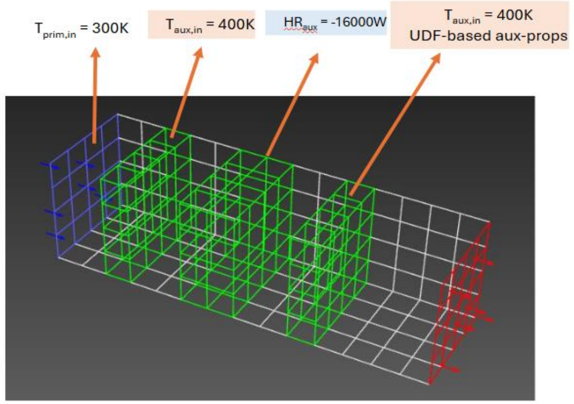
Target applications:
- Brake disc analysis

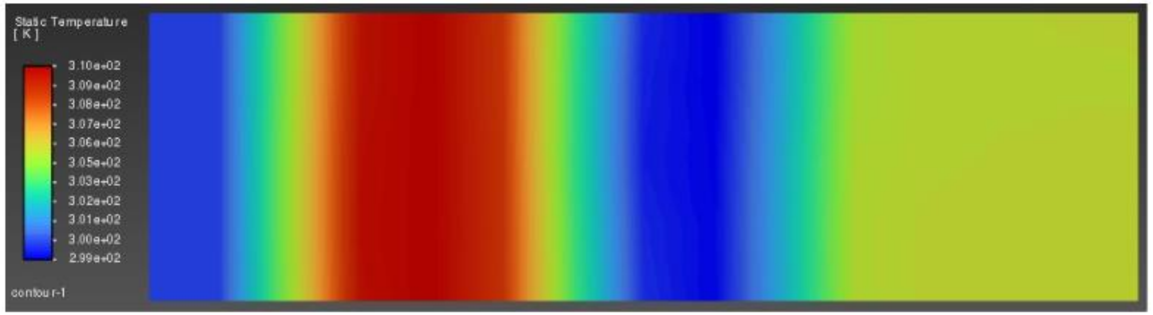
Ungrouped Macro Heat Exchanger Model [β]

- Constant and UDF-based aux-material-properties
  - No polynomial-based input
- Only NTU-based HX-performance-specification
  - No velocity-effectiveness curve
- Supported constraints:
  - Fixed inlet aux-temperature
  - Fixed heat rejection rate
- No HX-specific post-processing

Test-setup:

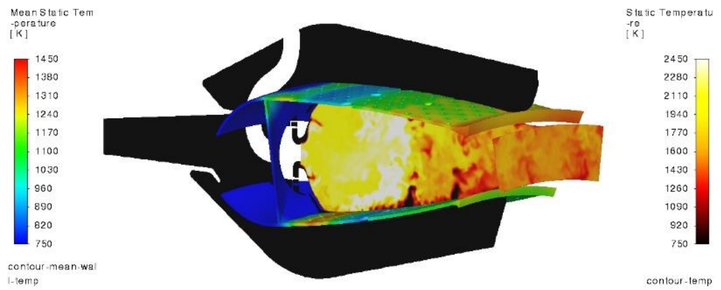
- 2-, 4- and 1-pass configs
- Aux fluid: different inlet directions
- HX-1: Fixed aux inlet temperature
- HX-2: Fixed heat rejection rate
- HX-3: UDF-based aux properties





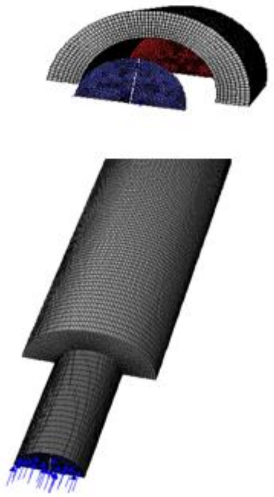
Combustion With Adjacent Solids

- Flamelet Generated Manifold (FGM) model and premixed combustion compatible with CHT:
  - Temperature-dependent properties of solid
  - Solves total enthalpy equation in solids
  - Solid time-stepping is supported



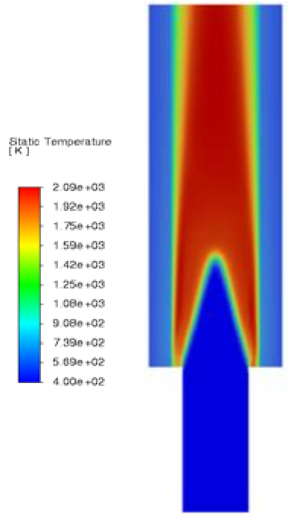
FGM with CHT

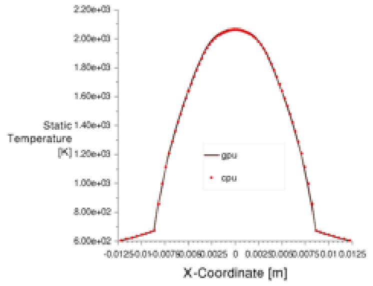
- **Premixed Combustion + CHT**
- Polynomial Cp and Piecewise linear thermal conductivity for Solids



Geometry

Mid plane temperature contour





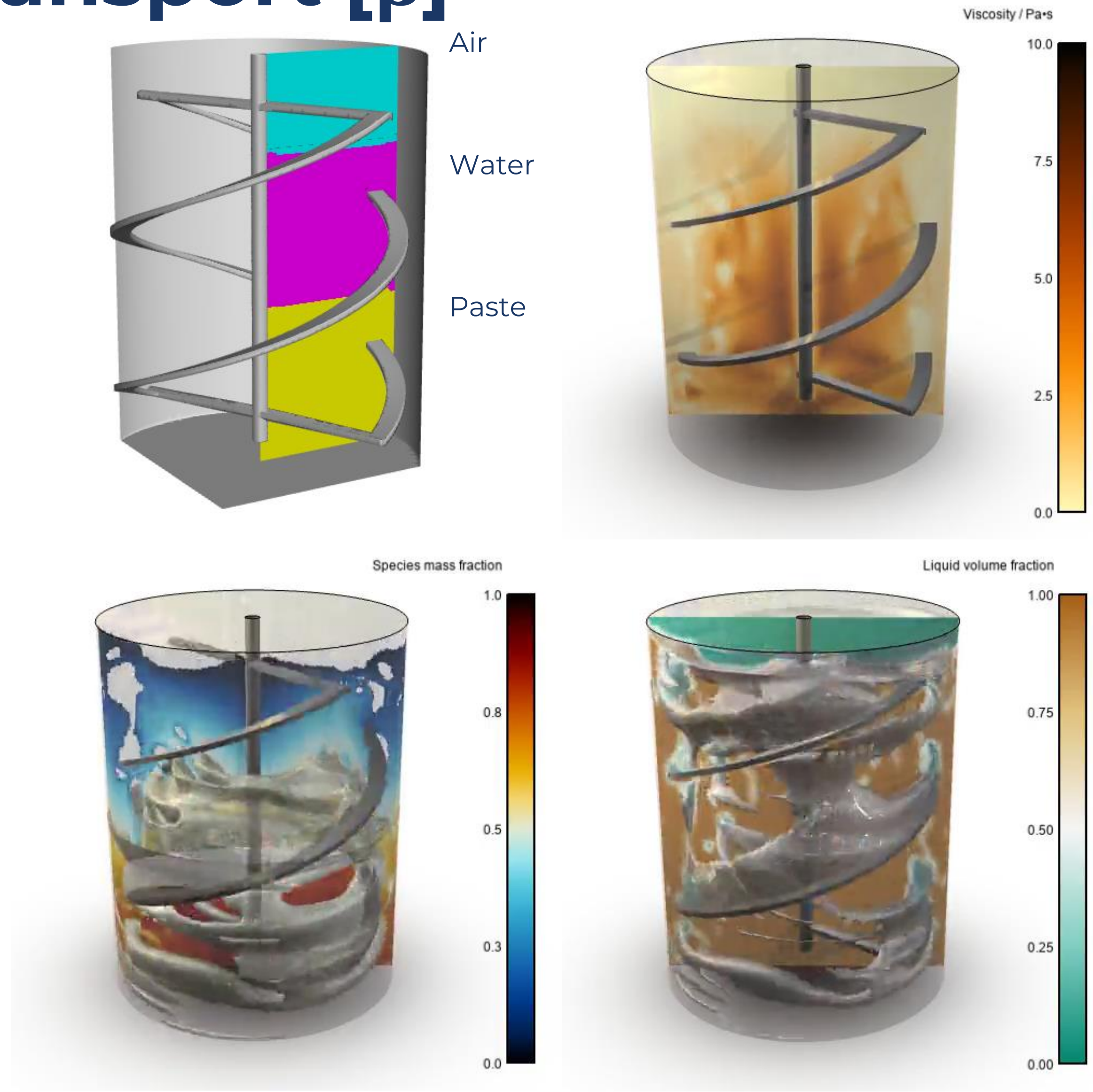
Outlet Temperature radial profile

Premix Combustion with CHT

VOF and Species Transport [β]

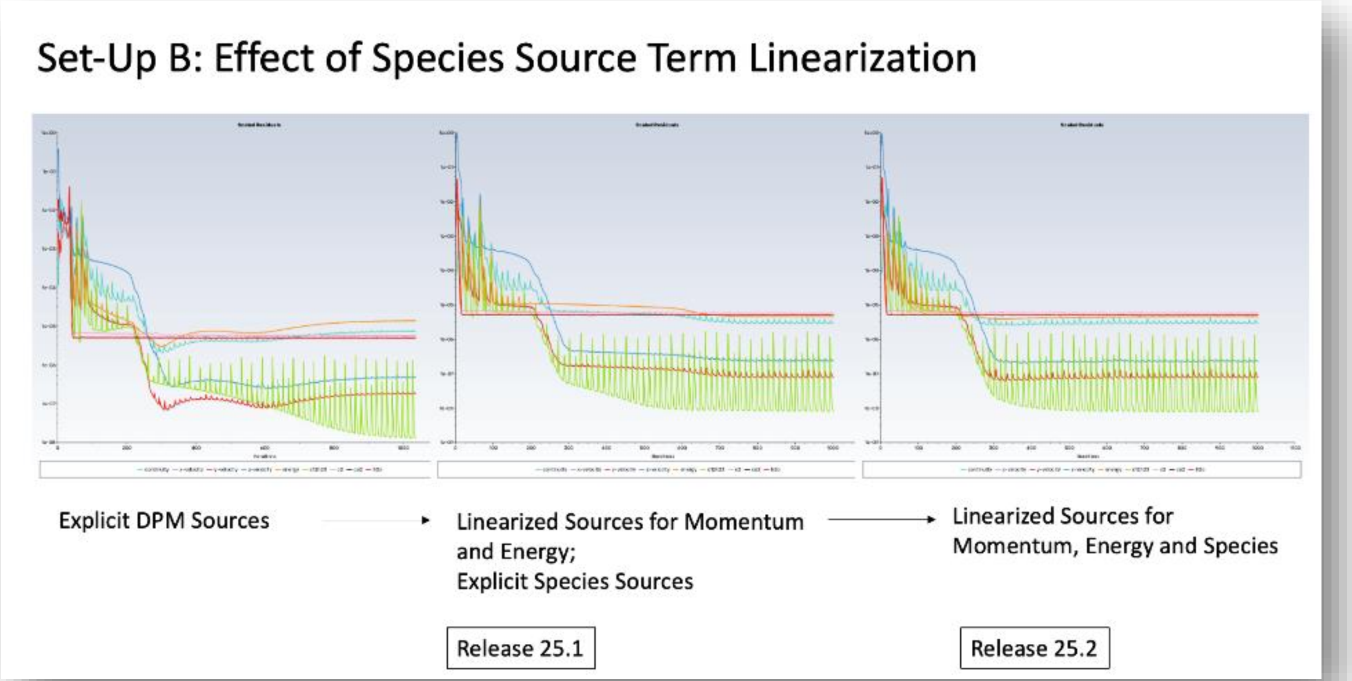
- Compatible with:
  - Steady and transient numerics
  - MRF
  - Sliding Mesh
  - Variable density framework*
  - Multiple mixtures*
  - Non-Newtonian flow*
  - Compressible liquid*

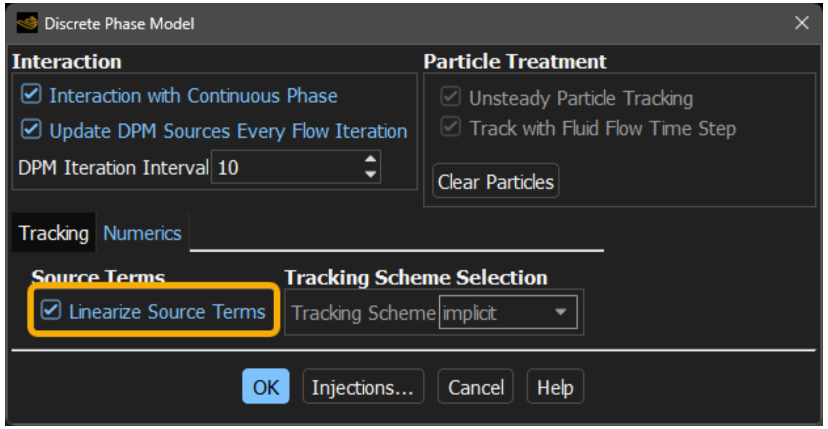
*New features



DPM Species Source Term Linearization [β]

- Activated via general DPM source term linearization flag





Target applications:
- Gas turbine combustion

DPM Properties in Python-Based UDF [β]

- DPM material properties supported by UDF:
  - `density`
  - `specific-heat`
  - `surface-tension`
  - `viscosity`
- Available particle variables:
  - Particle temperature: `tp_t`
  - Particle mass: `tp_mass`
- Hooked by material name

```
ref_T = UDF.rpgetvar("'reference-temperature")
```

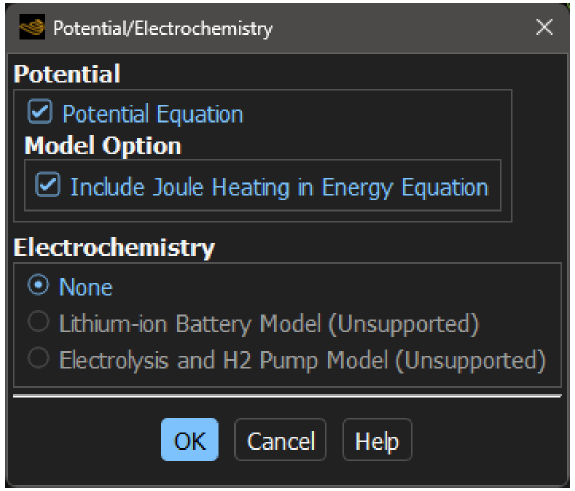
Access to RP variables must be outside of UDF function

```
def udf_particle_surface_tension(tp_t, tp_mass):
    particle_temperature = tp_t
    particle_mass = tp_mass
    surface_tension_default = 0.08
    surface_tension_min = 0.0003
    surface_tension = 0.003
    if particle_temperature < ref_T:
        surface_tension = surface_tension_default
    elif particle_temperature <= 650.0:
        surface_tension = 1.3e-01 - 3.5e-04 * particle_temperature …
    else:
        surface_tension = surface_tension_min
    print(surface_tension)
    return Real(surface_tension)

UDF.add_particle_material_parameter(udf_particle_surface_tension,
                                    'surface-tension',
                                    'n-hexane-liquid')
```

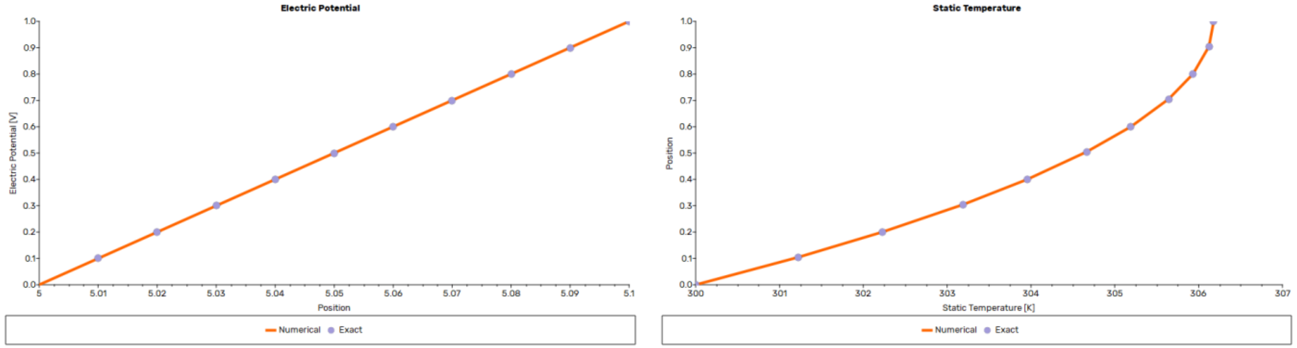

DRD

Electric Potential / Joule Heating

- Electric potential available for solid zones with GPU solver
- Supports
  - Periodic boundaries
  - Couple walls
  - Properties defined by expressions



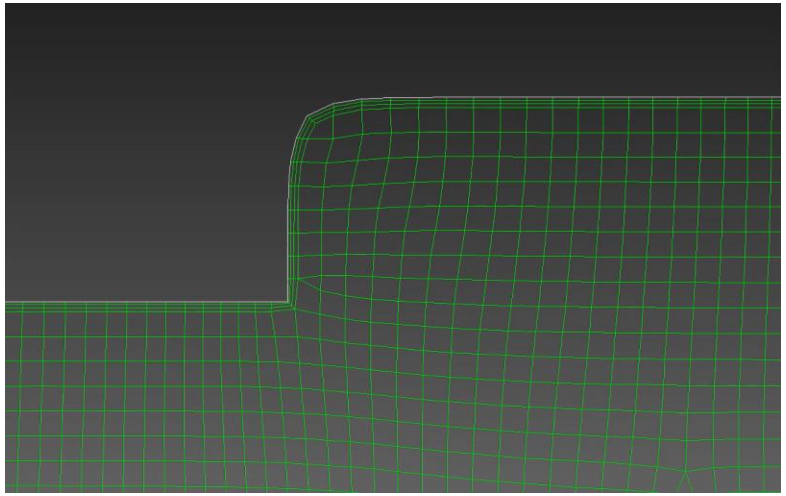
Verification example

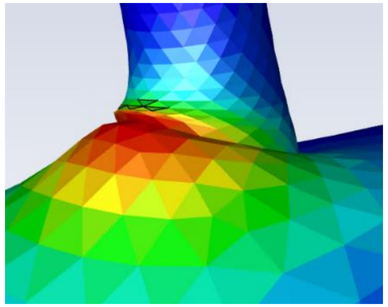


DRD

Dynamic Mesh: Ablation/Erosion Model

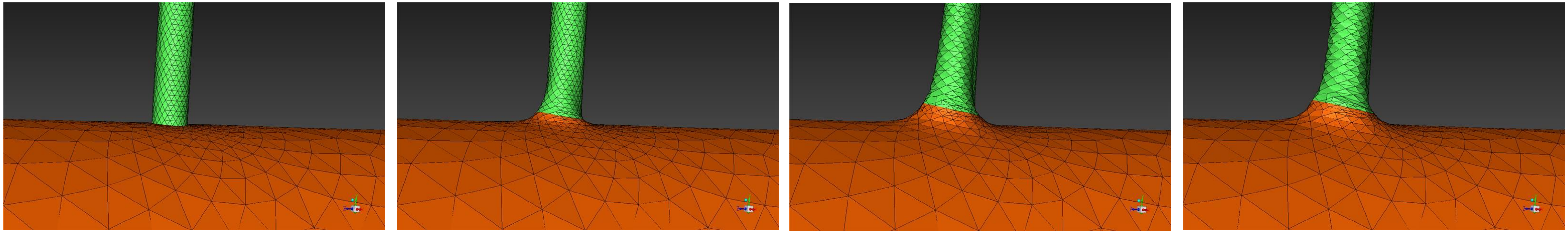
- New algorithm for node motion
  - Handling of sharp corners to reduce mesh degeneration
  - Ablation/erosion cases can run longer compared to previous versions





Old behavior

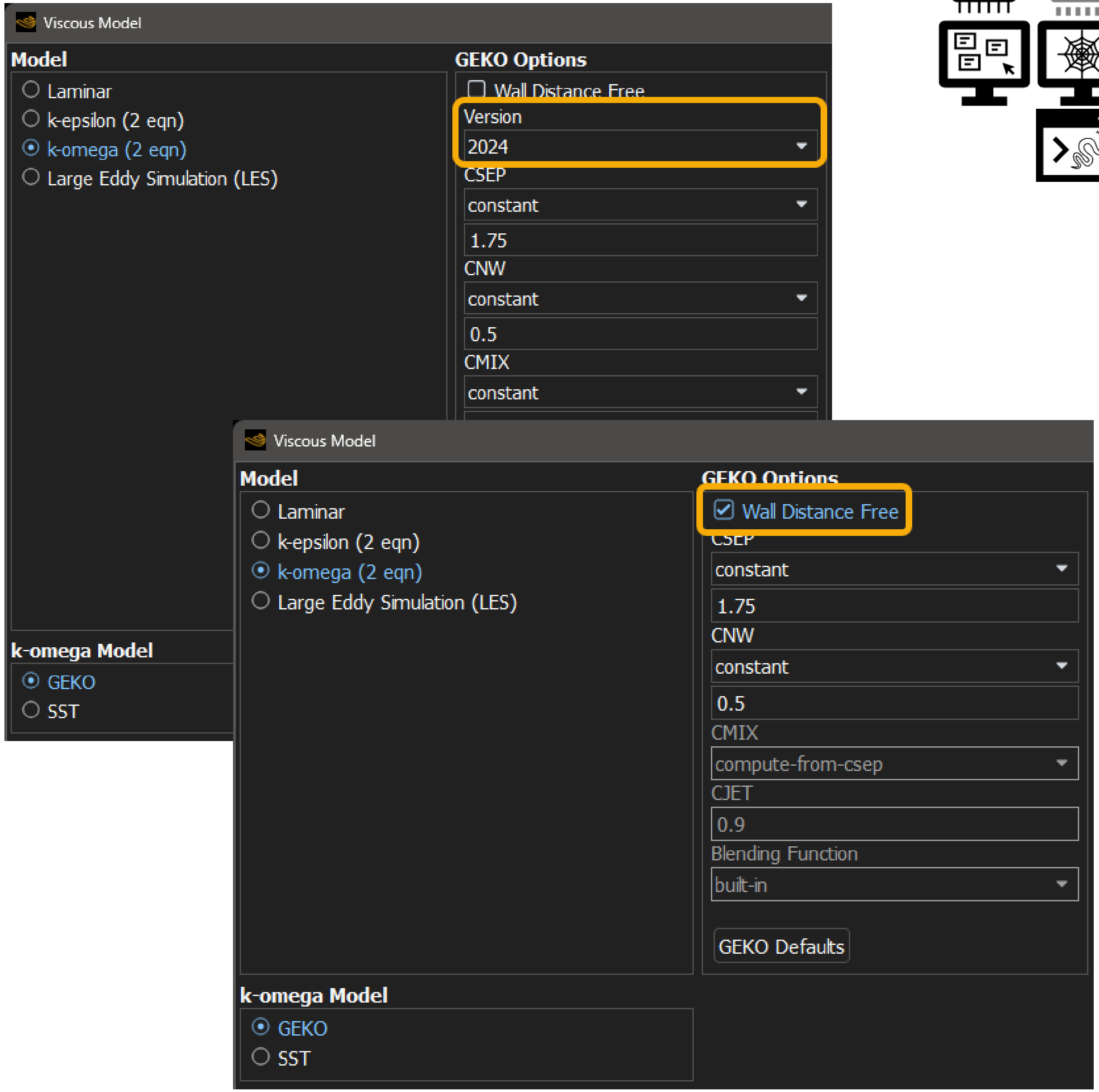
New behavior



GEKO Model

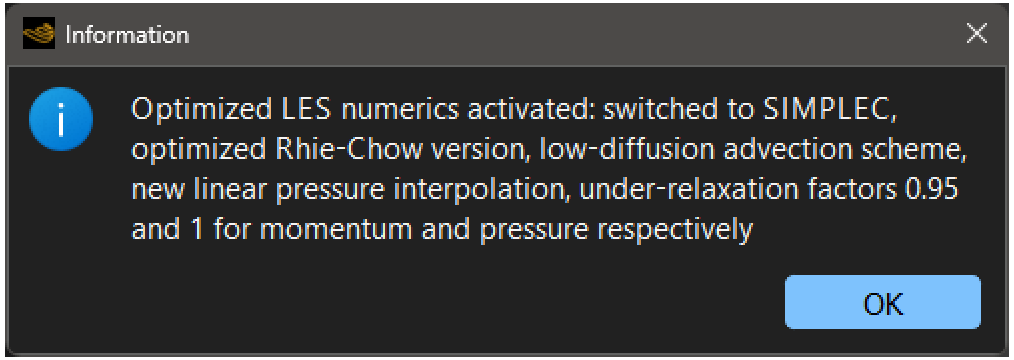
- 2024 version of the model [β]
  - C_{SEP} is also limited by the blending function
    - Avoids an effect of C_{SEP} on the outer flow
    - Simplifies tuning by avoiding interaction between coefficients
      - C_{SEP} and C_{NW} influence near-wall region
      - C_{MIX} and C_{JET} influence outer flow
  - C_{JET} no longer depends on C_{MIX}
    - Simplifies independent optimizations of coefficients

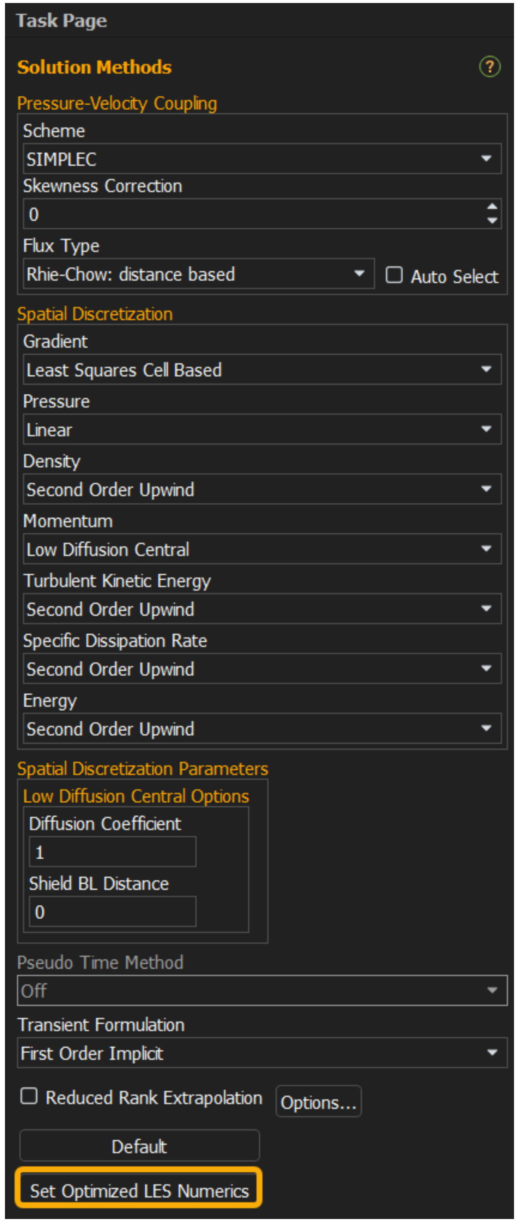
- Wall-distance-free formulation for GEKO-2024 [β] and GEKO-2015
  - Wall distance calculations can be expensive, the option modify the equations to avoid it
    - Also disables all other models that depend on a known wall distance
  - C_{MIX} and C_{JET} are deactivated
  - C_{SEP} and C_{NW} act globally, not just in the near-wall region



Large Eddy Simulations

- Adjust settings for LES calculations with Optimized LES Numerics button
  - Simplifies using recommended solver settings
  - Also available with Web UI and PyFluent



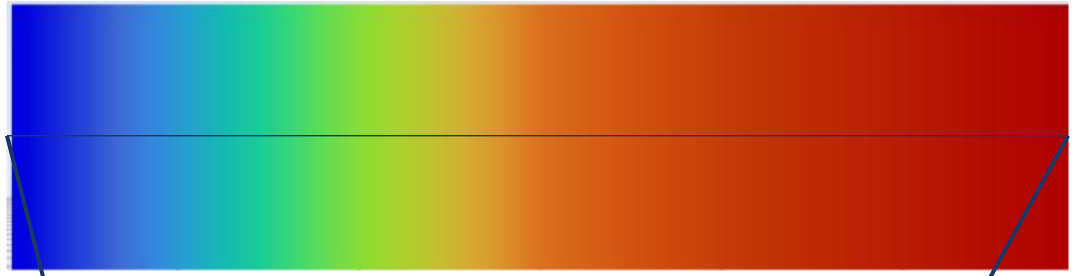


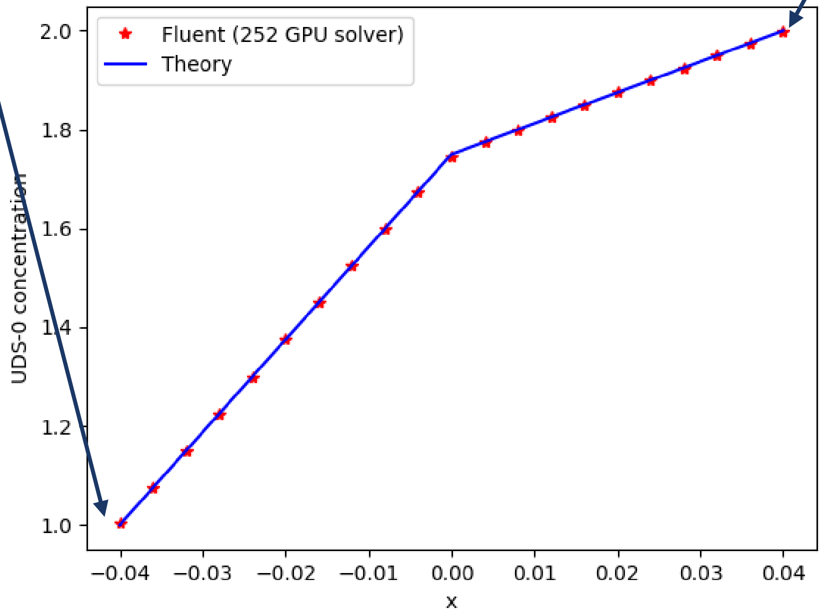
Python-Based UDF [β] – User-Defined Scalars

	Diffusivity	Boundary conditions	Source terms
Constant	2025 R1	2025 R1	2025 R1
Polynomials	2025 R2	2025 R2	2025 R1
Profiles	n/a	2025 R1	2025 R1
Expressions	2025 R2	2025 R2	2025 R1
Python UDF	2025 R2	n/a	n/a

- Diffusivity example:
  - The analytical solution is a piecewise-linear function formed by joining the following three (x, phi) pairs

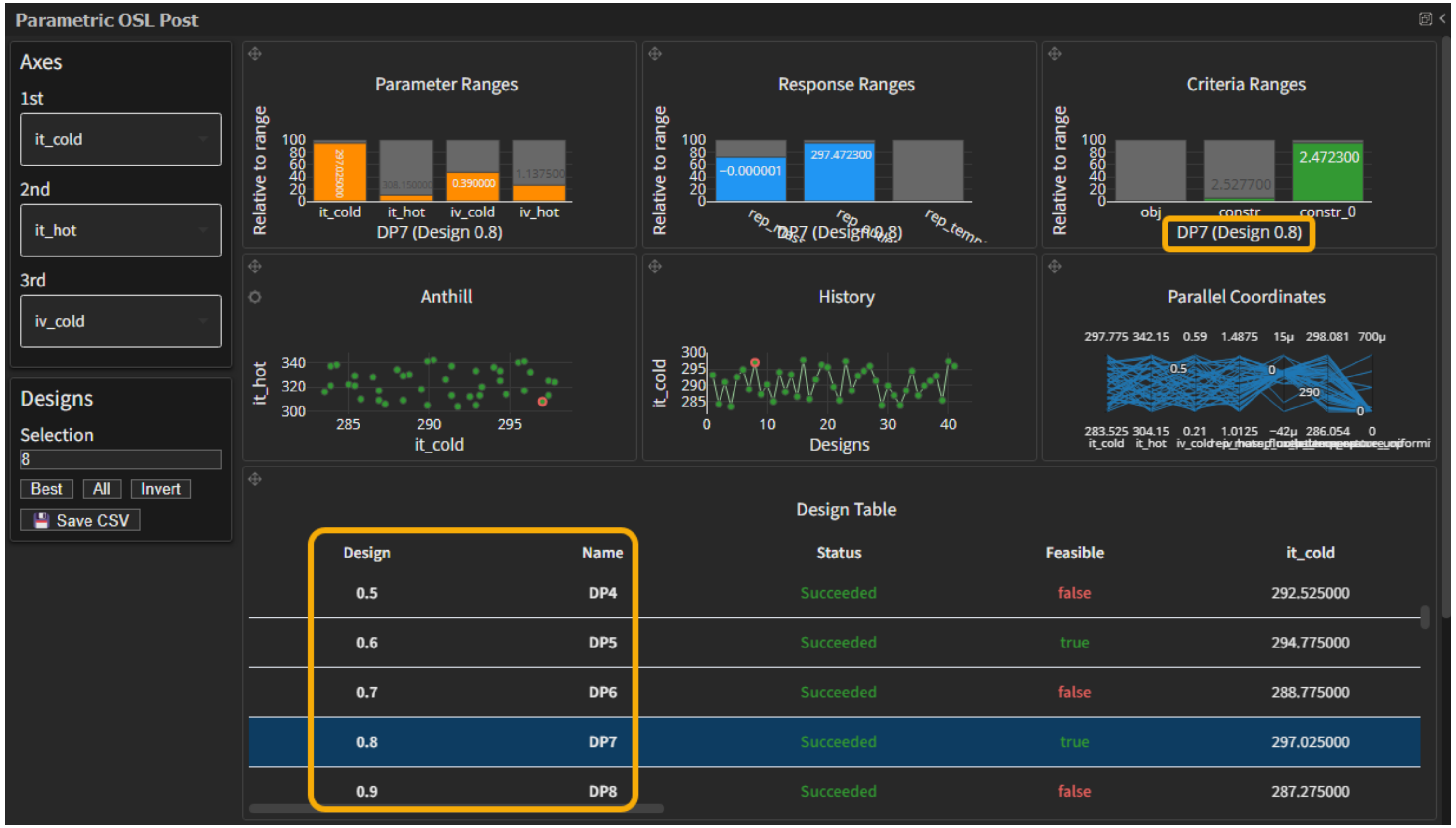
```
def udf_uds_diffusivity(c_centroid):
    x,y,z = c_centroid
    if x > 0.0:
        mag = 0.003
    else:
        mag = 0.001
    return mag

UDF.add_uds_diffusivity(udf_uds_diffusivity, 0,
UDF.all_cell_zones())
```





optiSLang Postprocessing in Fluent

- Embedded oSL post processing of Parametric Studies in Fluent
  - Parameter plot
  - Anthill plot
  - History
  - Parallel coordinate plot
- Mismatch between DP numbering of Fluent and oSL
  - DP name (Fluent) and Design number (oSL) shown where possible

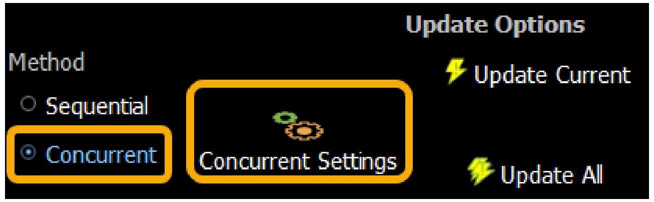


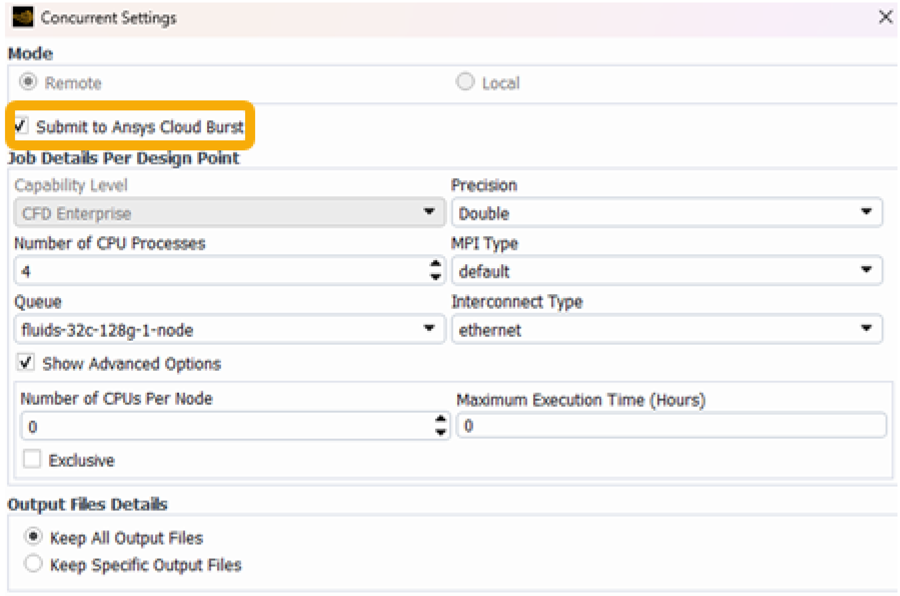
oSL number Fluent name

Parametric Study with Burst to Cloud [β]

- Only available with active beta features and logged into Ansys Cloud Burst
- Parametric ribbon: Update Options > Concurrent > Concurrent Settings
- Input files are uploaded only once
- Download behaves like for on-premise submission

- Limitations:
  - No support for HPC Parametric licensing
    - Every design point is one job and uses credits accordingly





Hybrid Precision Solver [β]

- The solver spends most of its time in AMG (algebraic multigrid solver) cycles
  - The AMG process does not benefit a lot by double precision
  - Approach: Keep the outer calculation at double precision and the inner at single precision
- Observed behavior across many cases:

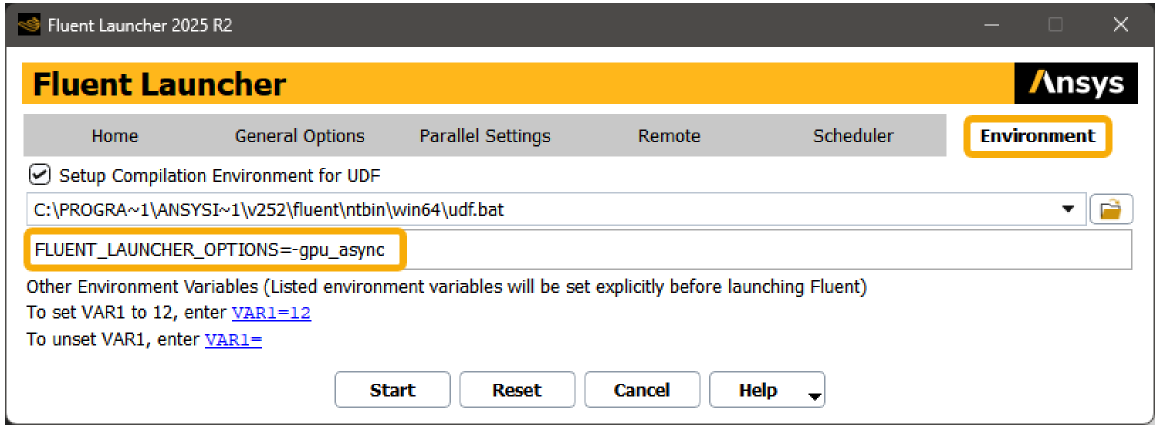
Coupled solver:
- Performance close to single precision
- Memory requirements in-between single and double precision

Segregated solver:
- Performance in-between single and double precision
- Memory requirements reduced by ~10% compared to double precision

- Supported equations: pressure, momentum, energy, k, epsilon, omega, radiation
- Must be started with command line argument in 2025 R2
  - `fluent 3ddp -gpu_hybrid_precision`

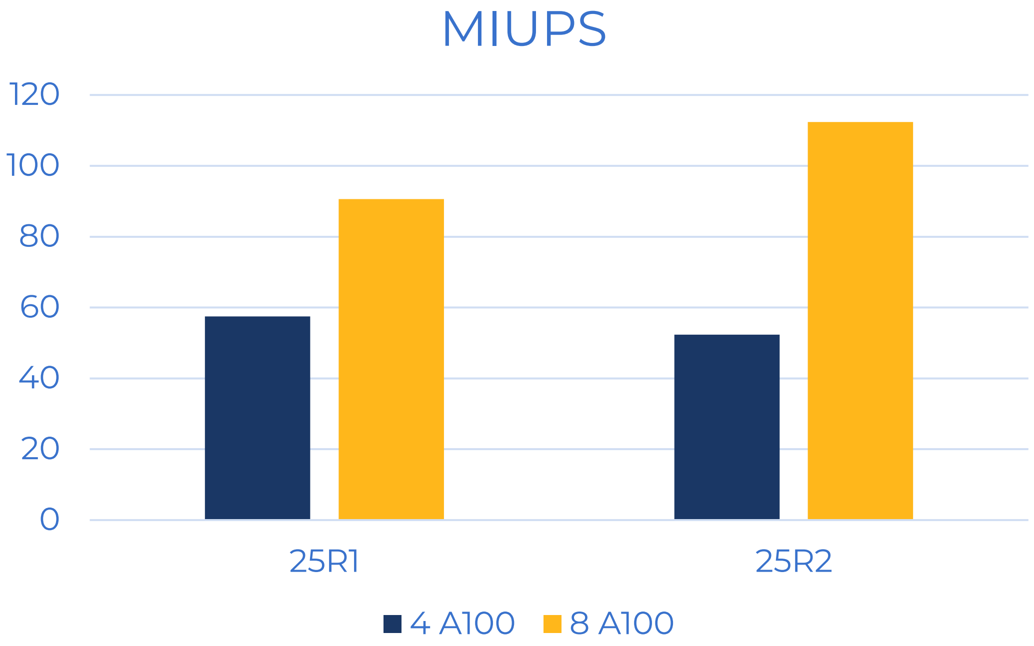
Asynchronous GPU Solver Mode

- Goal: overlap GPU and CPU operations to minimize waiting time
  - Maximum theoretical speed-up: 2X
- Start Fluent with the command line options `-gpu -gpu_async`
  - Can be combined with `-lite`
  - When using the Fluent Launcher, set the unsupported environment variable `FLUENT_LAUNCHER_OPTIONS=-gpu_async`



Sliding Mesh Improvements

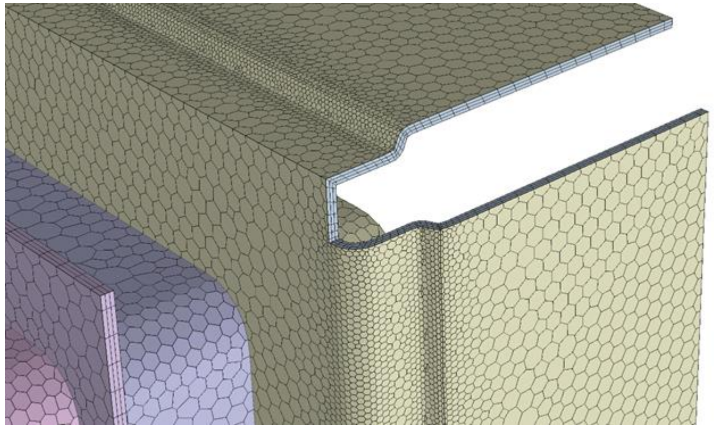
- Single precision solver is more robust
- Scalability on multiple GPUs is improved
  - Observed speed-up between 40 and 100%
    - Depends on the size of the sliding mesh zone
  - Example case:
    - Helicopter with 70M cells
    - 69% of cells in rotating zone
    - Nvidia A100 GPUs

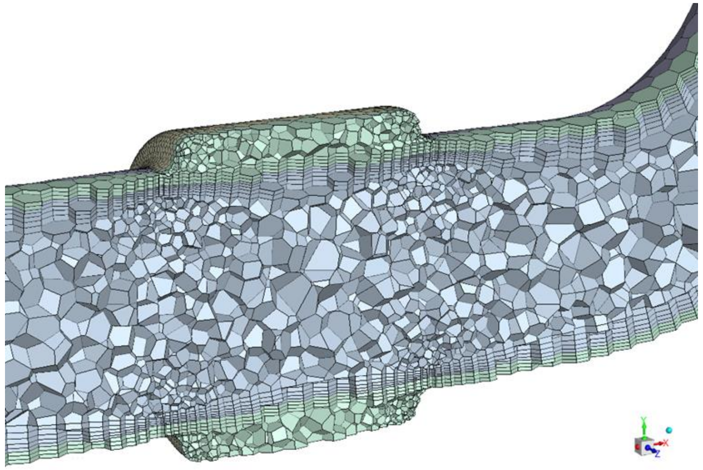


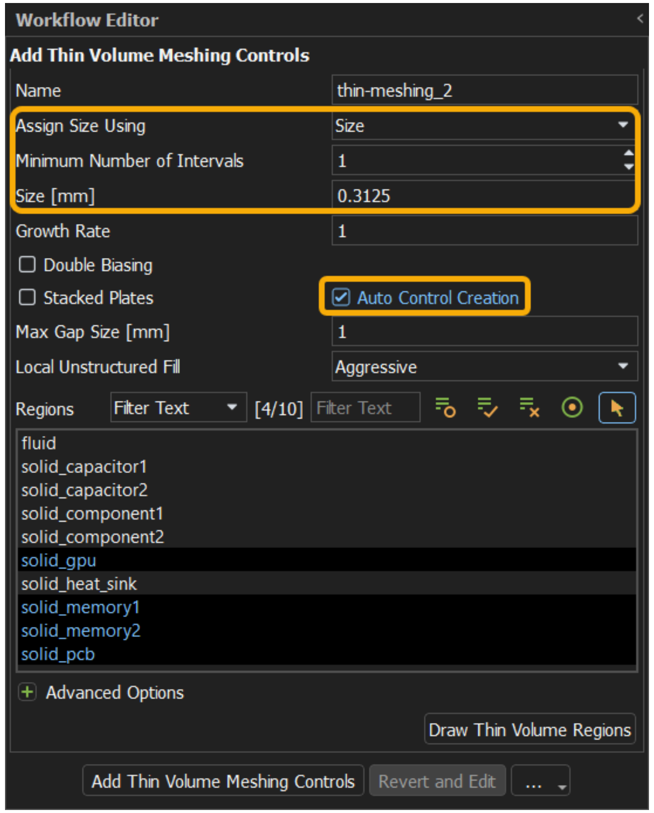
Fluent Meshing

WTM - Automatic Thin Volume Meshing

- Flexible layered thin meshing in Watertight Meshing Workflow
  - Automatic detection and ordering of suitable local regions and source/targets
  - Automatic transition to unstructured in non-thin areas
  - Note that with mesh import, zones need to be separated in the Surface mesh task or in Manage Zones task



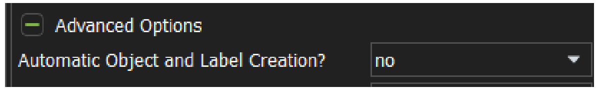


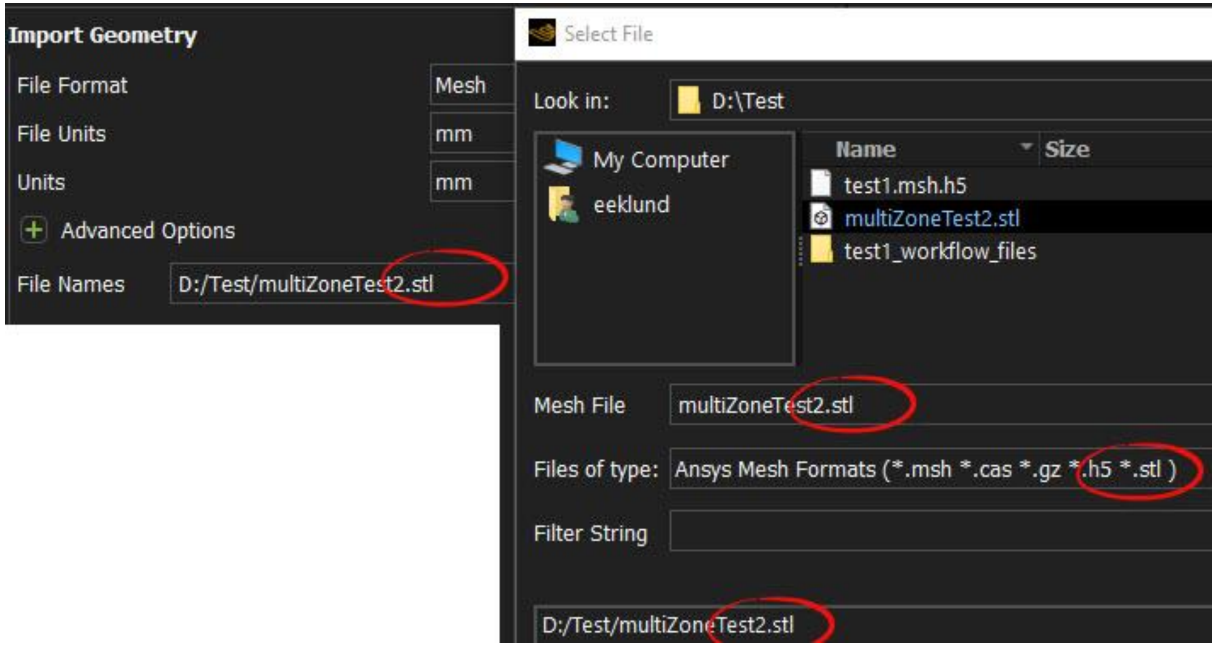


- Improvements:
  - Add Max Number of Intervals when specifying intervals by Size
  - Parallel volume meshing for non-conformal regions without any thin controls
  - Allow prism layer settings for regions which Auto Control Creation could not identify as thin

DRD

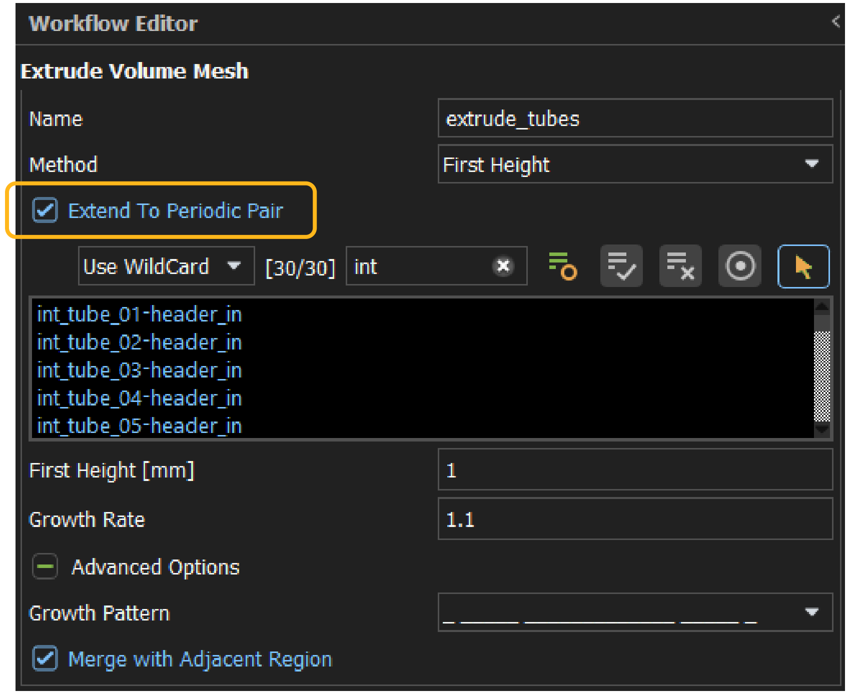
WTM – STL Import

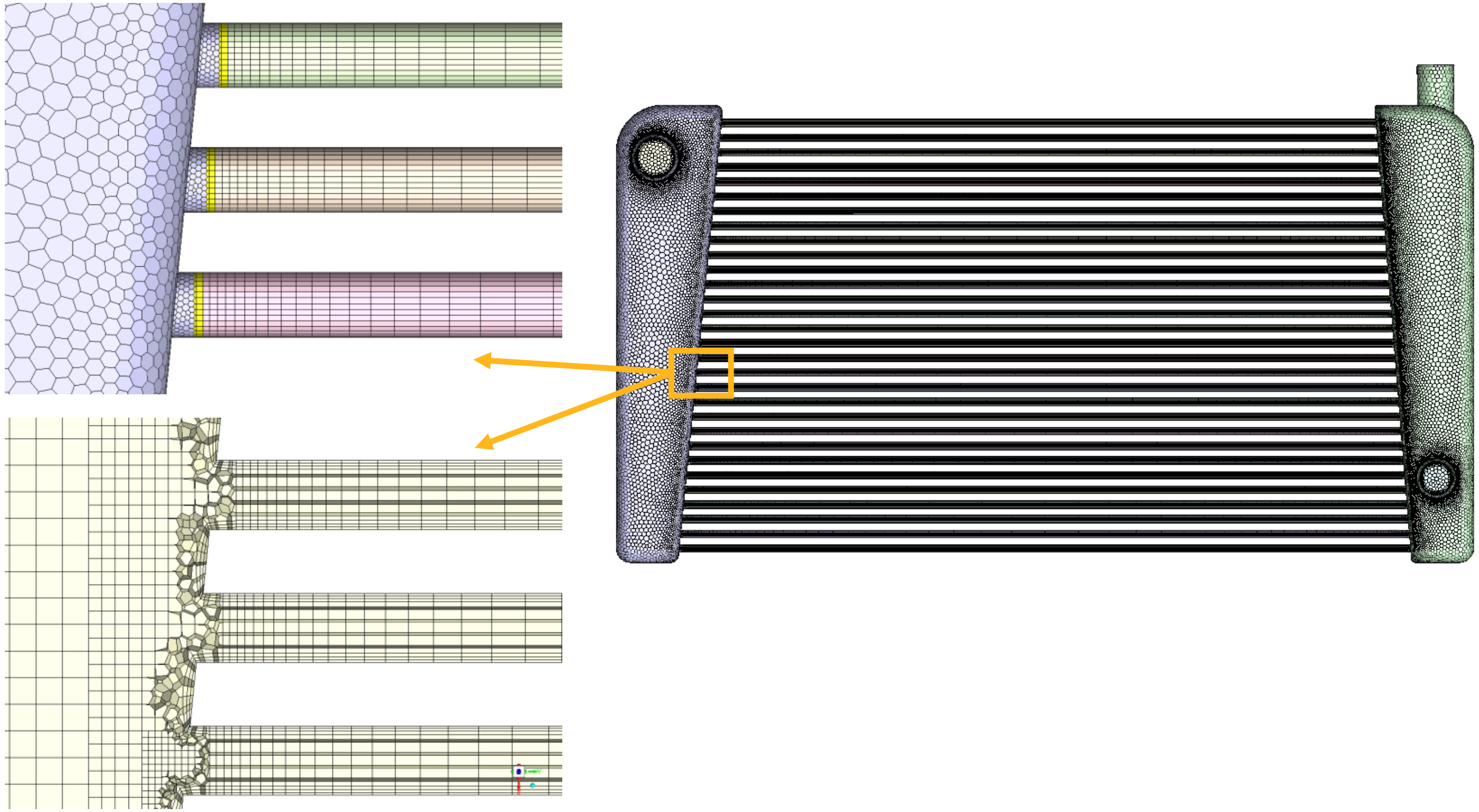
- Select *.stl file(s) in WTM Import Geometry task.
- Supports workflows where a high-quality initial surface mesh in *.stl format is available from a 3rd party tool (e.g. ANSA)
- Notes:
    - The *.stl file should be of "mesh" quality
        - i.e. connected and with reasonable quality
    - For a single part
        - Set automatic object creation to no
    - For multiple parts
        - Use one *.stl file per part
        - Use multiple file support to import all at once
    - 



WTM – Extrude with Extend to Periodic Pair

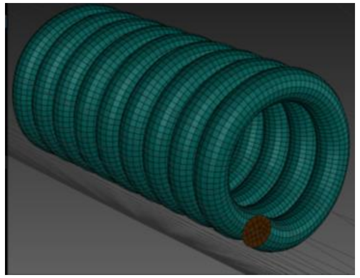
- Extrude multiples tubes of different dimension in one operation / task
- Fully conformal mesh



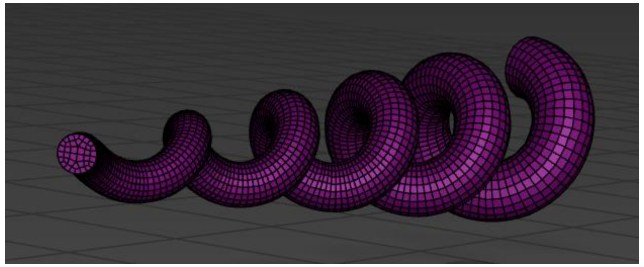


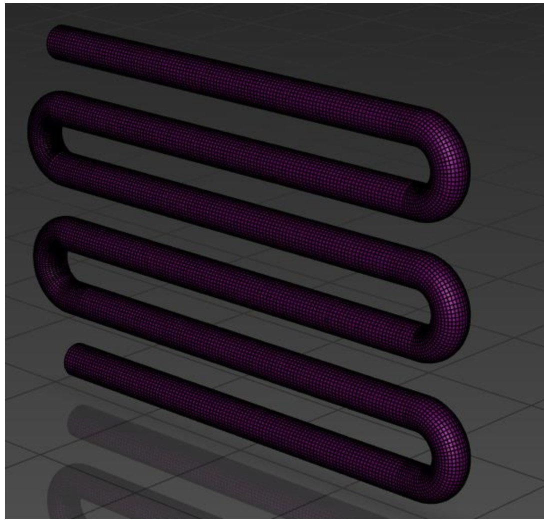
WTM – Medial Axis Multizone Meshing [β]

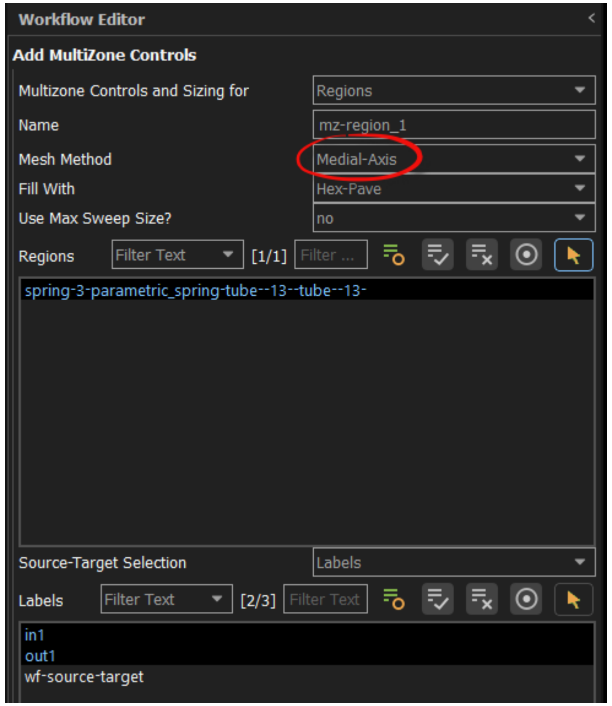
- New meshing method particularly suitable for
  - Long curved pipes and channels and 360° axisymmetric models
- Avoids skewed cells by better-aligning blocking



- Limitation:
  - Some complex arrangements may fail to mesh

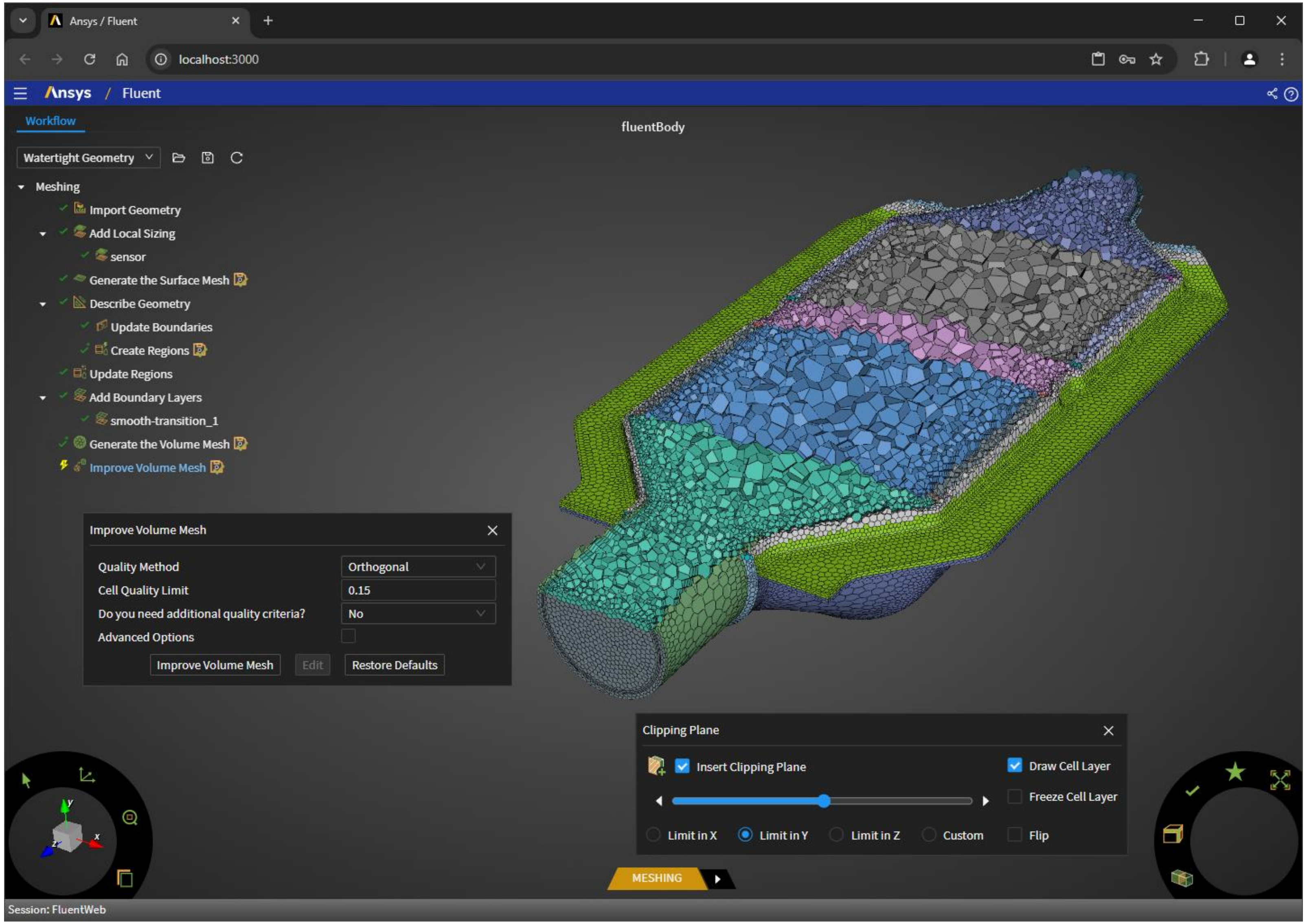






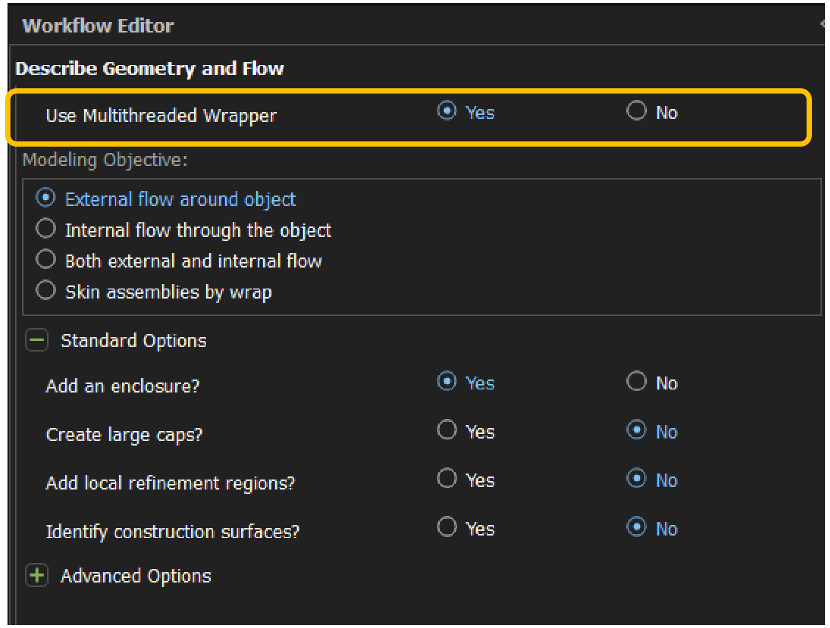
WTM – Web User Interface

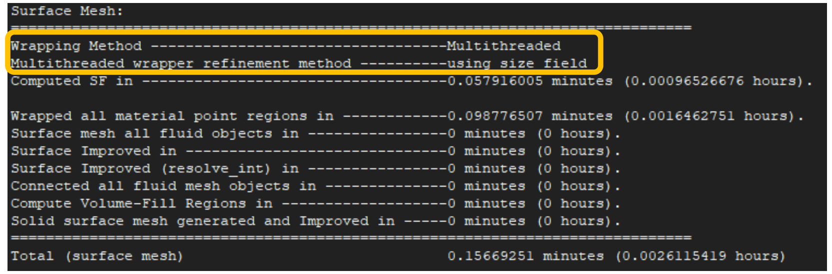
- WTM available with Fluent Web User Interface
  - Most workflow steps are exposed in the tree
  - Diagnostics options are available from the right arc
    - Only available if Fluent is started with full graphics support
  - RMB context menu to interact with the graphics directly



FTM – Multithreaded Wrap and Size Field Computation

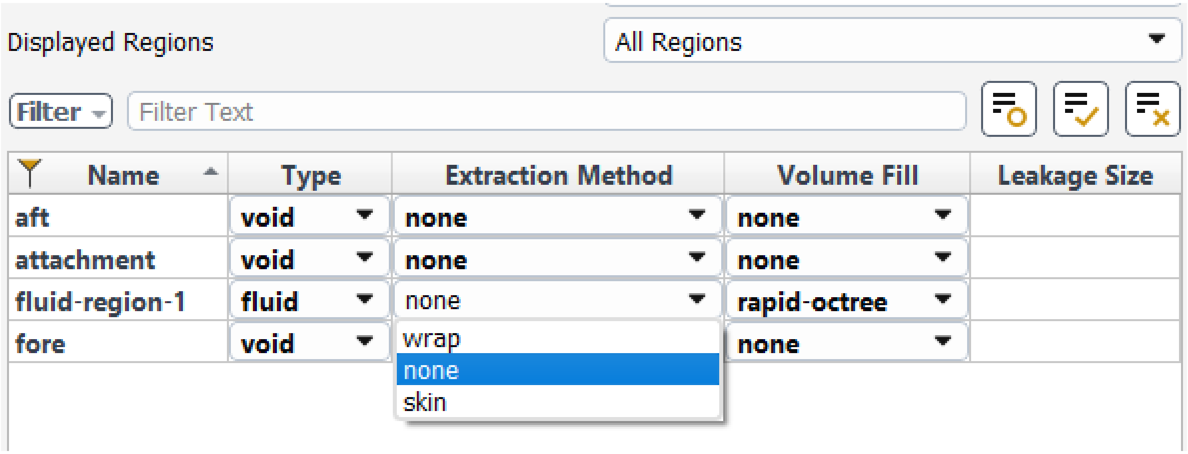
- New Prime wrapper technology available in FTM
  - Multithreaded wrapping will be applied to main fluid wrap. Other regions will use current wrapping technology
  - When enabled, size field computation is also multithreaded
- Yields significant speedup in Surface Mesh creation
  - **3x – 4x** surface mesh speedup on industrial ext. aero cases
- Unsupported options
  - Part-replacement
  - Overset mesh
- **Changes / Additions from Beta Feature**
  - Renamed in UI to "**Multithreaded Wrapper**"
  - Prime-based leakage prevention used automatically when multithreaded wrapping is used
  - Solid and Fluid objects can now use the same wrapper





Rapid Octree in FTM [β]

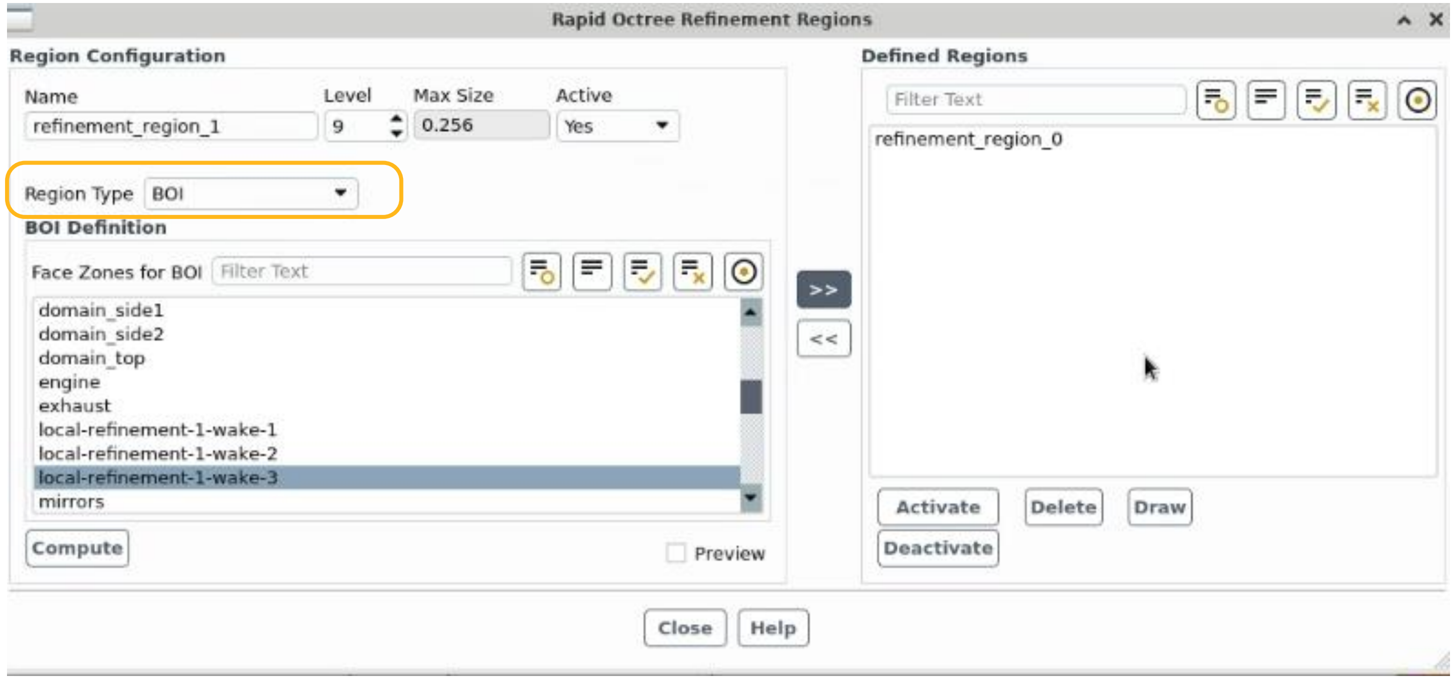
- **Rapid Octree** method can be used as a **Volume Fill** method in FTM Update Regions
  - Extremely fast volume fill for bulk regions
- Requires Extraction Method to be either:
  - None
    - Rapid-octree is applied directly on the geometry
  - Skin
    - The geometry is first skinned before applying rapid octree
    - Prime wrapper will be used for skinning if it is enabled
- Can be combined with all other volume fill types in other regions
  - Note: Rapid Octree scales much better than other fill methods, so if it is combined with other methods using very large core counts may not be beneficial.



- Notes / Limitations
  - Size controls on edges are ignored in rapid octree regions
  - Overset and Part Replacement not supported with Rapid Octree
  - Beta features must be enabled *before* reaching the **Update Regions** task

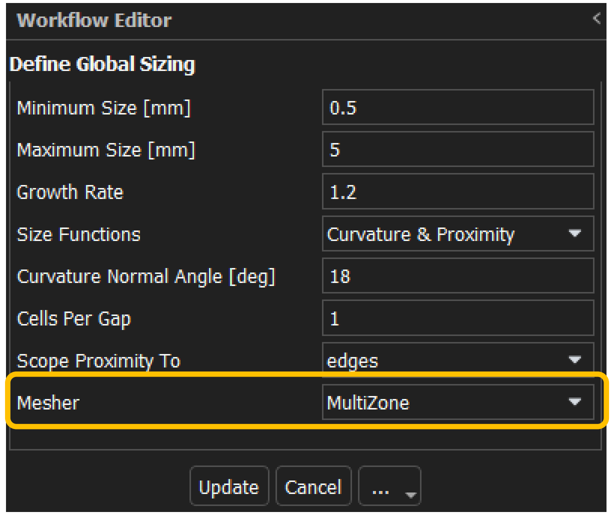
Rapid Octree – BOI support

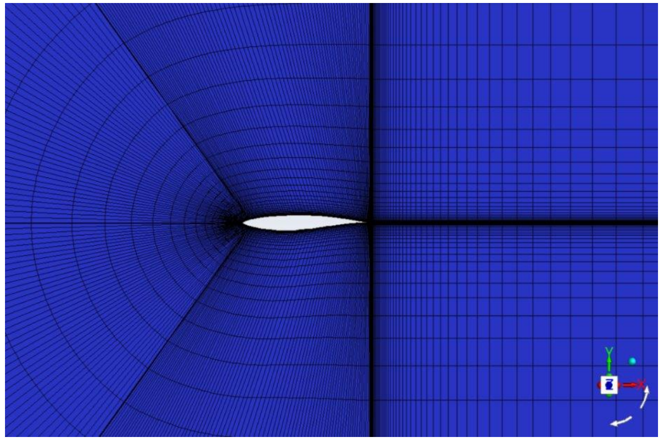
- BOI refinement region type enabled
  - Generic shapes are now supported in addition to box and cylinder/frustrum geometries
- Imported geometry should be a closed triangulated surface

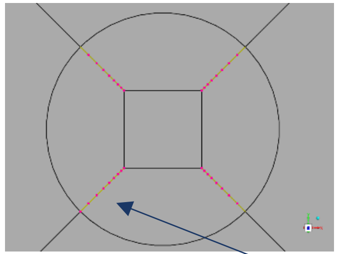


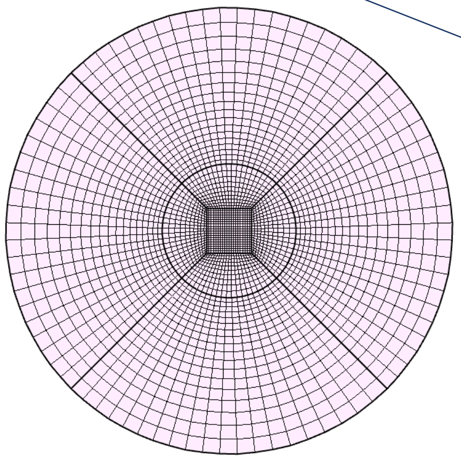
2D – Quad Mesh with 2D MultiZone Meshing

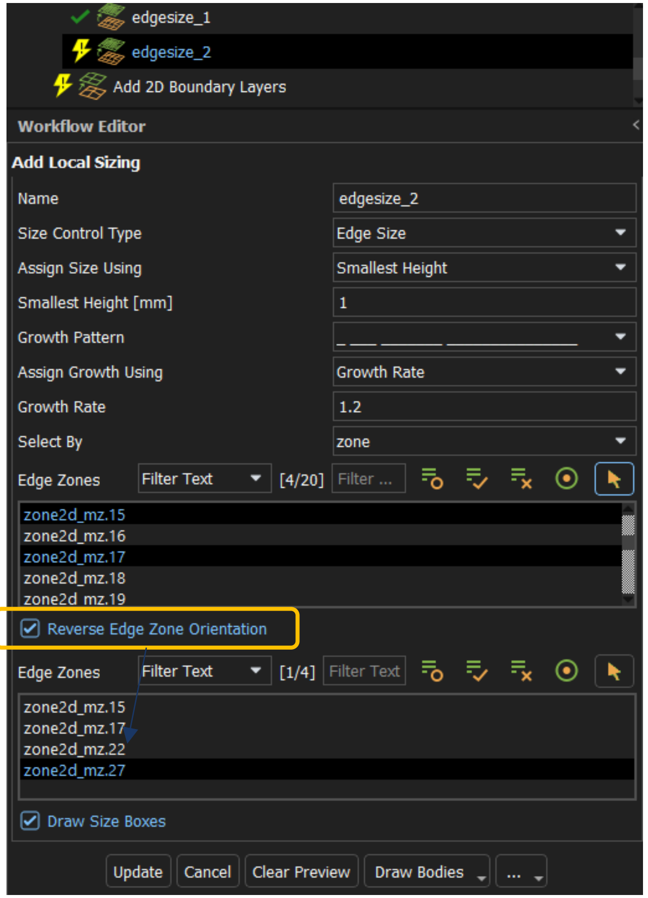
- Generation of *pure* quad mesh driven by edge sizing
  - Enable MultiZone Mesher in the Define Global Sizing task
  - Set up edge sizing for the boundaries
    - Size biasing available
    - Reverse biasing orientation option for selected boundaries
      - Assign Size using “First Height” or “Interval options”, but not “Size”







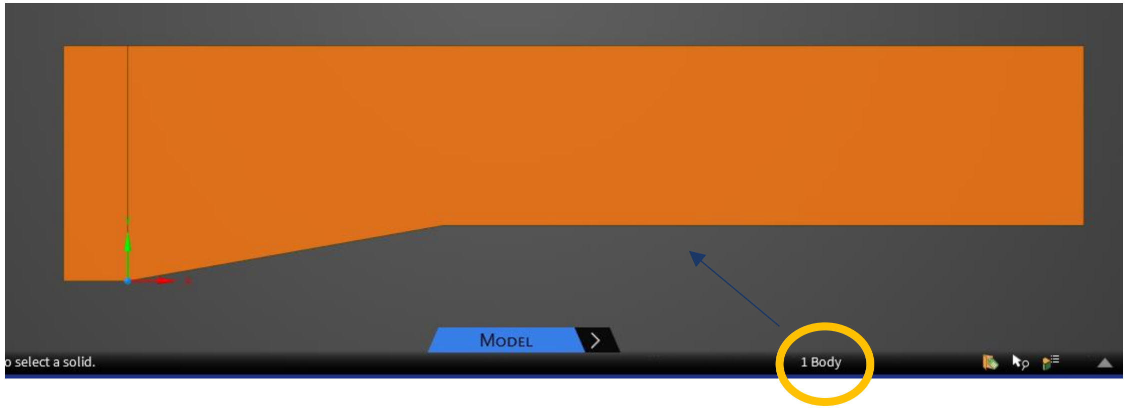


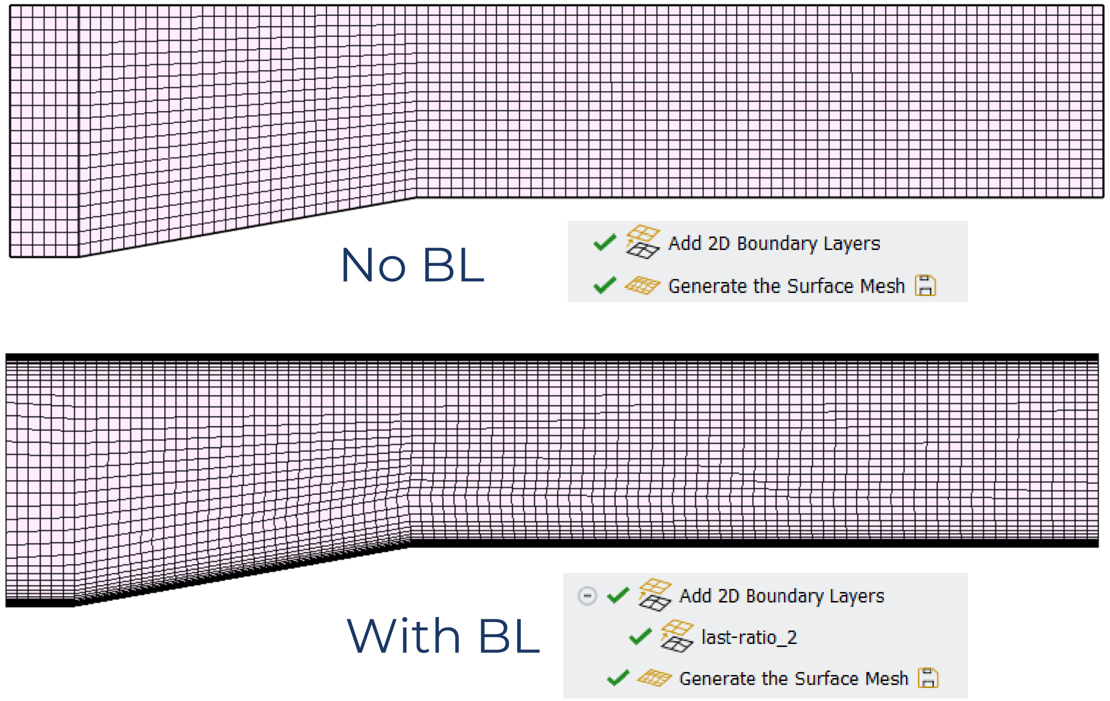


DRD

2D – Quad Mesh with 2D Multizone Meshing

- Quad mesh is compatible with boundary layers growth
- Only single body geometries supported for multi-face geometries

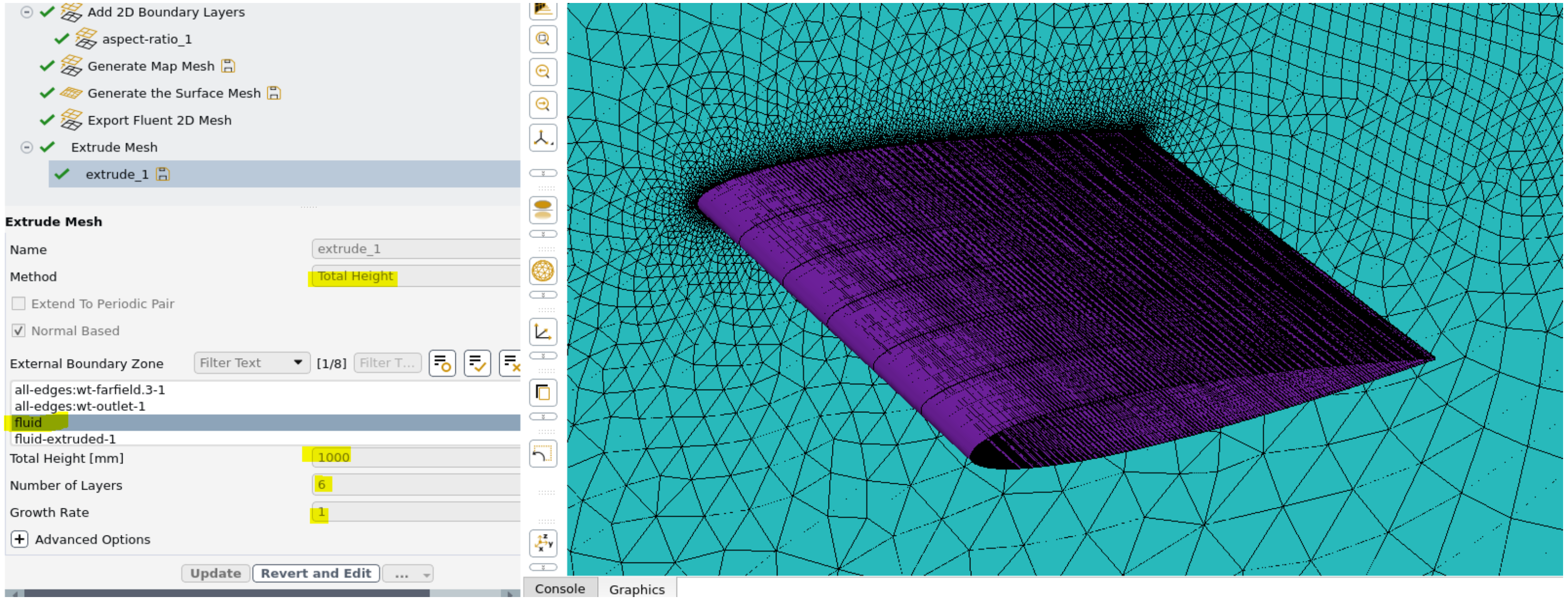




DRD

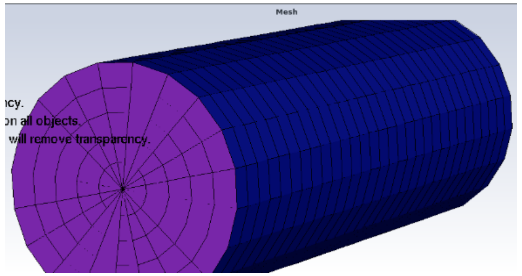
Extrude 2D to 3D

- Insert **Extrude** task(s) after 2D mesh creation
- 3D face zone types/names are taken from the 2D source edges
- Limitation
  - Baffle edges in 2D will not be preserved as internal faces in the extruded 3D geometry

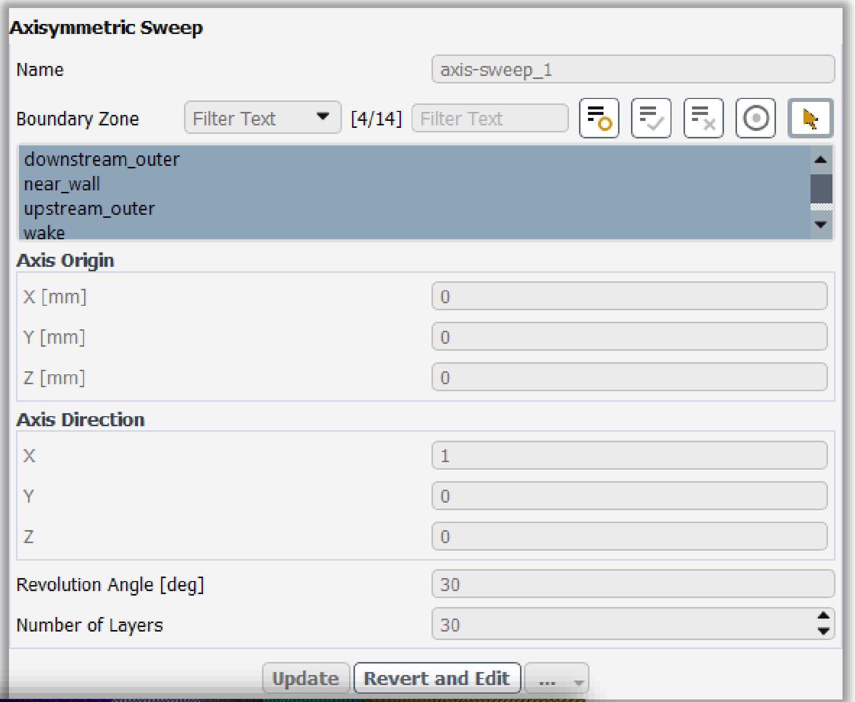


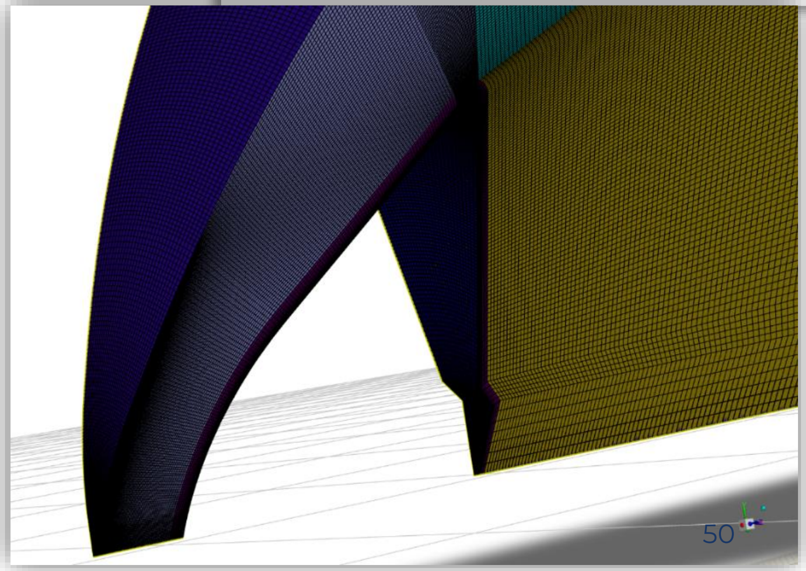
2D Axisymmetric Sweep Task [β]

- An **Axisymmetric Sweep** task can be added in the 2D Meshing workflow to revolve the 2D mesh into a 3D mesh.
- Faces adjacent to axis are revolved as wedges to avoid degenerate cells.
- User specifies:
  - Axis origin and direction
  - Revolution Angle
  - Number of Layers along the revolution (uniform only)




```
Meshing/mesh> man lis

  id                  name   type   count   tet   hex  pyramid  wedge   hex-prism
 ----  ------------------  -----  ------  ----  ----  -------  -----  ---------
 305              fluid-305  fluid   10440     0  2400        0   8040          0
```

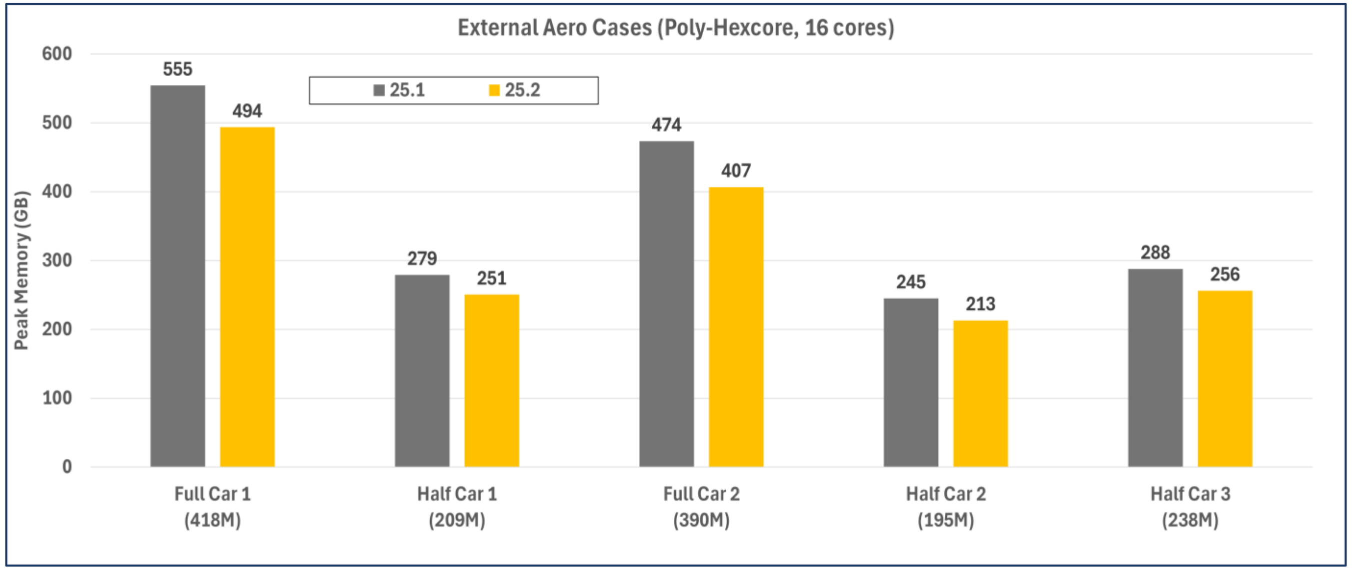


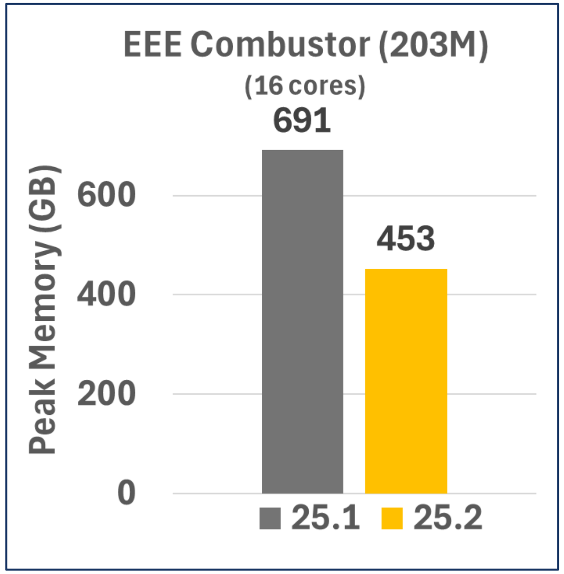


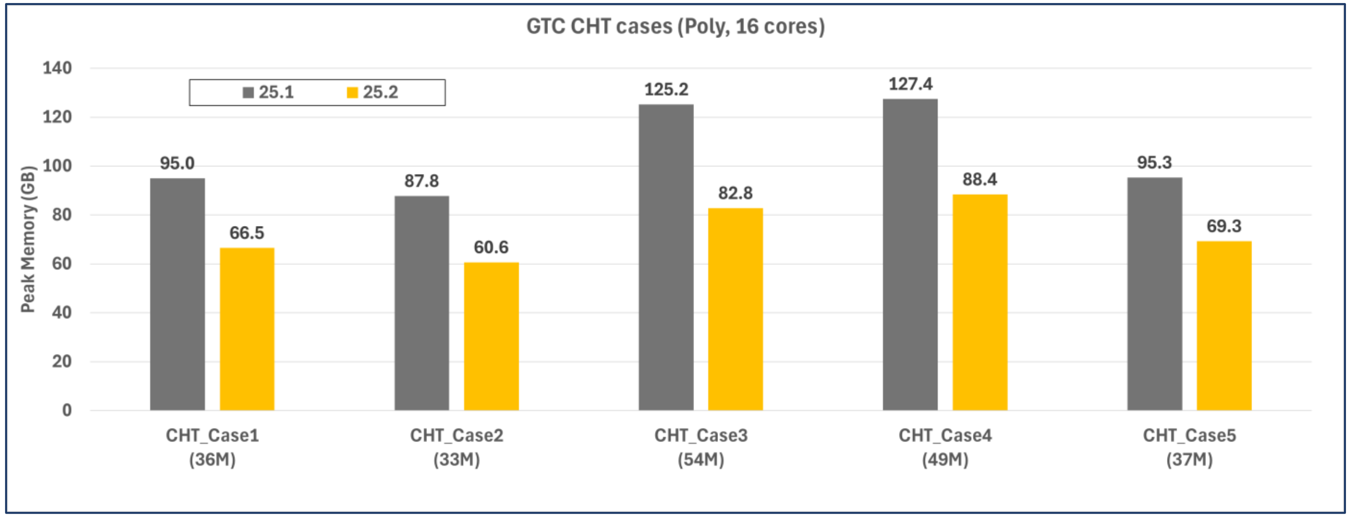
Poly/Poly-Hexcore meshing memory reduction

- 10-15 % reduction in peak memory requirement for Poly-Hexcore volume meshing

- 25-35 % reduction in peak memory requirement for Polyhedral volume meshing



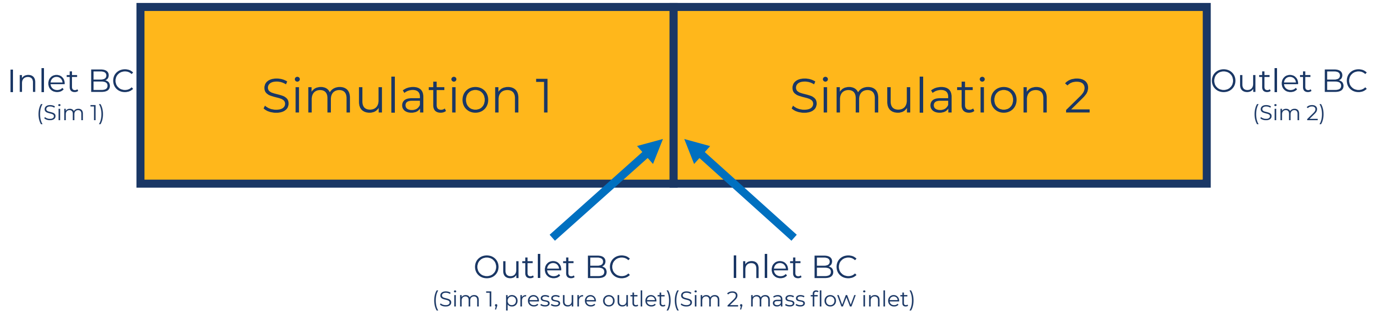




System Coupling

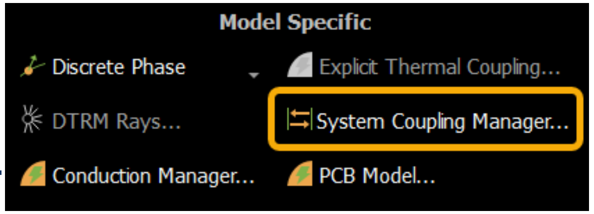
Flow Boundary Coupling

- Couple flow boundaries of Fluent and CFX [β]
- Couple flow boundaries of two Fluent sessions
  - Transfer quantities like pressure, mass flow, turbulence, and temperature at boundaries between sessions
- Especially useful for different:
  - Physics
  - Time scales

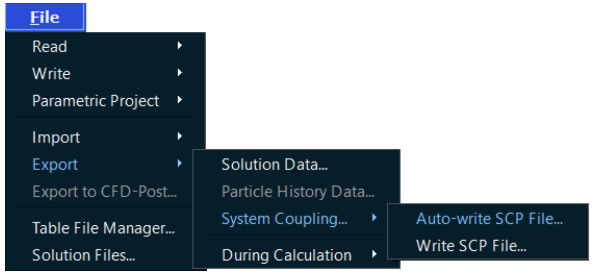


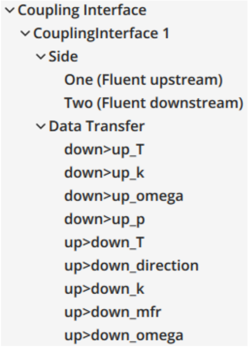
Flow Boundary Coupling: Fluent Workflow

- Activate from TUI or with PyFluent for all participants:
  - `/define/models/system-coupling-settings/flow-boundary-coupling/activate-flow-boundary-coupling-model? yes`
    - New option in Physics ribbon: Model Specific > System Coupling Manager
      - To define the zone where boundary conditions should be transferred
  - `solver.settings.setup.models.system_coupling.flow_boundary_activate_flow_boundary_coupling_model=True`
    - Define coupled zones under `solver.settings.setup.models.system_coupling.flow_boundary_coupling.specify_zones_to_activate`



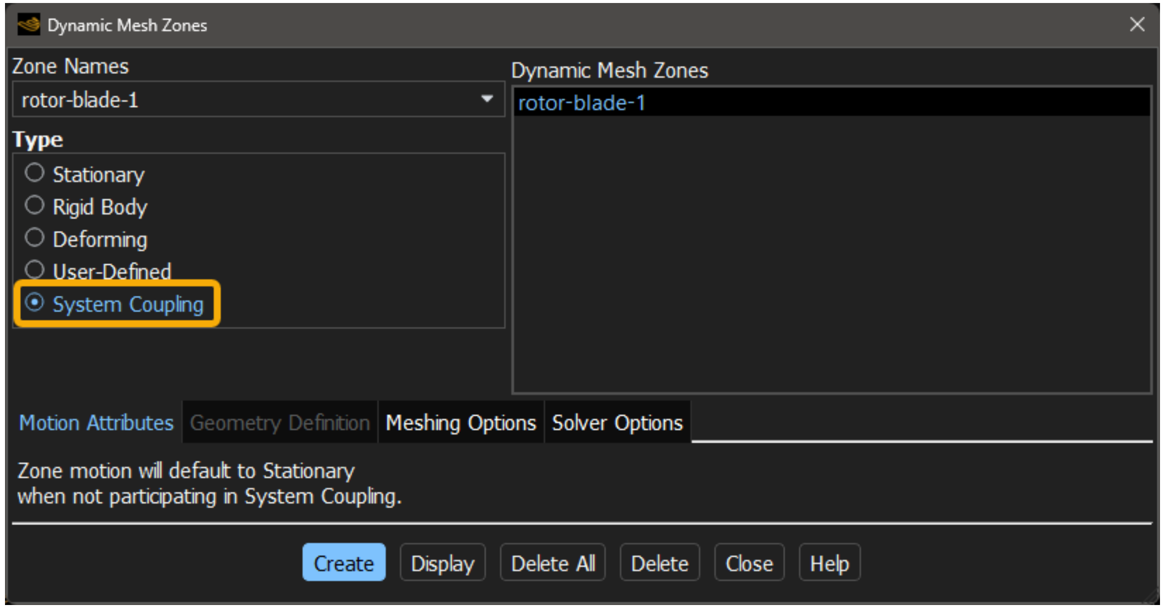
- Export scp file to use it in the standalone System Coupling application
  - Alternatively, use the Auto Write SCP File option
  - Full setup *not* available within Workbench





Rigid Body Motion FSI

- Can couple Fluent and rigid body solvers (for example, Ansys Motion FMU) to perform 2-way FSI
- Translational and rotational motion parameters supported via SyC
- Motion attributes (parameters) are transferred to Fluent via SyC

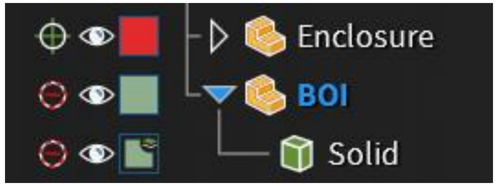


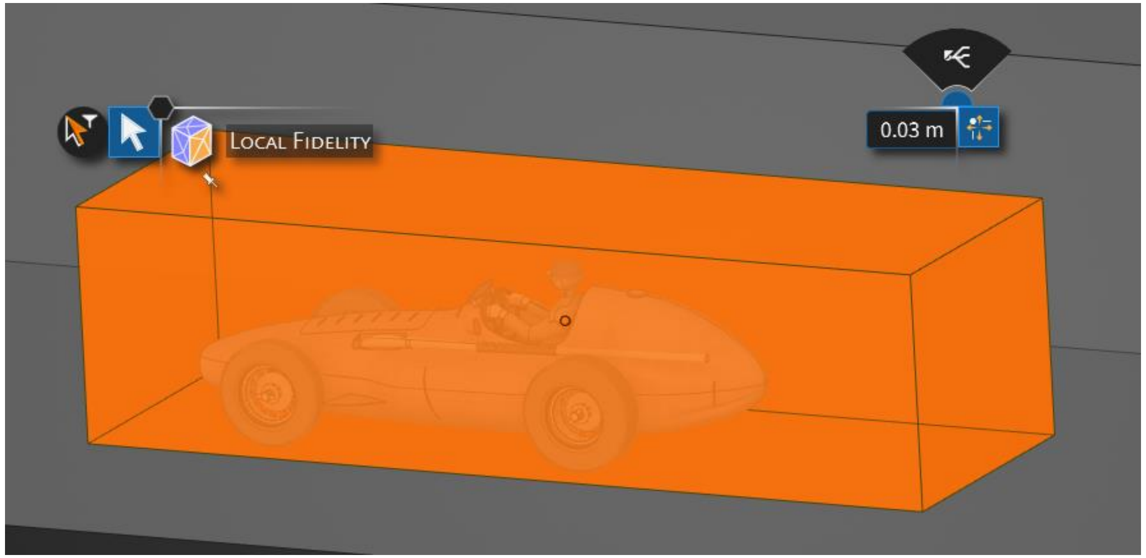
DRD

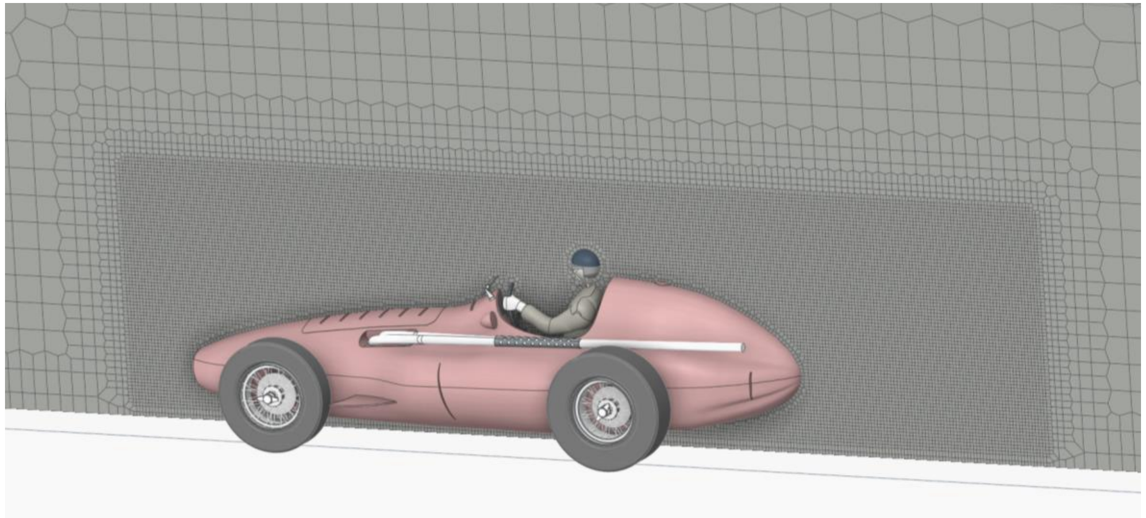
Discovery

Body of Influence Local Fidelity for Fluids (Explore)

- New option to apply local fidelity to a body suppressed for simulation
  - Body can be any shape and either fully or partially overlap fluid or solid regions
  - Can be combined with other local fidelity controls with smallest size winning
  - Provides additional flexibility for controlling mesh density and optimizing GPU memory usage

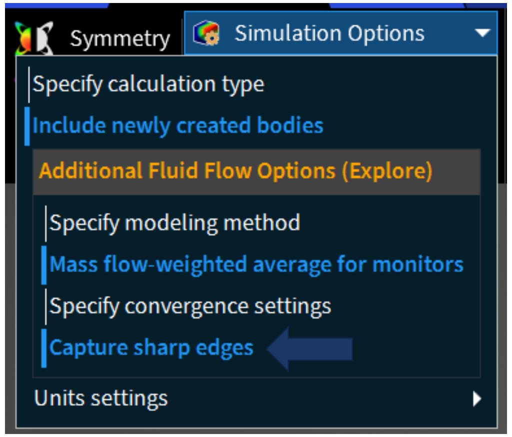


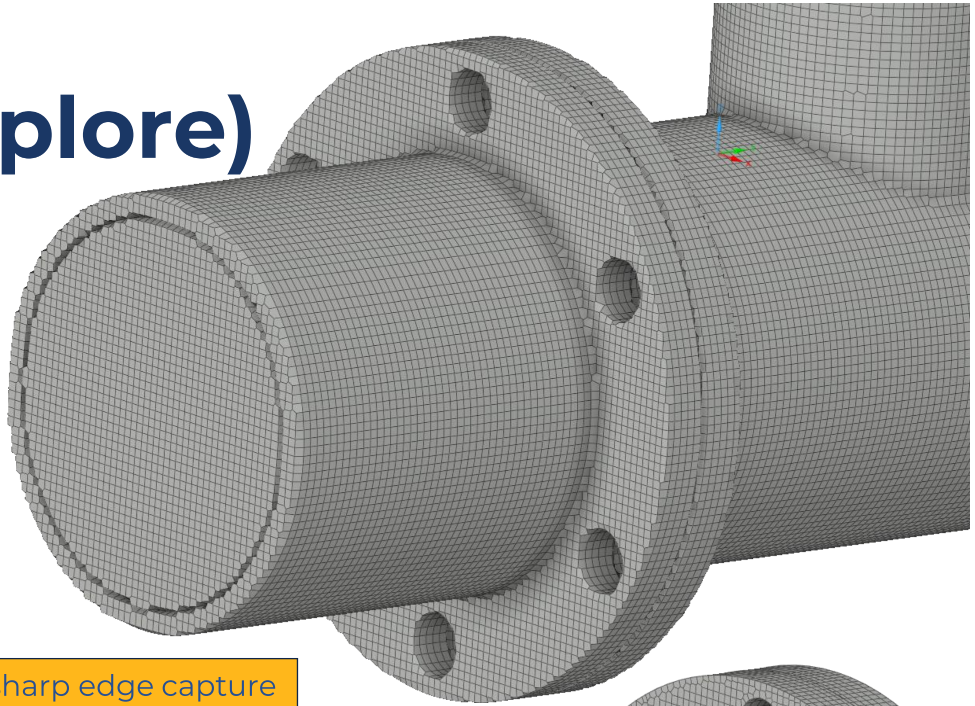




Sharp Edge Capture (Explore)

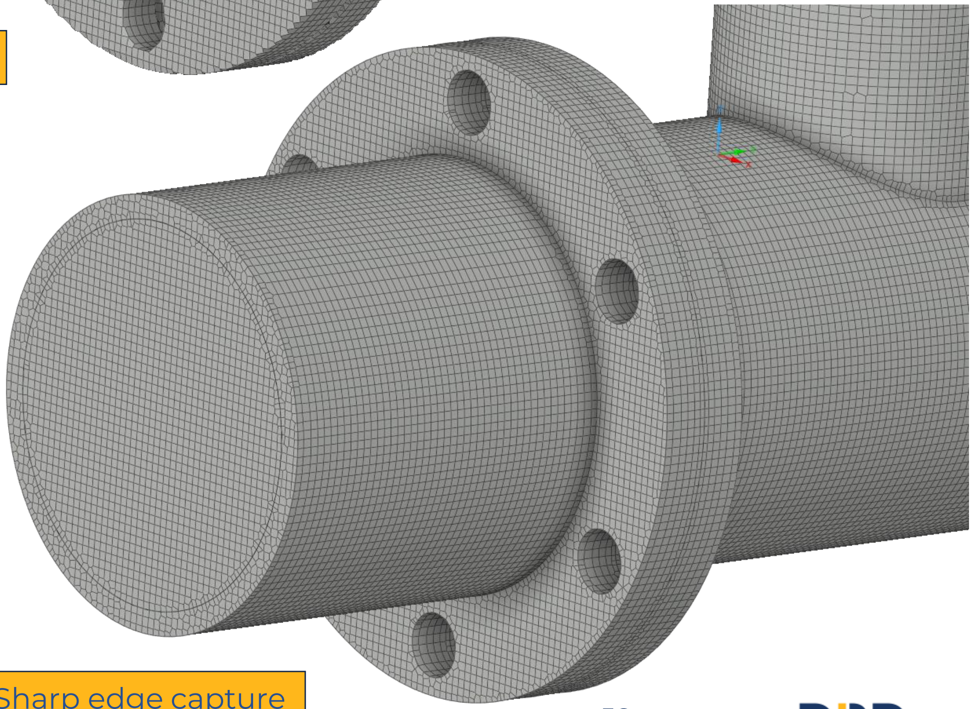
- New option to capture sharp edges for fluids meshing
  - Improves sharp feature resolution
  - Enhances solution accuracy





No sharp edge capture

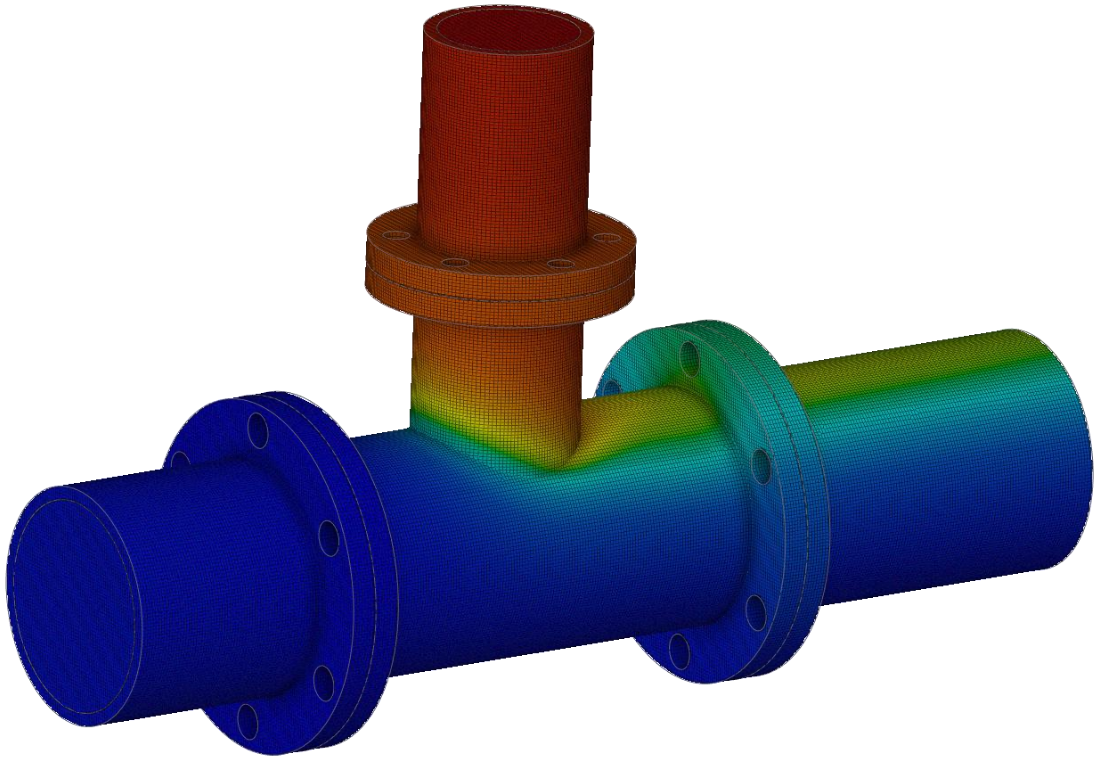
Better resolution of inlets and outlets with sharp edge capture, improves inlet and outlet monitor results

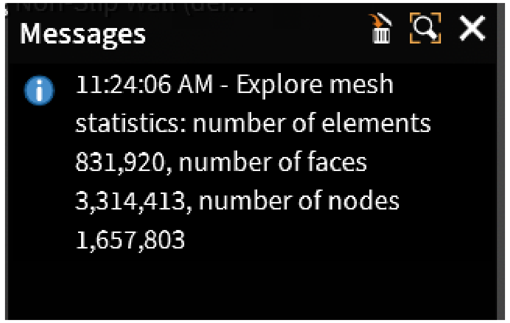


Sharp edge capture

Node and Element Count Messages (Explore)

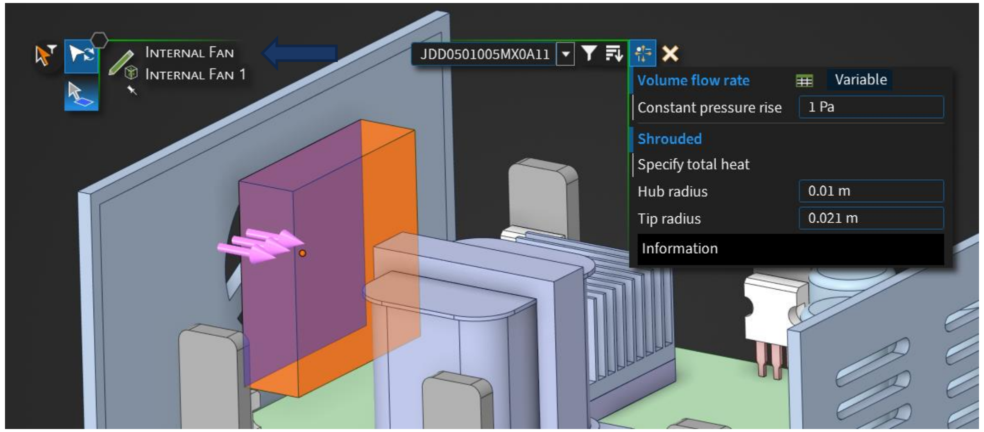
- Node, element and face count for fluid and CHT simulations
  - Message shown as soon as discretization completes
  - Improves ability to determine model size and estimate GPU memory requirements

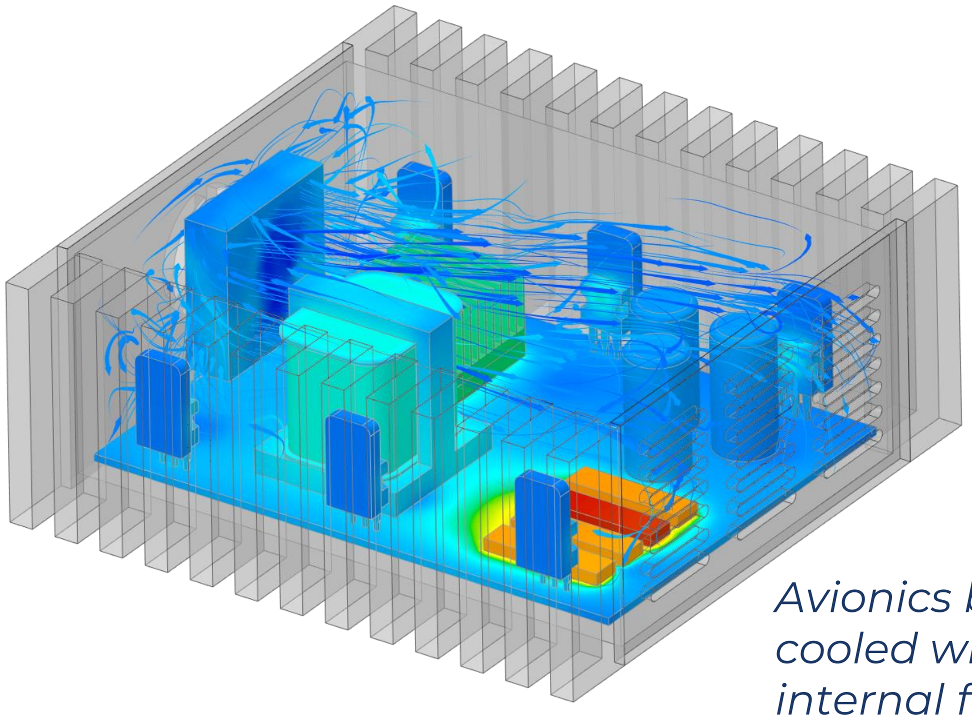




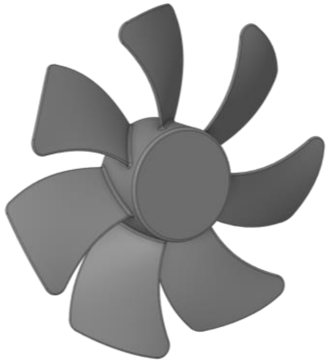
Internal 3D Fans

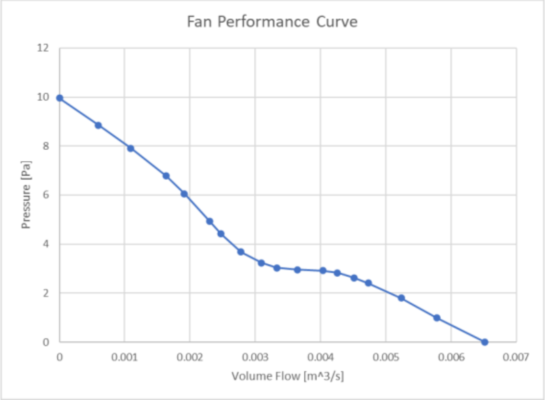
- New 3D internal fan
  - Specify on cylindrical or rectangular fluid bodies
  - Automatic selection of inlet face with user override
  - Fan performance curve or constant pressure rise
  - Fan library with common electronics cooling fans
  - Shrouded or unshrouded fans
  - Monitors for fan operating point
    - Fan pressure rise and volumetric flow rate
  - Enables additional electronics cooling applications





Avionics box cooled with an internal fan

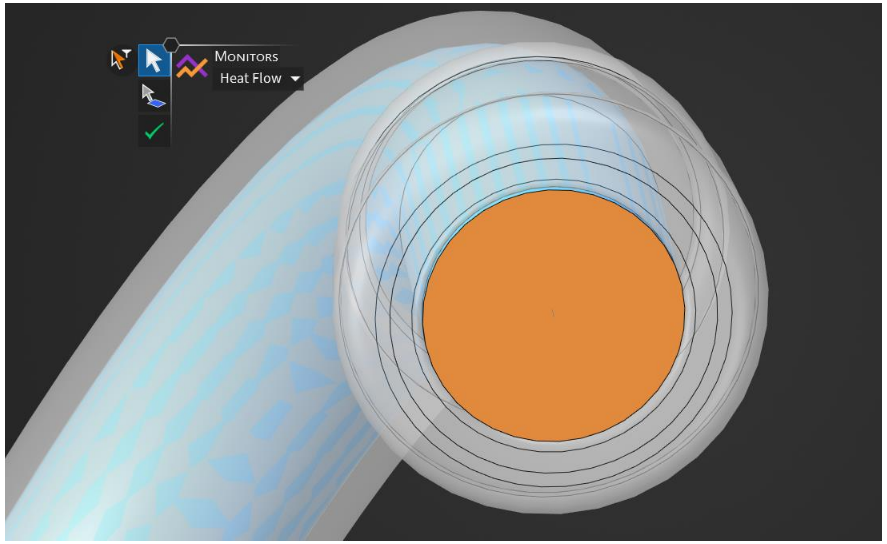


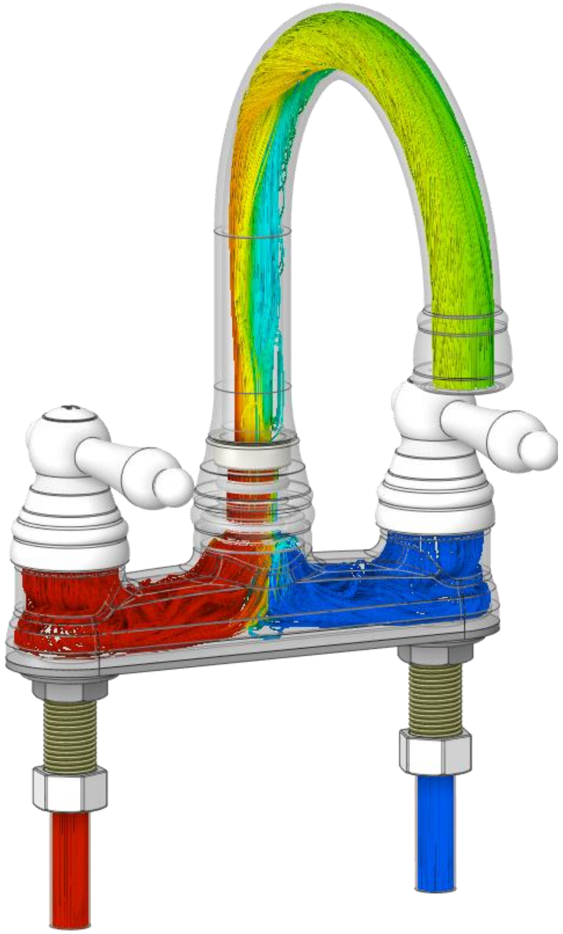


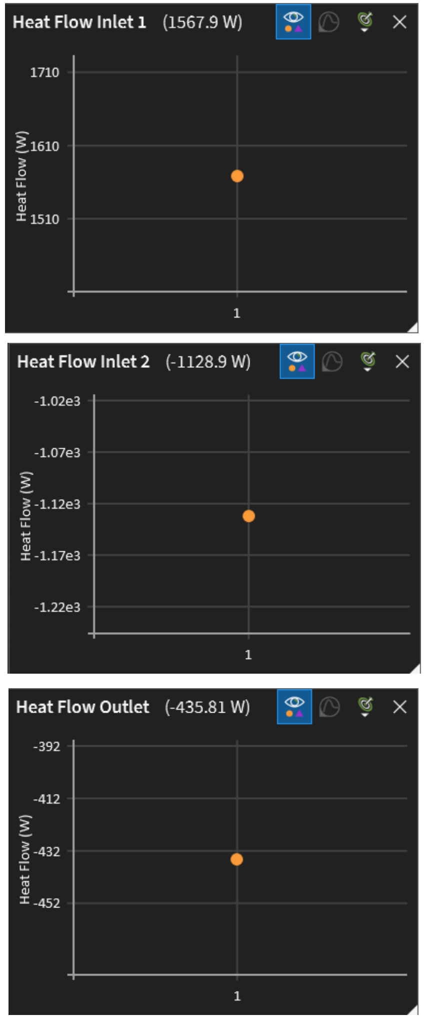
DRD

Heat Flow Monitors (Explore)

- New heat flow monitors on inlet, outlet and walls conditions
  - Improves ability to check heat balance fluid-thermal and CHT simulation



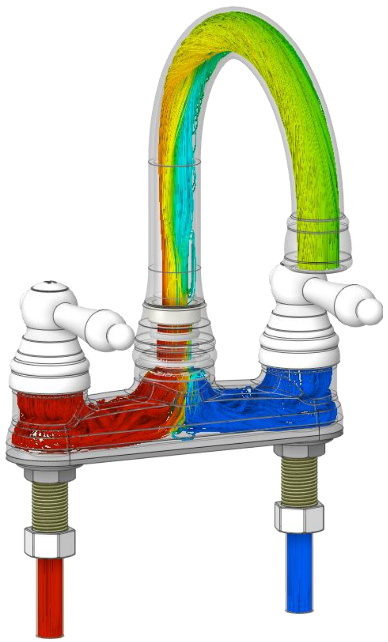


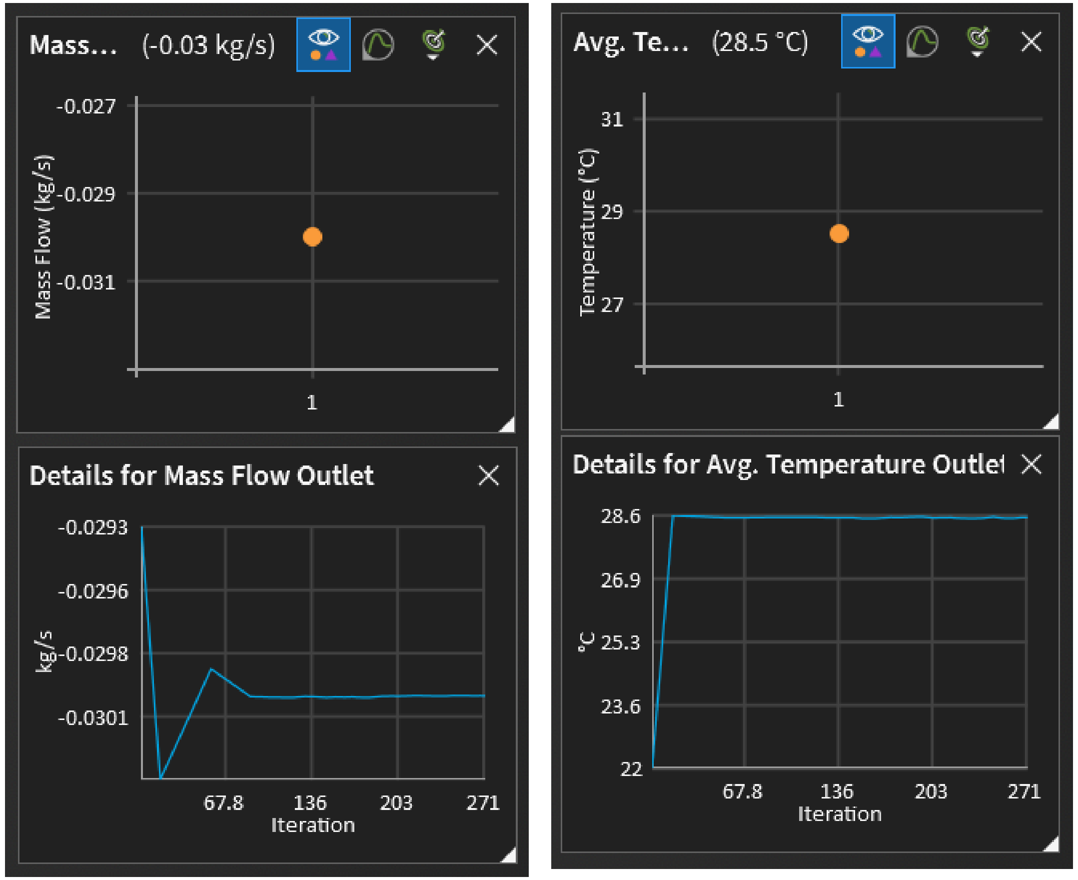


Faucet thermal-fluid simulation, $Q_{in} = Q_{out}$

Monitor Details Saved in Project

- Monitor details now saved in project file
  - Enables review of solution convergence or transient behavior in a saved project





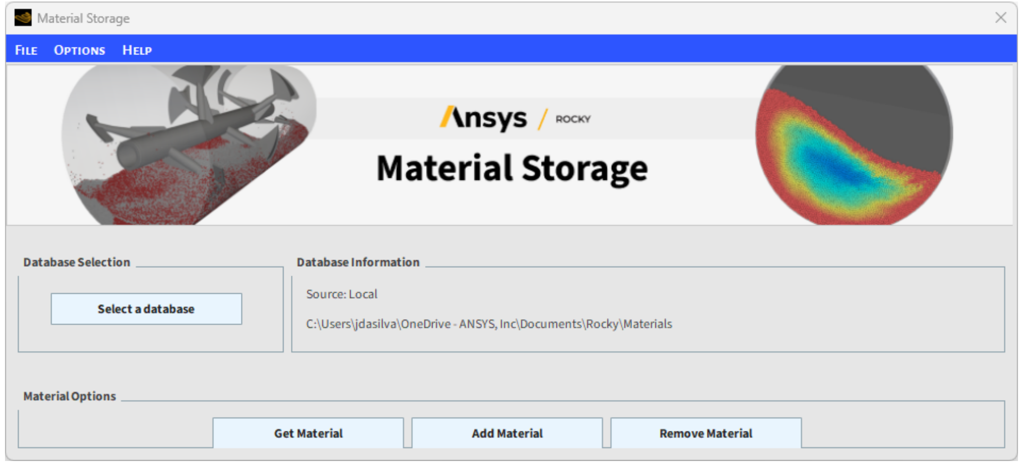
Review monitor details in saved project

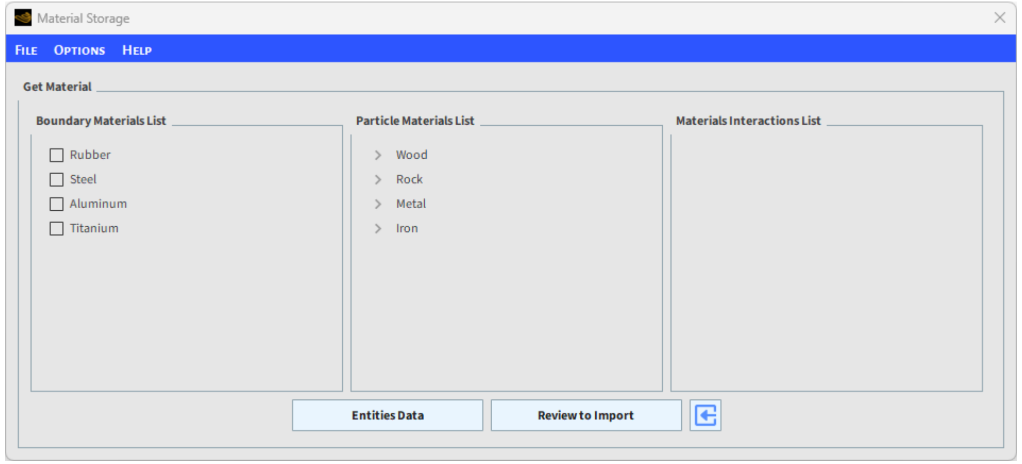
Ansys Rocky

Material Storage

A ready-to-use script, for particle properties management (import, save, share, remove) using a local or Ansys Granta databases.
All properties to be stored, used later, shared with the team, etc.

- Material and Material interaction (particle-particle and particle-boundary, particle shape, size, physics, ...)
  - Reuse calibrated materials
  - Save new materials
  - Fix incorrect or outdated entries
  - Material data preview and editing
  - Share Custom Data Base
  - Supports custom STL (default scale: meters)
  - No support for flexible/breakable particles



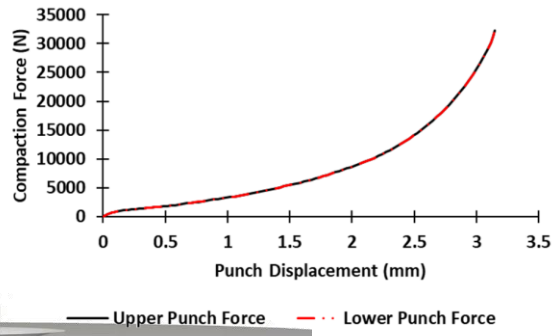


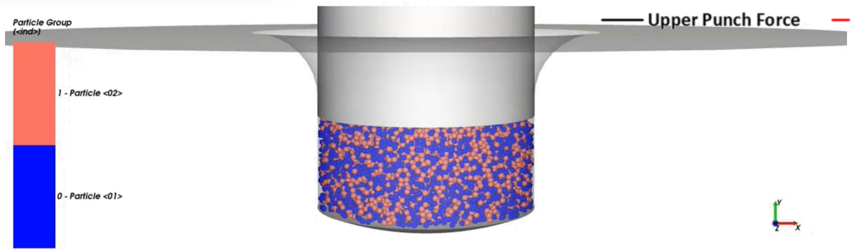
In addition to the existing Material Wizard and Calibration Suite, this is a new feature to facilitate particle data management.

Deformable Particles

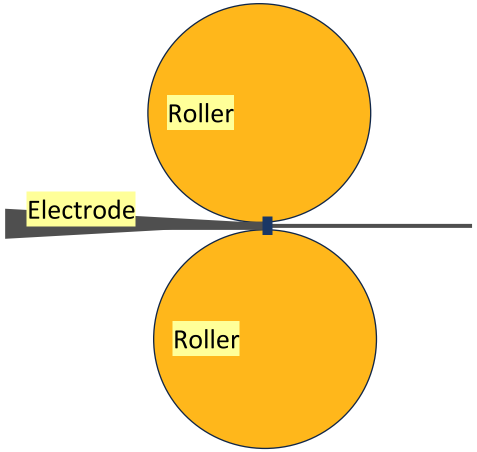
- Beta feature for applications involving significant plastic deformation of particles, such as calendering in battery manufacturing and tablet compaction in pharmaceuticals.
- Enables particle large deformation through an FEM-inspired element deformation approach.
- Discrete Breakage model compatible.

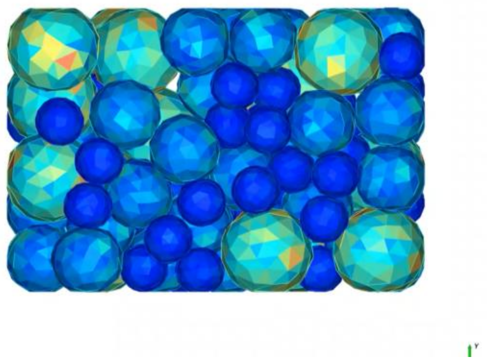
Tablet Compaction





Calendering Process



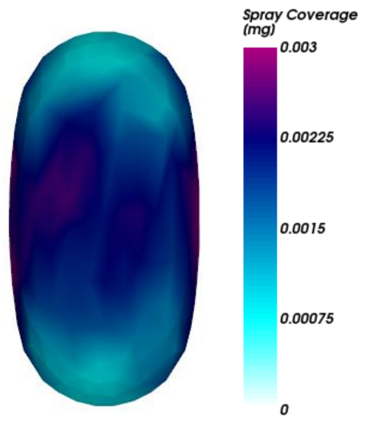


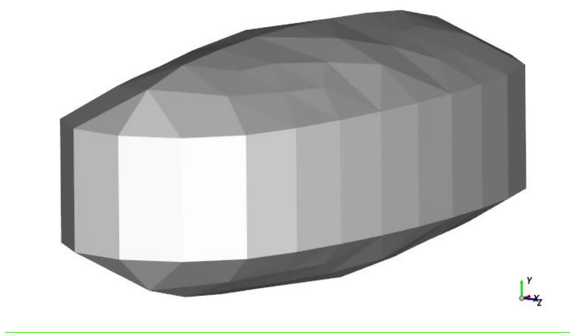
DEM Liquid Spray (coating using DEM particles*)

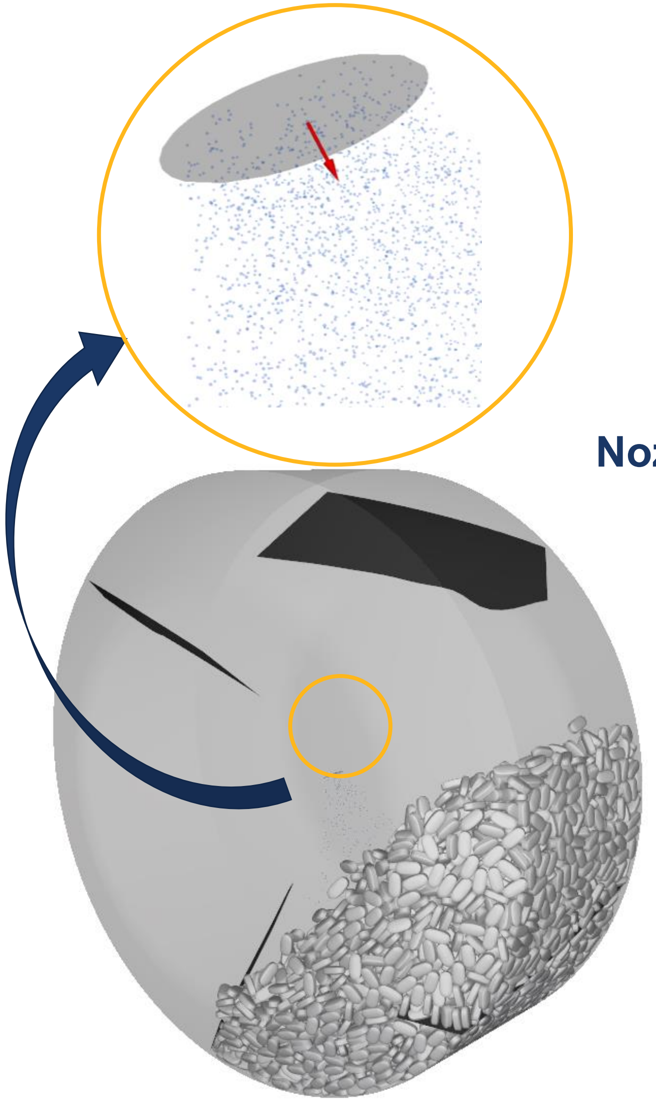
Solver SDK – New ready-to-use module available

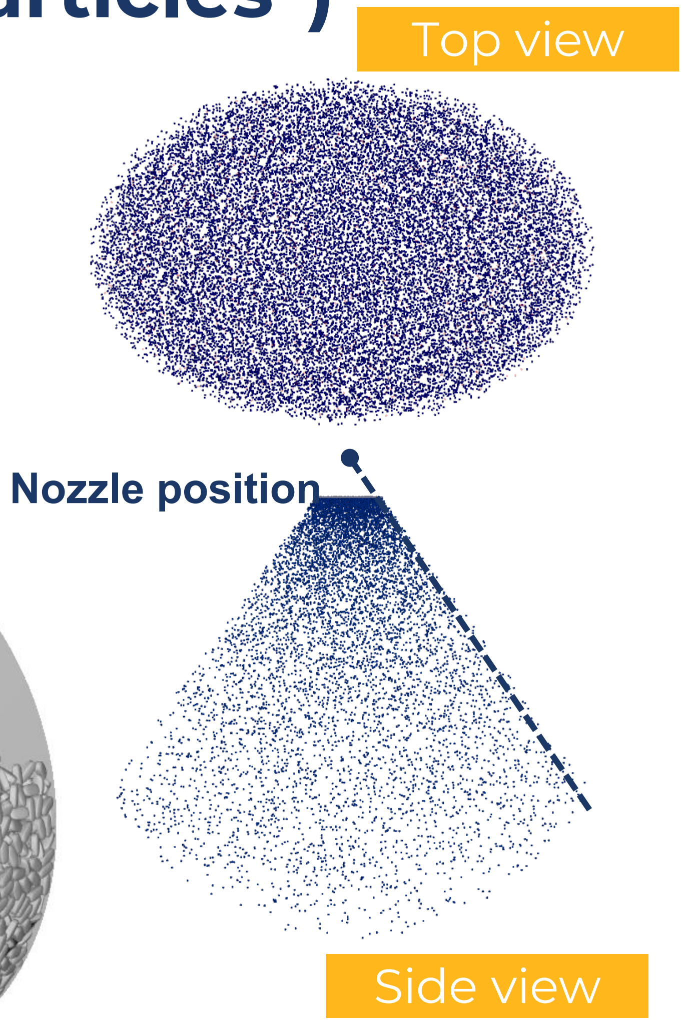
Advantages:

- GPU-compatible
- PSD and CGM can be considered
- Spray pattern (elliptical, spray angle)
- Mass, momentum, energy transfer with tablet and gas flow can be accounted for
- Inter and **Intra-tablet** variability





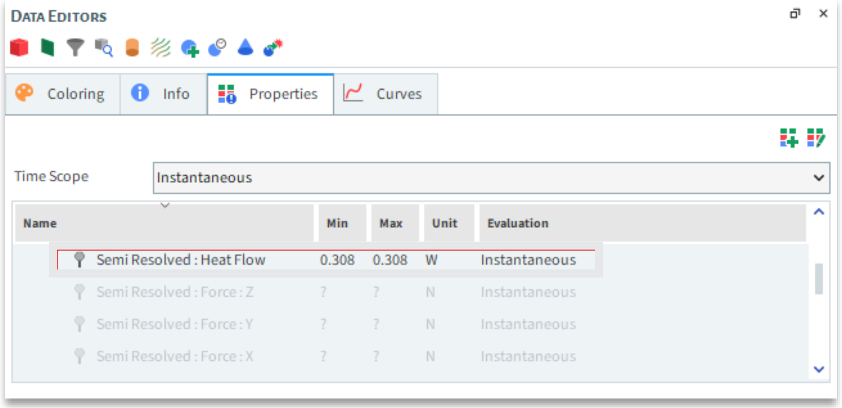


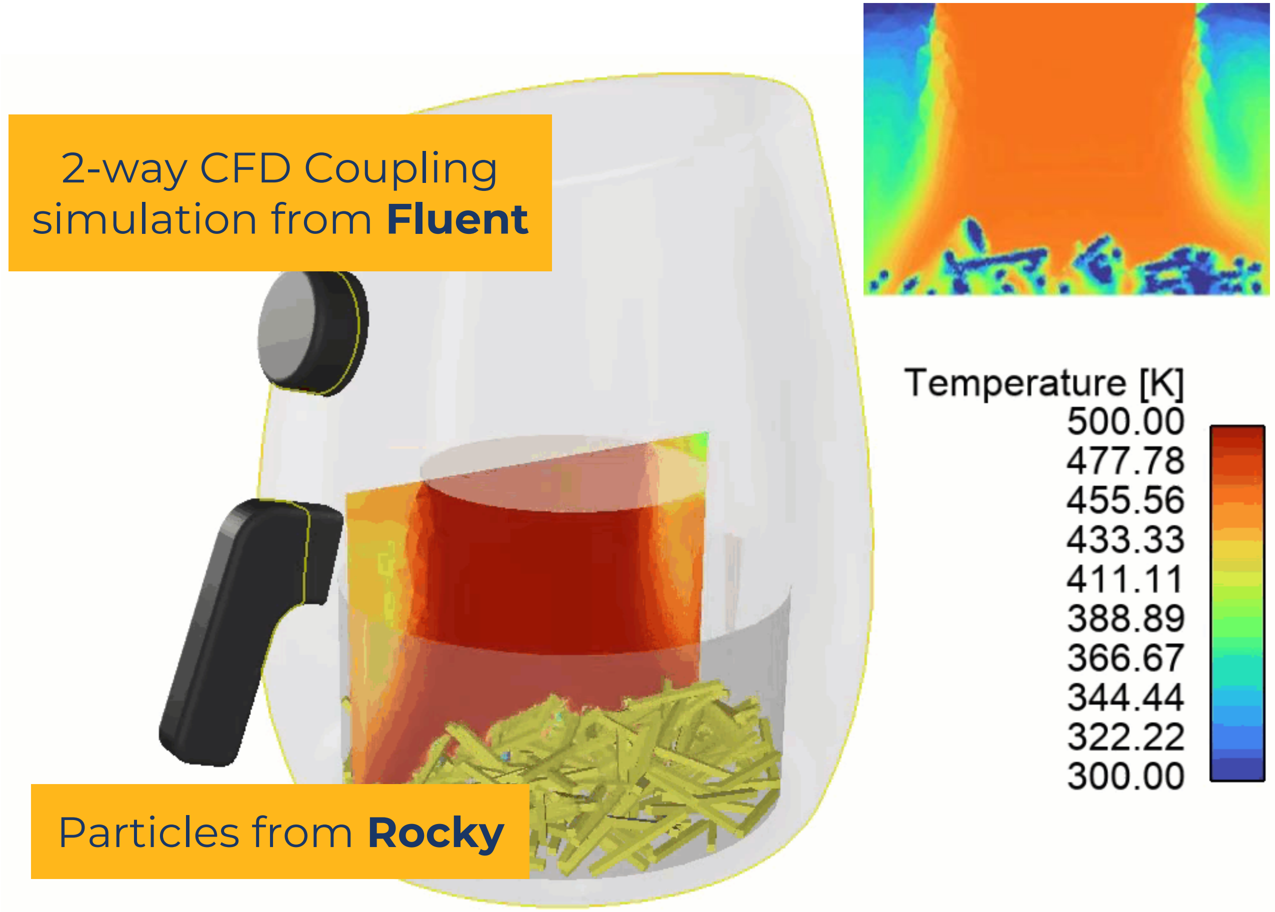


* Lucilla Almeida1, Tim Hoogenraad1, Vinicius Daroz2, Rahul Bharadwaj2, Ariel Muliadi1 - A Novel Approach for Modeling Pharmaceutical Tablet Coating Using Coupled DEM-CFD Modeling. AiChE, 2024

Semi-Resolved Thermal Support

This feature adds support for Heat Exchange between Particle and Fluid in Semi-Resolved coupled Rocky-

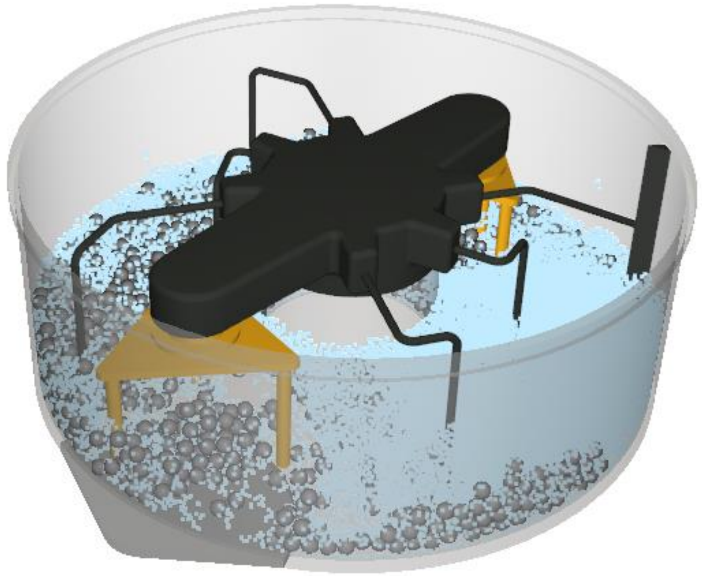




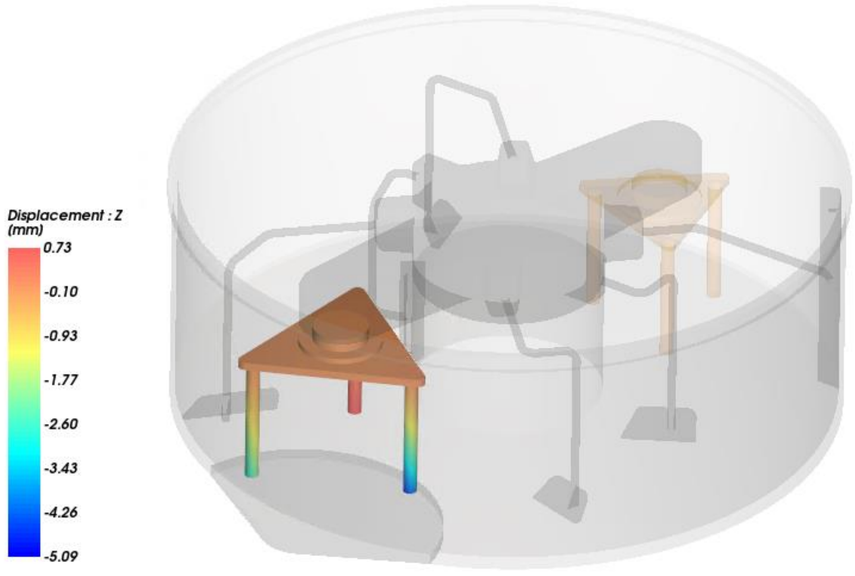
There are two versions of the implementation: one supporting laminar flows and another supporting turbulent flows (modeled with two-equation turbulence models, like the κ-ε and κ-ω turbulence model families).

2-way Structural Coupling DEM/FEA

Rocky sends the **DEM/SPH** forces to **Mechanical**, which calculates deformations and send the **Displacements** back to Rocky.



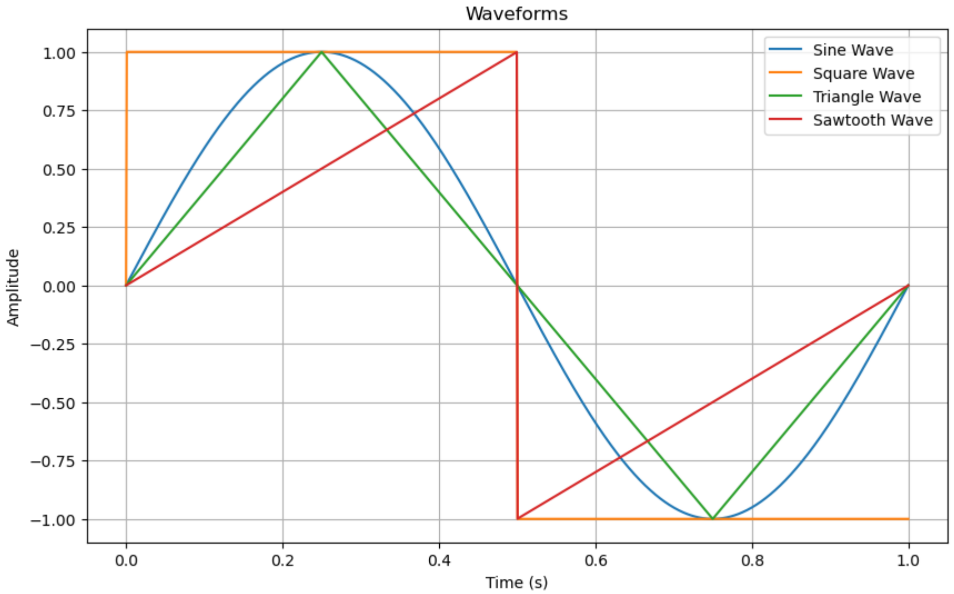
Ansys System Coupling



For this DEM/SPH Mixing example, all the forces acting in the Mixer are sent to Mechanical, which calculates the deformation and sends back the incremental displacement to Rocky. Here, both the DEM/SPH behavior and the Mixer structural integration can be analyzed.

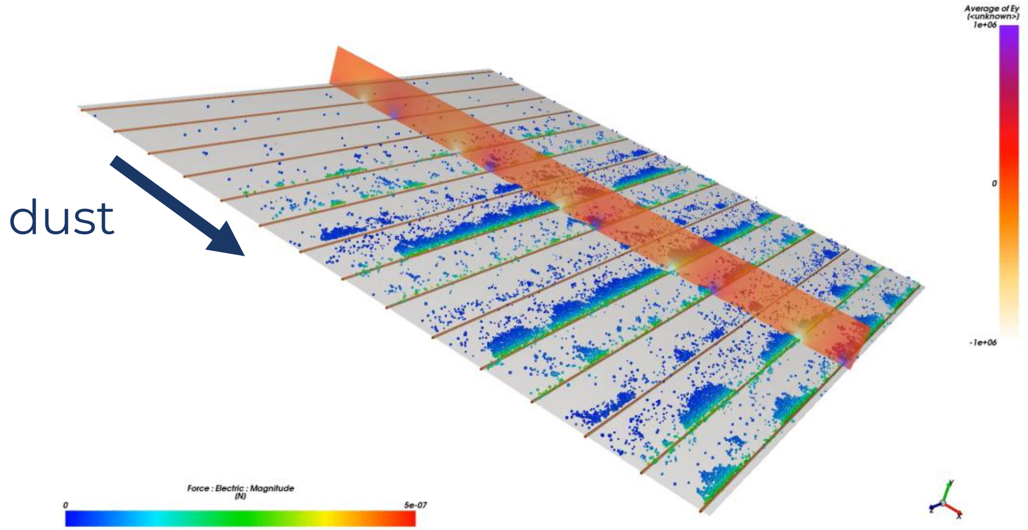
***FSI: Fluid-Structure Interaction**

Electronics Coupling Module (Solver SDK)



$\boldsymbol{k}$	Field Gain [-]
$F(t)$	Field scaling factor [-]
$\boldsymbol{d}$	Vertical displacement [-]
$\boldsymbol{f}$	Modulation Frequency [Hz]
$\boldsymbol{\phi}$	Phase Angle [deg]

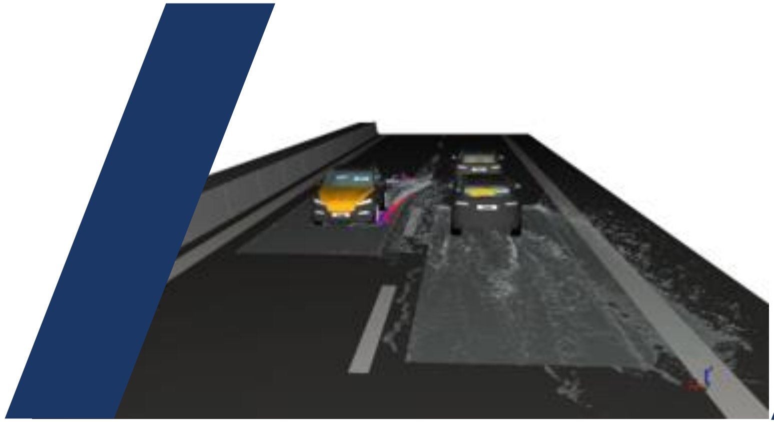
- **It was implemented the field modulation. At this moment, 4 models are available: Sine, Square, Triangle, and Sawtooth**
- Any evaluated field property $\boldsymbol{\psi}$ can be modulated according to one of the available models
- Enables static point clouds to represent transient phenomena

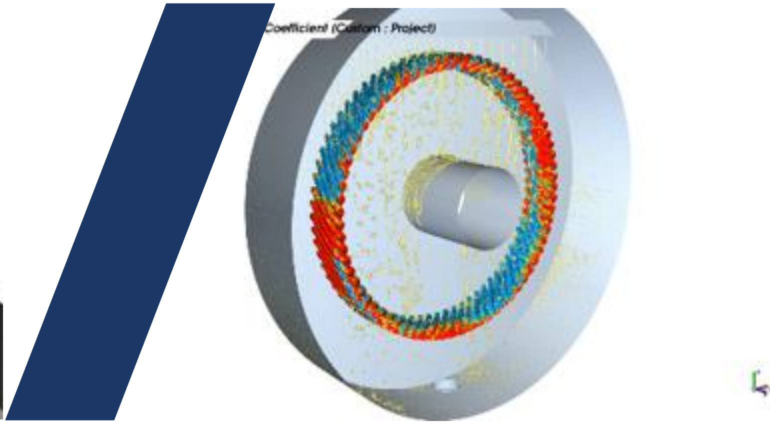


EDS (Electrodynamic Screen) process, for **solar panel cleaning**. Particles (dust) are removed by a **custom electric force** generated from a 3 static point clouds (Ansys Maxwell) and the modulation feature. This was implemented using Rocky Solver SDK.

Ansys Freeflow

Ansys FreeFlow for free-surface flow applications





SPH Features

- ✓ Surface Tension Modeling
- ✓ Multi-GPU
- ✓ Use of Ansys HPC
- ✓ ISPH* Solver
- ✓ WCSHP* Solver
- ✓ Cumulative Wet Time
- ✓ Cumulative Shear Stress

Multiphysics

- ✓ 2-way Coupling with Ansys Mechanical
- ✓ Coupling with Ansys Motion
- ✓ Coupling with Ansys EnSight
- ✓ Expose Force and Heat Transfer Coefficient values

Automation and Customization

- ✓ Modules examples available for downloading
- ✓ Functional modules available, with examples of source code
- ✓ FreeFlow SDK*
- ✓ Powerful Automation PrePost Scripting tool

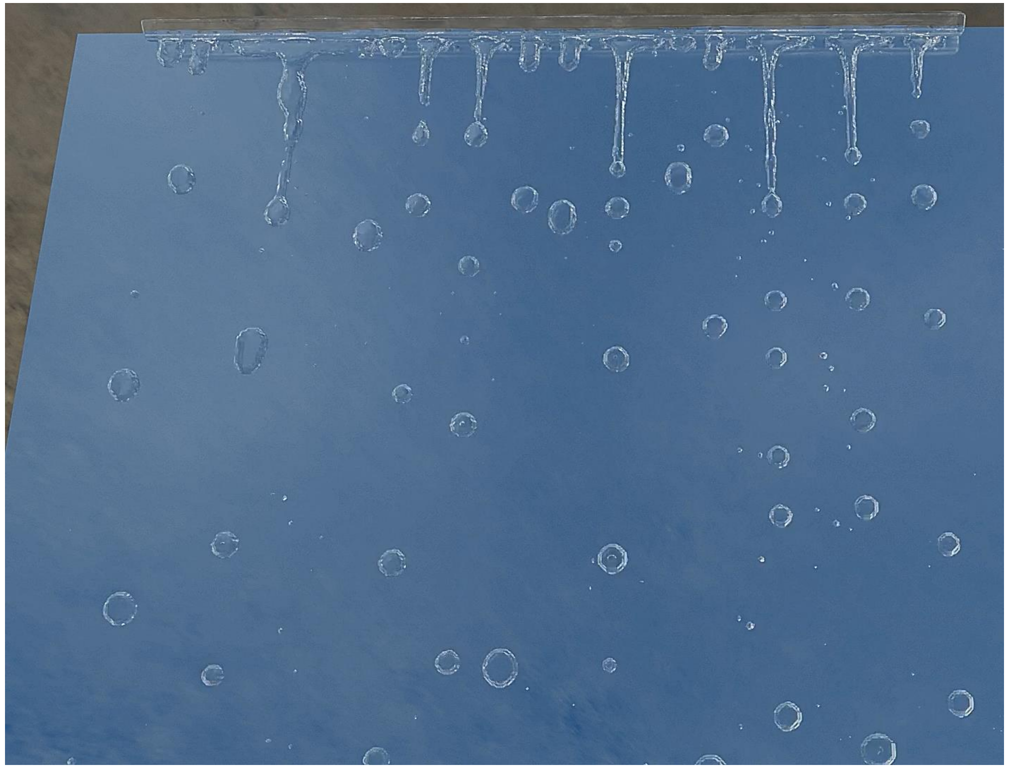
*ISPH: Incompressible Smoothed-particle Hydrodynamics
*WCSPH: Weekly Compressible Smoothed-particle Hydrodynamics
*SDK: Software Development Kit

Surface Tension Model

Pairwise Potential Surface Tension (beta)

Correlate pairwise surface tension parameters with **surface tension** physical parameters.

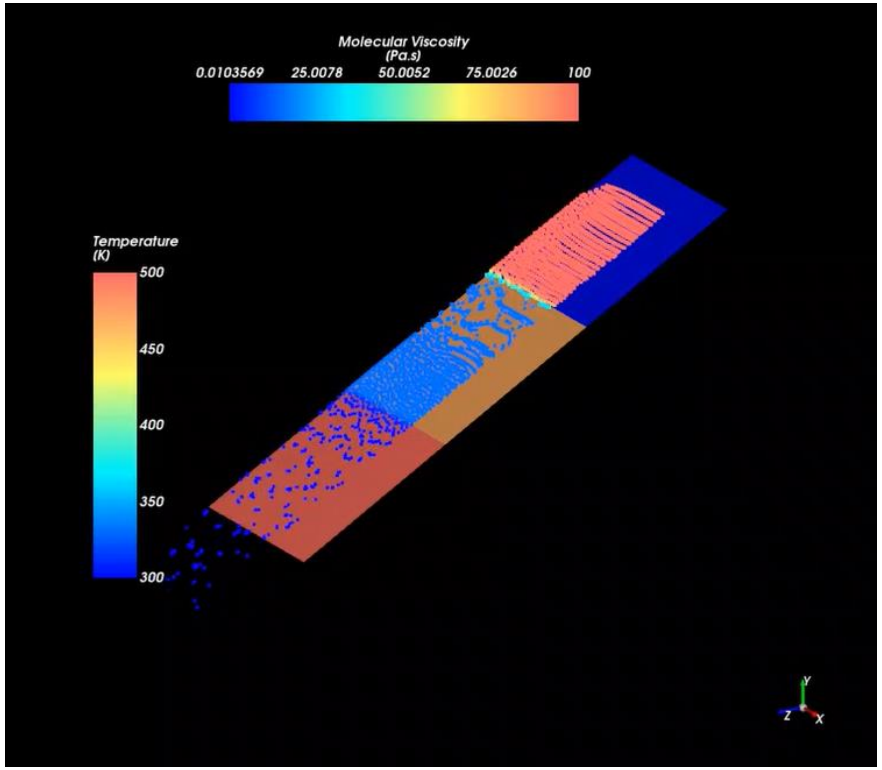
This novel model can be used for applications where the surface tension of fluids needs to be determined with precision, **such as droplets formation and flow over/around geometries.**



Temperature Dependent Viscosity (Beta)

A module in which users import a **.csv file with Temperature X Viscosity** table that FreeFlow uses to set the viscosity of each SPH element.

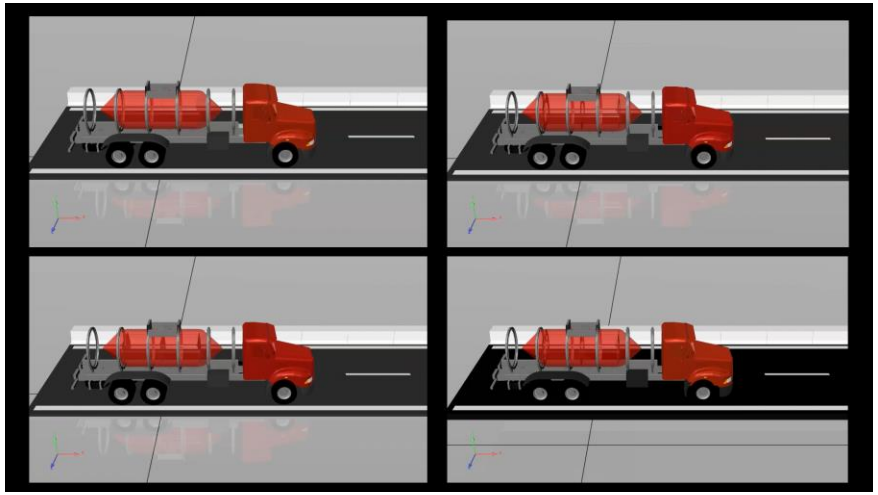
In the **piston gallery simulation case**, it is important to consider viscosity variability, with different temperatures.

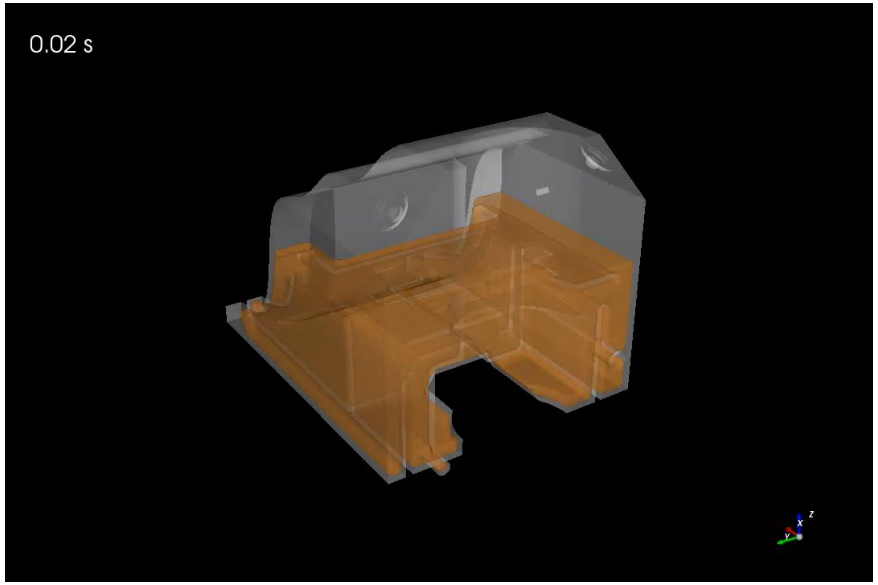


SDK has new hooks so users can calculate their own HTC correlations depending on fluid local velocity and for calculating custom viscosity per SPH element.

Variable Force Field

- User loads a Time X Gravity **table through a .csv file** and gravity will change overtime according to what was defined in the user's table
- Benefit: Simulate tank sloshing of a car in movement without having to move the tank geometry, thus, the gravity changes represent the acceleration due G-Force experienced by the tank.
- **Cases:** Fuel Tank Sloshing.
- **Module's name: variable gravity**





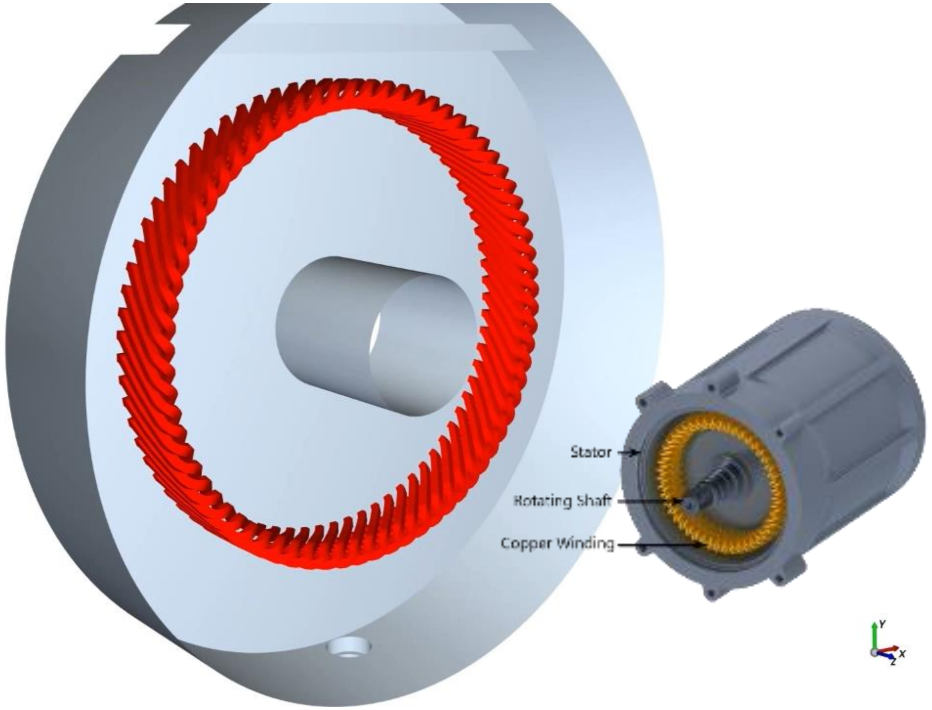
DRD

Heat Transfer Coefficient on Walls

Ansys / MECHANICAL + Ansys / FREEFLOW

- New module that calculates HTC on wall triangles based on correlation dependent on Nusselt Correlations.
- Rocky thermal model not required for HTC calculation.
- **1-way Thermal Coupling** allows users to export **HTC** to **Ansys Mechanical** for further thermal analysis.

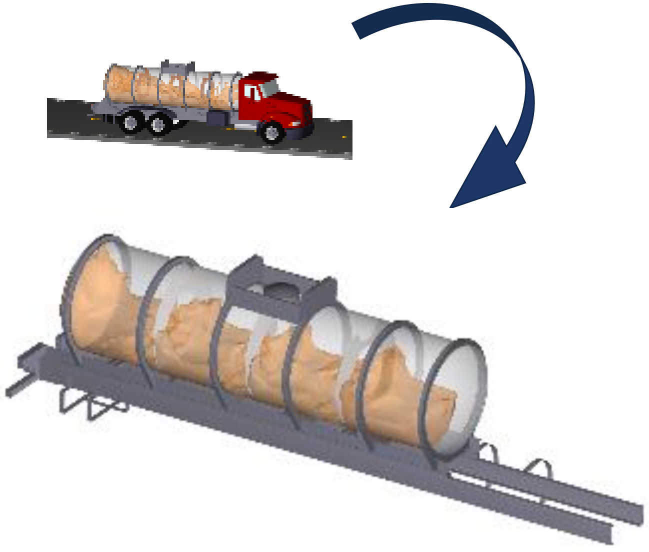
Target Application: E-Motor Cooling



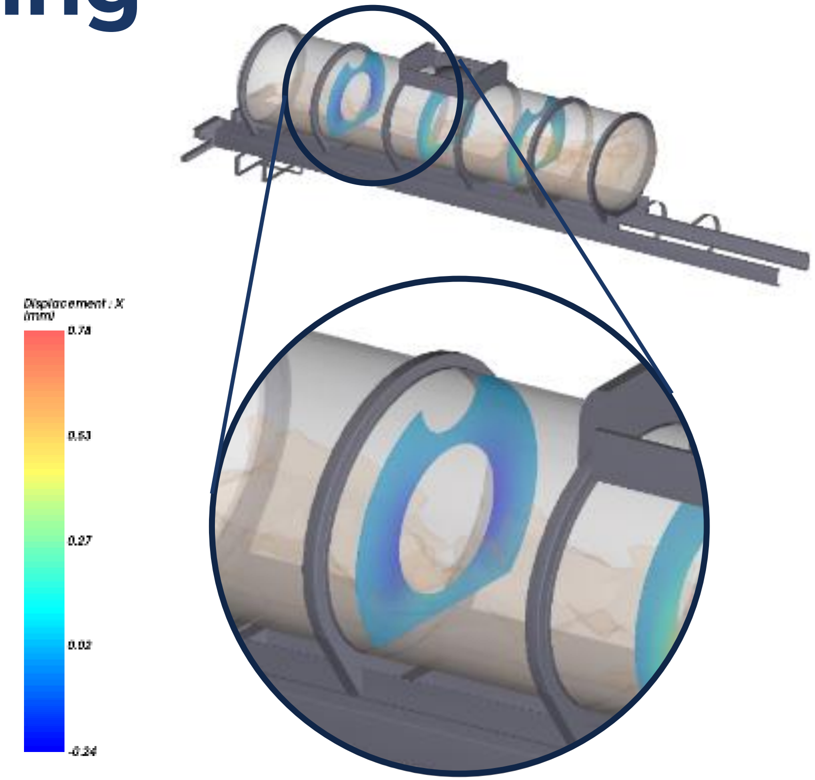
The main objective of e-motor cooling simulations is to evaluate and improve oil distribution for different cooling branches and to maximize heat transfer between the oil and the e-motor components.

Ansys Mechanical 2-Way Coupling

FreeFlow sends the **SPH** forces to **Mechanical**, which calculates deformations and send the **Displacements** back to FreeFlow.



Ansys System Coupling



For this **2-way Structural** Fuel Tank Sloshing **FSI*** example, all the forces acting in the Tank are sent to **Ansys Mechanical**, which calculates the deformation and sends back the incremental displacement to FreeFlow. Here, the SPH behavior and the Tank structural integration can be analyzed.

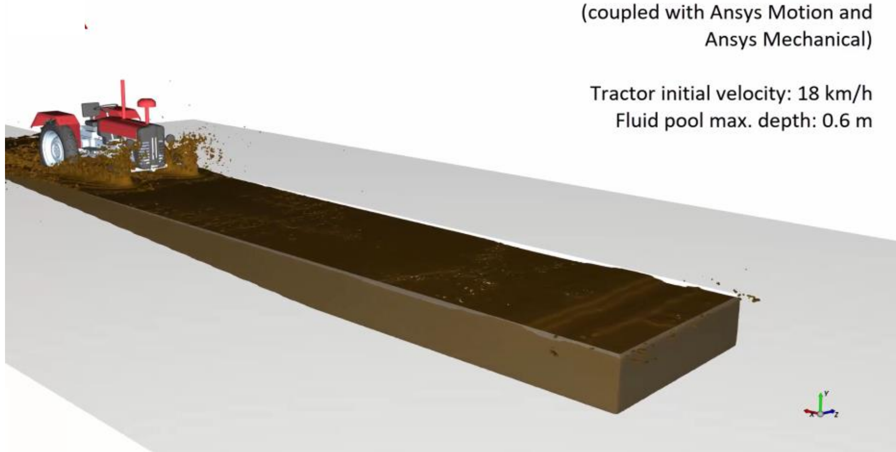
Similar to what here have for the Structural Coupling, FreeFlow also allows the **2-way Thermal Coupling** with Mechanical.

***FSI: Fluid-Structure Interaction**

DRD

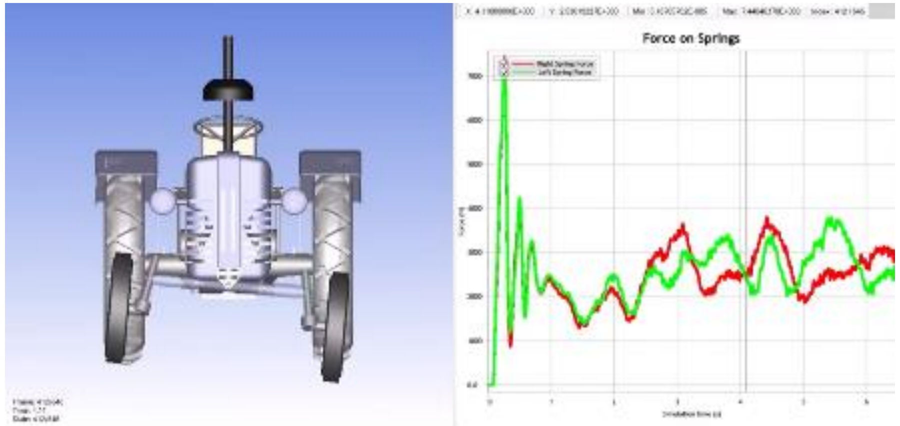
Ansys Motion 2-Way Coupling

Ansys / MOTION + Ansys / FREEFLOW

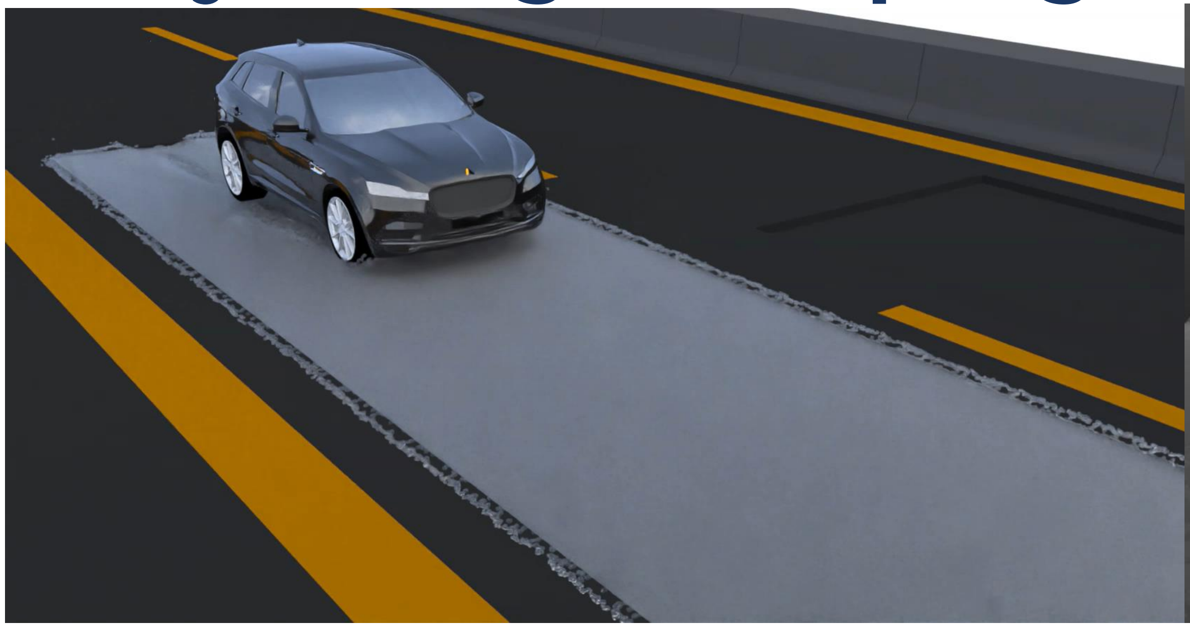


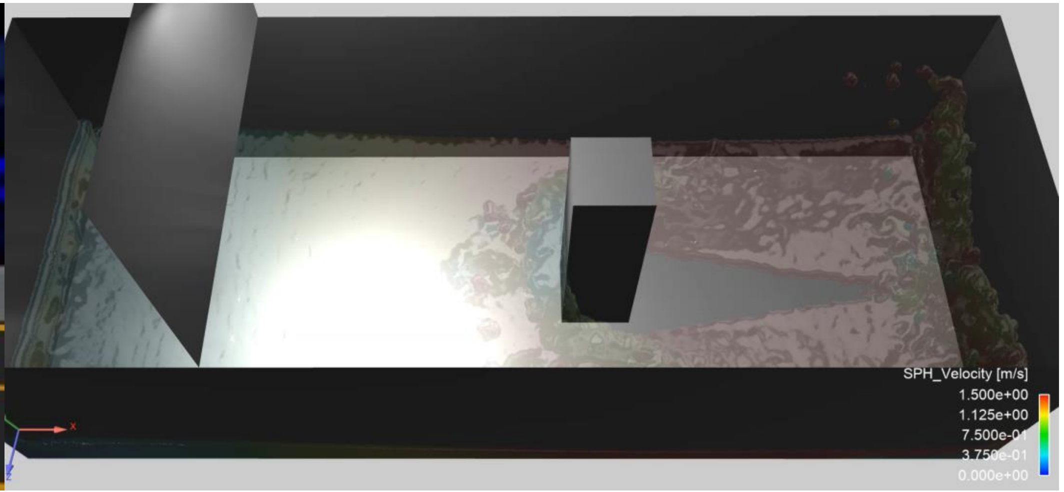
Ansys Motion co-simulation enables:

- Complex nested and chained motions
- Stress and deformation of flexible body components
- Geometry-to-Geometry Interaction
- Advanced Mechanisms Logics and Behaviors (PLC, Matlab, ...)
- Coupling using FMU Interface for data exchange

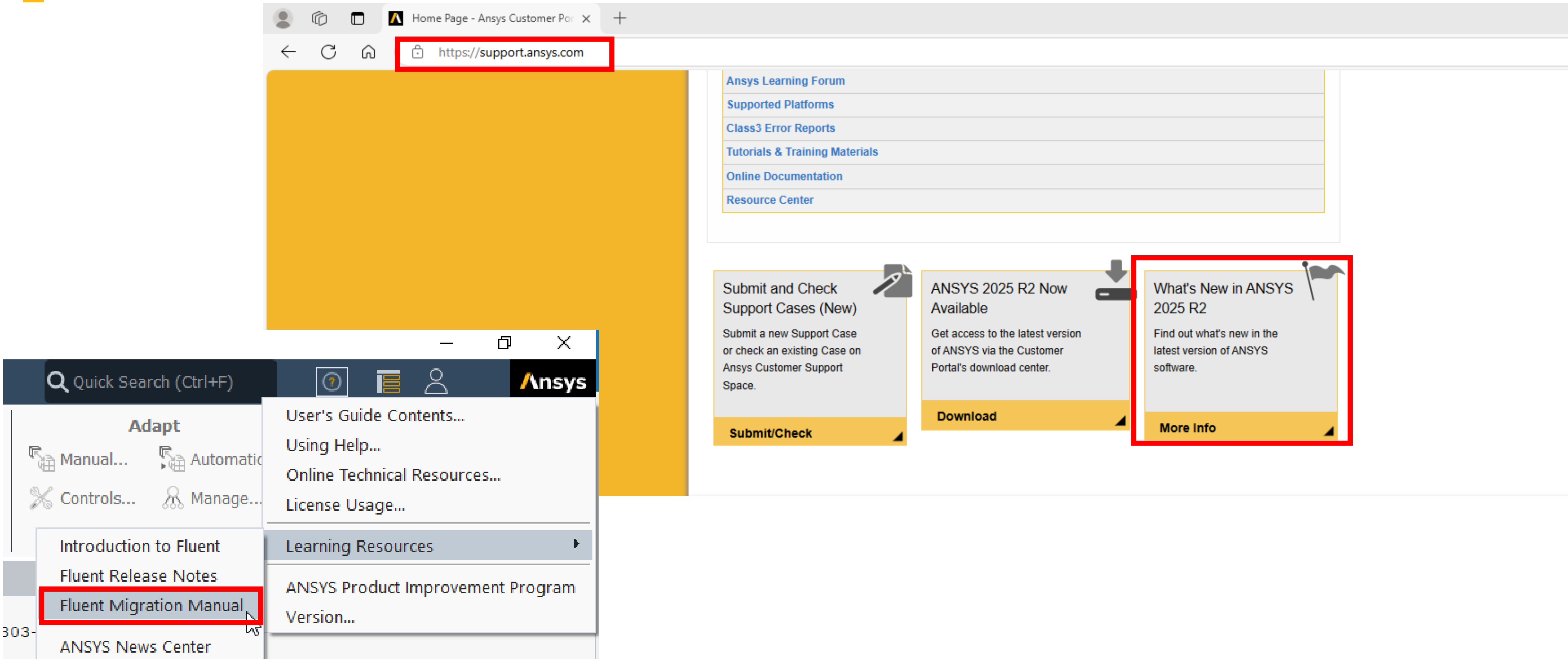


Ansys EnSight Coupling





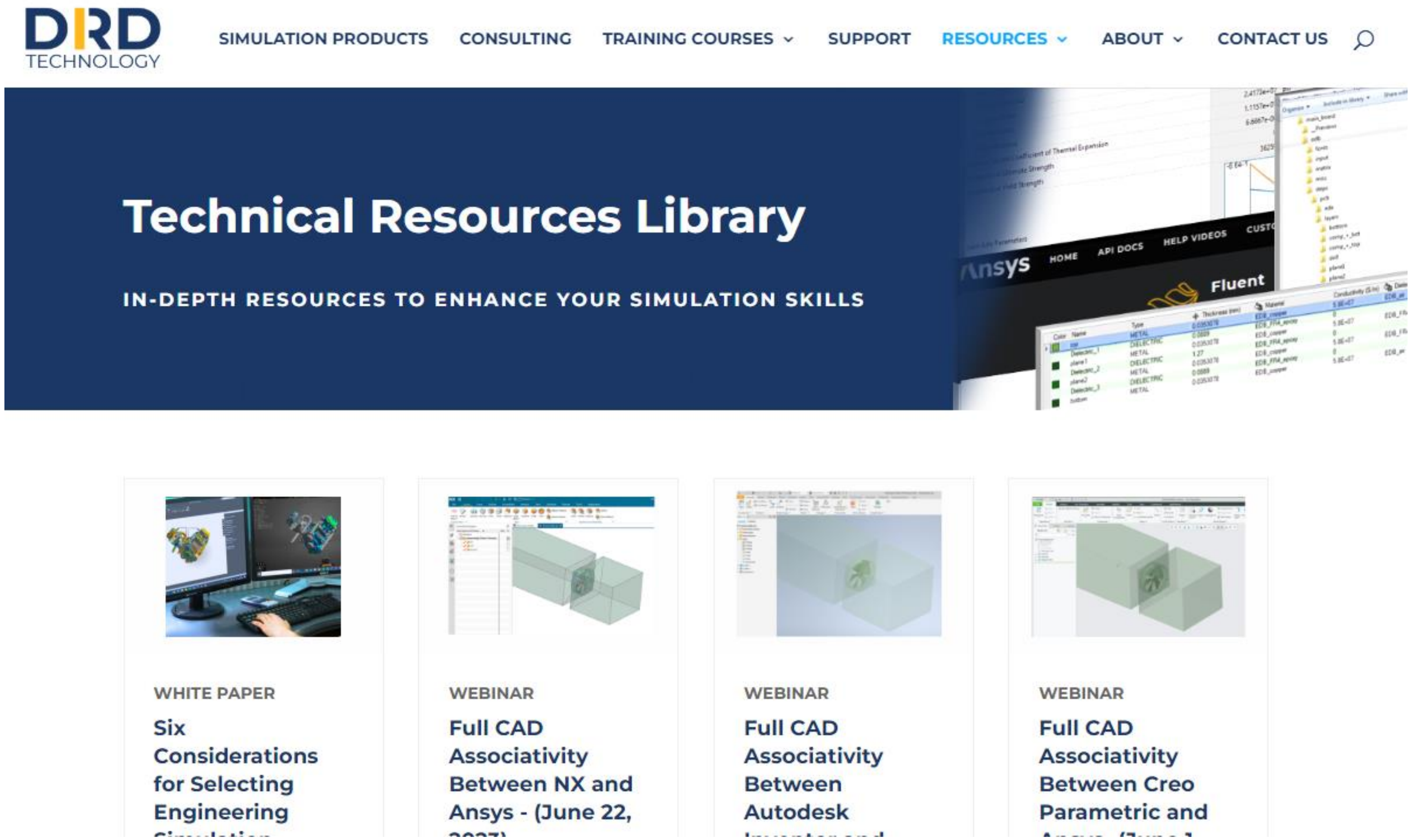
More Updates



Wrap Up

The recording and slides for this webinar are in our Technical Resources Library.

If you are not on our mailing list, or are unsure if you are, please let us know at support@drd.com and we can add you!



Wrap Up

Thank you for your attention!

May I address any questions?