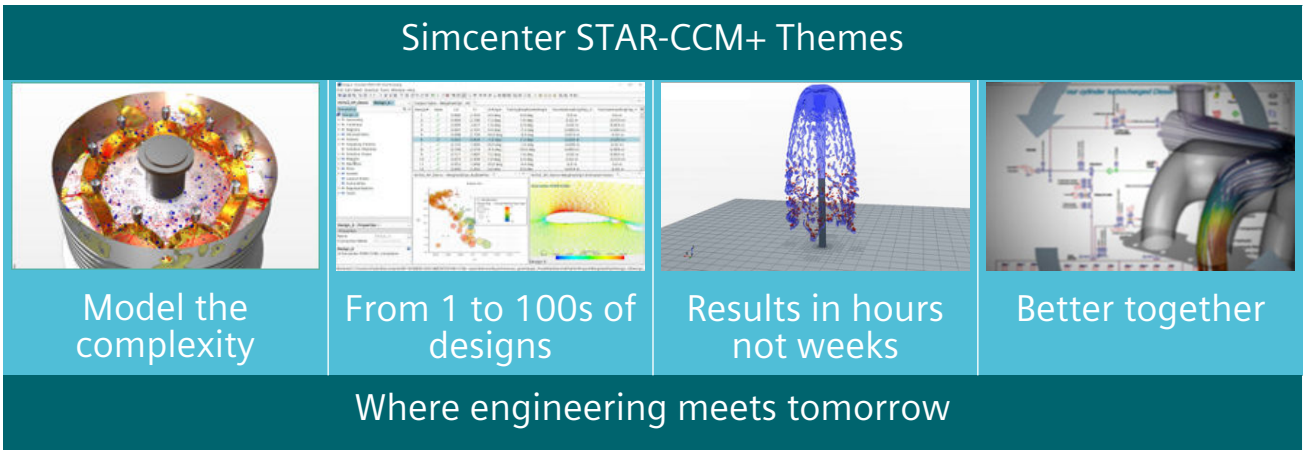

SIEMENS
Ingenuity for life
Version
2020.3
Siemens Digital Industries Software
Simcenter STAR-CCM+
New features and enhancements
Where engineering meets tomorrow.
www.siemens.com/mdx

New Features and Enhancements in Simcenter STAR-CCM+ 2020.3



Top new features and enhancements for this release are:

- gPROMS multizonal coupling [1]
- 3D-CAD Visualization speedup
- Validation of Simulation Operations
- Adjoint based topology optimization [1]
- Asynchronous post-processing in Design Manager [1]
- Scale-Resolving Hybrid turbulence model [1]
- VOF to Lagrangian transition [1]
- Mixture Multiphase - Large Scale Interface Model
- FE Excitation coil model for EMAG
- Liquid Film for Simcenter STAR-CCM+ In-Cylinder

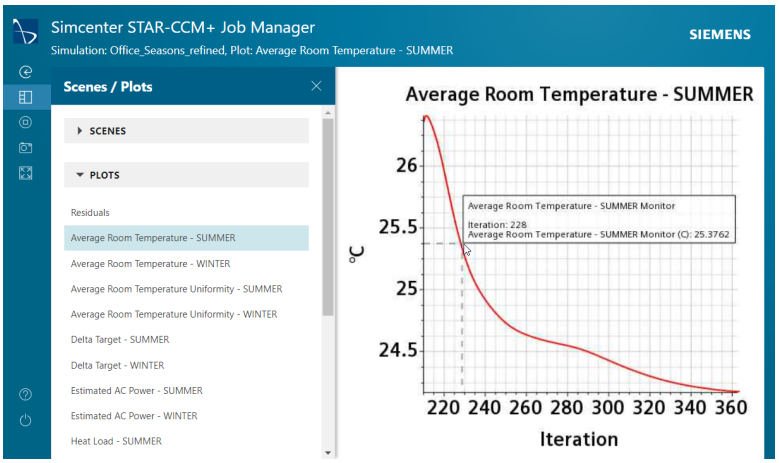
1 Posted on IdeaStorm
A total of 29 new features and enhancements from IdeaStorm in this version.

Enhancements to Simcenter STAR-CCM+ 2020.3 are presented by category:

Platform

Deployment

- **Job Manager updates**
  - **Installation process is now more IT friendly**
    - The default user and group information is no longer automatically populated
    - Blank fields to be manually filled out with the proper information to avoid likelihood of errors
  - **Job Manager URL published after installation**
    - Information provided right after the installation to facilitate its communication to end users
    - Default user and group information not automatically populated during installation
      - Blank fields to be manually filled out with the proper information to avoid likelihood of errors
- **Web Monitor plots improvements**
  - Gain an immediate grasp of your simulation progress with the residuals plot opening automatically when Web Monitor is launched
  - Explore plots to deepen understanding of the results and make better decisions
    - Look at small changes by zooming with rubber band
    - Find trends easier by panning
    - Get an unobstructed view of information by moving the legend

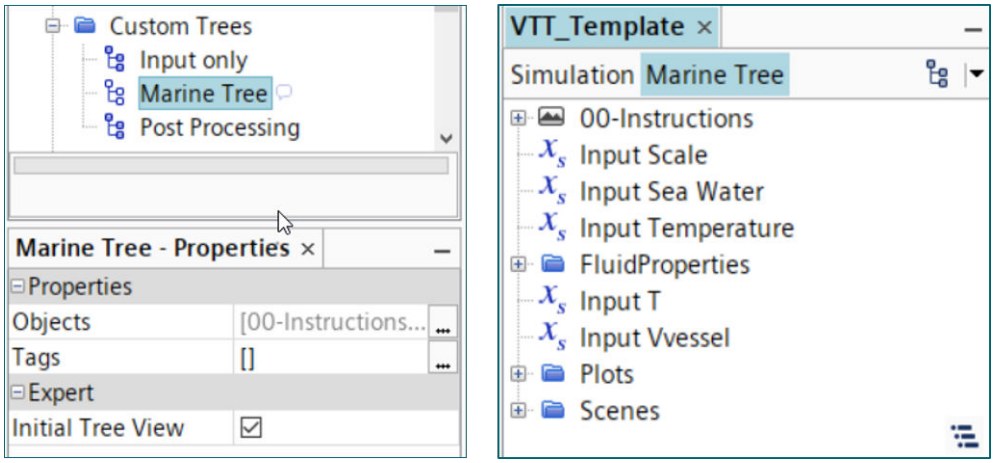


- **Newly certified operating systems (OS)**

    - CentOS/RHEL 8.1
    - Windows 10 Version 1908
- **Newly supported operating systems (OS)**
    - SLES 12 SP5
- **Retired operating systems (OS)**
    - CentOS/RHEL 6.10
- **Planned operating systems (OS) changes for 2021.1**
    - Adding RHEL 7.8, RHEL 8.2, OpenSUSE 15.2, Windows 10 Version 2004
    - Retiring CentOS/RHEL 7.4, OpenSUSE 15, OpenSUSE Leap 42.3
- **Newly certified Message Passing Interface (MPI) versions**
    - Linux
        - Open MPI 4.0.3 (new default)
        - Intel MPI 2019.7, Open MPI 3.1.6
    - Windows
        - MS MPI 10.1.1 (new default)
        - Intel MPI 2019.5
- **Retired Message Passing Interface (MPI) versions**
    - Linux
        - Intel MPI 2019.5, Open MPI 3.1.3
    - Windows
        - MS MPI 10.0, 8.1.1, Intel MPI 2018.1
- **Planned Message Passing Interface (MPI) changes for 2021.1**
    - Linux
        - Intel MPI 2019.8
    - Windows
        - Intel MPI 2019.7, Intel MPI 2019.8

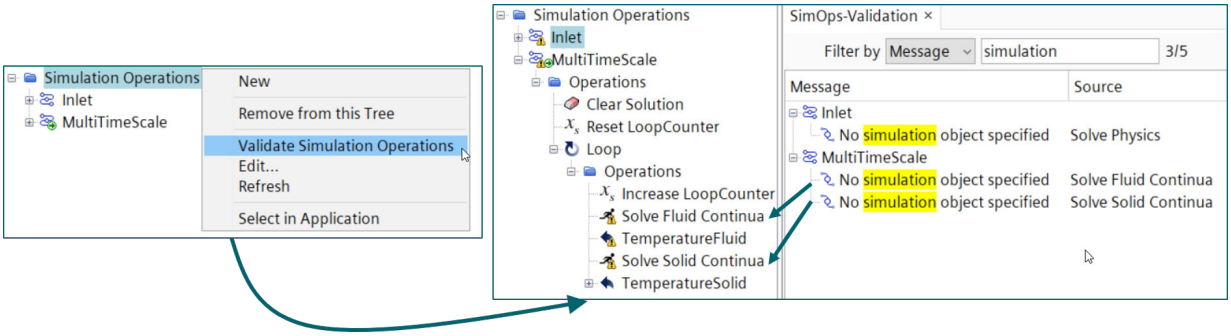
User Experience

- **Use Custom Tree as your initial tree view on simulation load**
    - Easily deploy robust and reproducible workflows
        - Apply consistency across many users across an organization
        - Enable improved productivity for many by leveraging expertise
    - For the Custom Tree authors
        - Create streamlined workflows
    - For the Custom Tree users
        - Use streamlined workflows
        - Improve ability to focus on relevant information
        - Easily navigate the trim Custom Tree



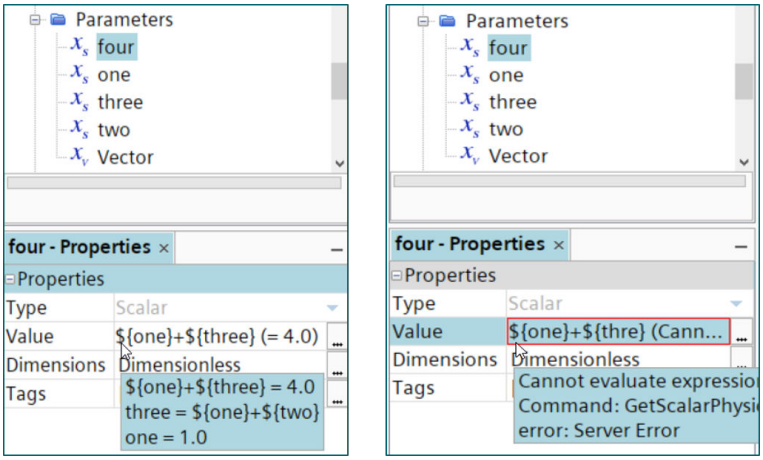
  - Important note: Support for the Custom Tree initial view at startup is planned for a later release for Design Manager

- **Simulation Operations: Validation**
  - Enhance productivity by catching errors in setup
    - Improve usability, ease of use and increase user confidence
  - Dynamic study validation with error badging
    - Scans set-up to identify missing information or errors and flags them with a warning badge
    - Explicit user guidance on how to fix the issues provided in a warning window
    - Reduce your time spent to modify setup with warnings linked to related tree object
    - Manually validate operations before a batch run or let the validation run automatically when playing the workflow

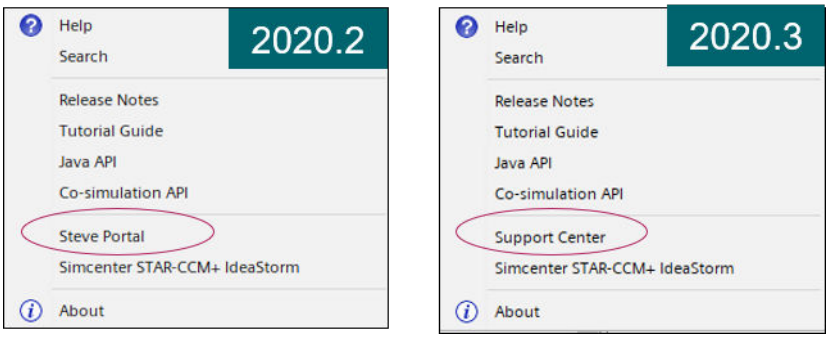


- **Simulation Operations improvements**
  - Immediate grasp of simulation progress with the residuals plot opening automatically when Solve Continua action is executed

- **Global Parameter validation and value preview D4773**
  - Improved ease of use for global parameters
    - Reduced time to set up and verify values
    - Quick expression check
    - Preview parameter values
  - Easier to obtain the value of a parameter
    - View evaluated expression for scalars and vectors
    - Expression validation and error highlighting to draw attention to incorrect expressions
    - Understand results and errors immediately with hints



- **Help menu updated**
  - Reflects transition to the new online Support Center



- **OpenJDK updated to version 11.0.6**
  - Users should not see any differences from the previous release
- **Simulation Assistant Wizard Retirement**
  - Simulation Assistant Wizard (NetBeansWizard.nbm) is planned to be removed in version 2021.3
    - The Simulation Assistant Wizard creates a template Simulation Assistant project
    - Simulation Assistants will have to be created manually
    - Your existing Simulation Assistants will continue to run as expected

CAD Integration

CAD Clients

- **CAD Client upgrades**
  - NX 2020.2 (1926 series)
  - Simcenter 3D 2020.2
  - CATIA R30
  - Creo 7.0
  - Inventor 2020
- **Hidden geometries are no longer transferred by default when using Client for Creo**
  - The default behavior has changed but it can be restored within the options
- **Planned end of CAE mode for CAD Clients**
  - Starting 2021.1

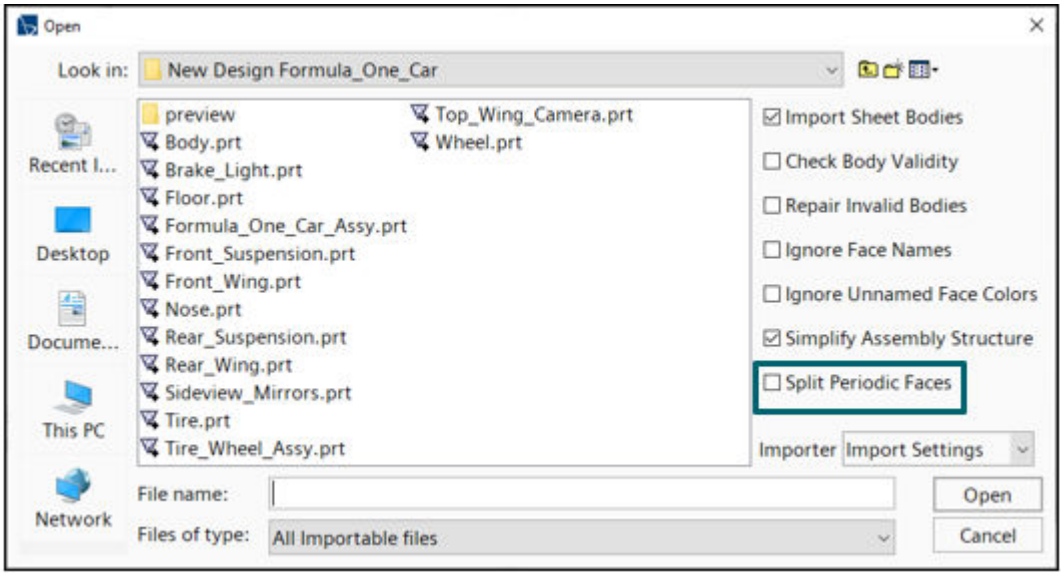
- The CAE mode (ability to setup a simulation in the CAD package environment) will be retired for all but CATIA V5
- CAD Clients are intended to solely focus on direct geometry transfer and bi-directional CAD information transfer including parameters, expressions, coordinate systems, and reference planes

CAD Exchange

- **CAD Exchange upgrades**
  - NX
    - Siemens Adapter
      - Windows - Up to NX1899
      - Linux - Up to NX12
    - HOOPS Exchange
      - NX 2020.2 (1926 series)
  - Solid Edge
    - Siemens Adapter
      - Windows - Up to 2020
      - Linux - No Support
    - HOOPS Exchange
      - Up to 2020
  - CATIA
    - Siemens Adapter
      - Up to V5-6 R2019 SP4
  - JT
    - Siemens Adapter
      - Up to 10.6.1.0
  - STEP
    - Siemens Adapter
      - AP 203, AP 214, AP 242
  - SolidWorks
    - HOOPS Exchange
      - Up to 2020
  - Parasolid
    - HOOPS Exchange
      - Up to 32

- **Export step file from 3D-CAD no longer requires a CAD Exchange license D3500, D548**

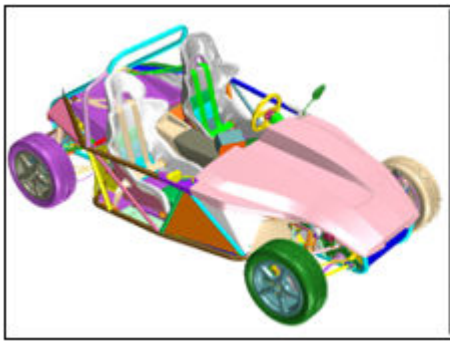
- **Split Periodic Faces**
  - New Import option in 3D-CAD to split periodic faces
    - When importing to parts, periodic faces are automatically split to make meshing easier
    - Now periodic faces can be split also when importing to 3D-CAD
    - Off by default



Geometry

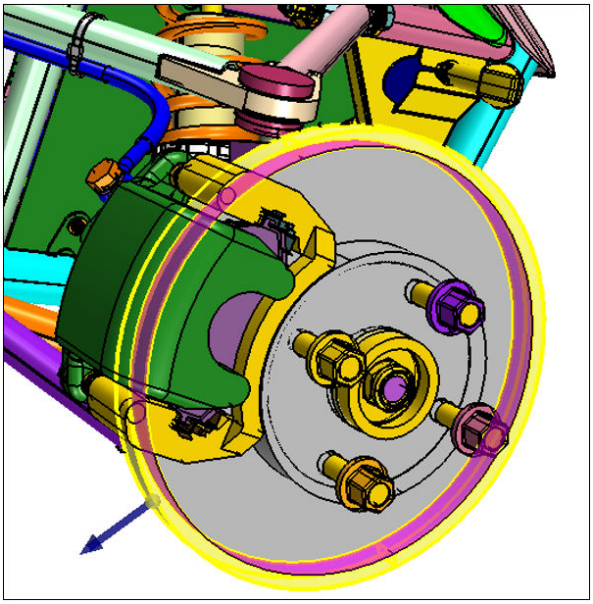
3D-CAD

- **Visualization performance improvement**
  - Significant improvement in interactive visualization performance
    - Prepare models with 1000 to 5000 bodies ~2-3x faster
      - Up to 40 to 60 fps while interacting with large CAD models

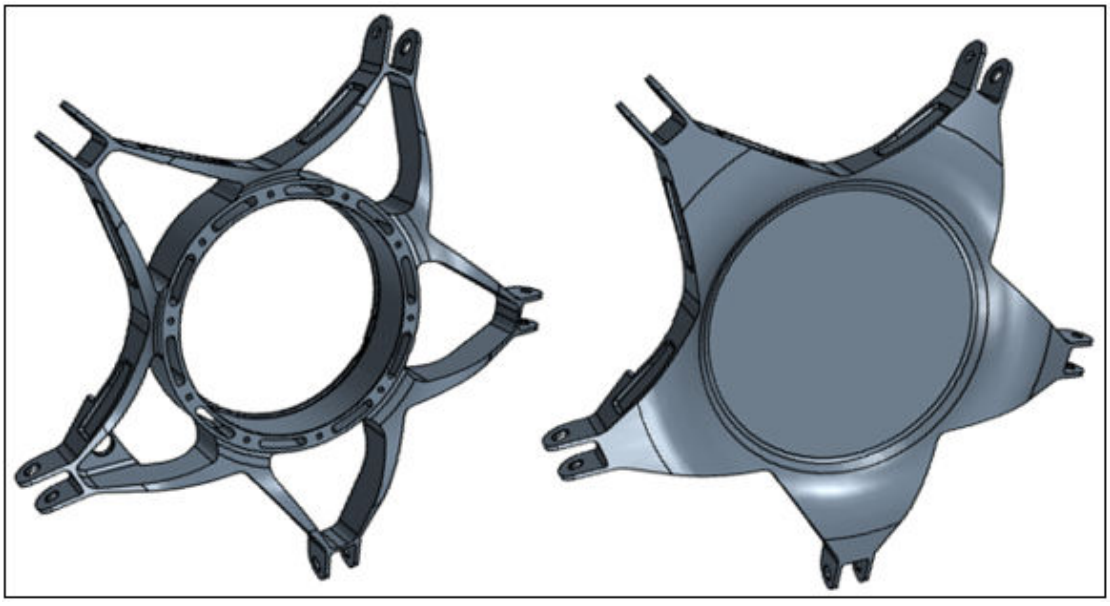


3196 Bodies	2020.2	2020.3
Rotate all bodies	2 fps	62 fps
Zoom in/out	2 fps	50 fps
1 body Show Only	17 sec	4 sec
Defeature Pedal Operation	66 sec	36 sec
Unite Operation	40 sec	22 sec

- **Interactive modeling**
  - Quickly parametrize and prepare geometry through interactive pull/push in 3D-CAD scene
    - Arrow added in 3D-CAD scene for Extrude, Extend solid, and Thicken features
    - Triad added in 3D-CAD scene for Transform > Translate feature
    - Distance parameter in feature panel automatically updated based on arrow location



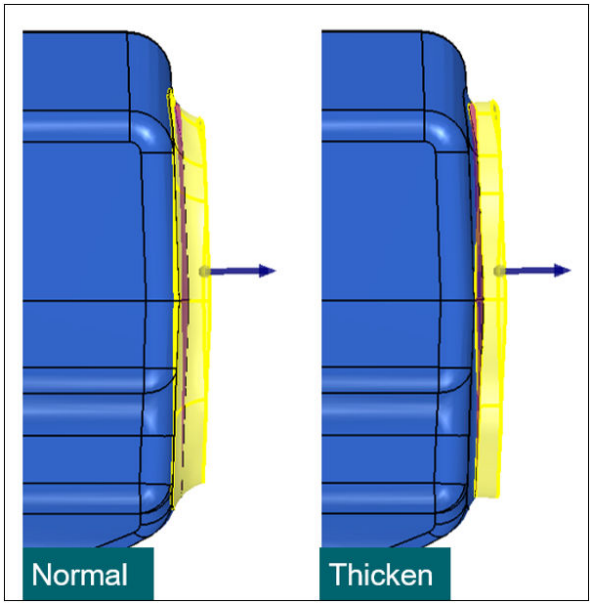
- **Force feature deletion**
  - Easily edit existing feature list by deleting unwanted features
    - Force delete a specific feature or delete a feature with all dependent features
    - Dependent feature list shown in delete panel
    - Using Force delete without deleting dependent features allows fixing failed dependent features manually
- **Defeature internal faces**
  - Through advanced defeaturing, easily remove unwanted internal faces that are not of interest for meshing
    - Interior holes or boss surrounded by multiple faces can now be defeatured
    - Includes new option **Faces to Retain** for preventing unwanted removal of faces



- **Search Tool**
  - Search for entities within a bounding box through a predicate for X, Y, and Z bounds

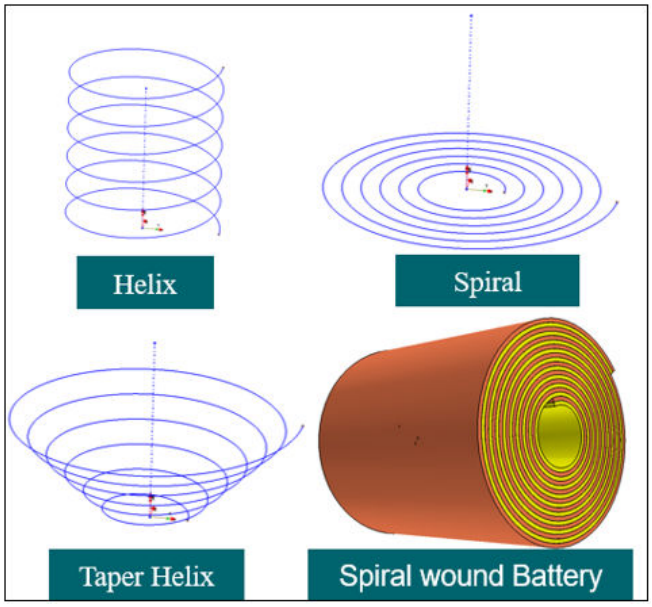
        - Supports BodyGroup, Body, Face, Edge, and vertex
    - Easily find pipes, wires, or holes in a model with “Radius” predicate
        - Supports both Face and Edge

- **Thicken in Extend solid D3732**
    - Parameterize more easily through offsetting selected faces without considering neighboring face topology
        - New option “Thicken” in Extend Solid feature



    - Easily close gaps through extending multiple non-contiguous faces up to a face
        - Example: Close gap between electronic components and circuit board for electronics cooling simulation
- **Combine Curves to 3D Curve**
    - Easily create a 3D curve in an arbitrary space by combining two planar sketches
        - Combine curve is a result of intersect between the extrusion of two curves
        - Can be used as a guide curve for a sweep or loft feature
        - Limitation: only individual curves can be combined
- **Plane from Curve D971**
    - Create a plane without any need to create points or axis as an input
        - Plane can be either Normal or Fitting to any edge of a Body or Sketch
        - Limitation: Supports just a single curve as input
- **Sketch selection in Scene**
    - Easily loft or sweep based on sketches through sketch selection in 3D-CAD scene
        - New Sketch entity option in the selection filter menu
        - Off by default
        - Right-click menu with applicable operations
        - Limitations: Rubber band and Zone Select not supported
- **Helix/Spiral Option in 3D-Sketch D3254**
    - Easily create a Helix or Spiral curve that can be used as a guide for a sweep or loft feature to generate a heat coil, suspension spring, helix blade in machinery, or spiral wound battery
        - Axis for the curve can be a straight line or curve in 3D-Sketch

- Define Helix by number of Turns or by specifying start and end angles

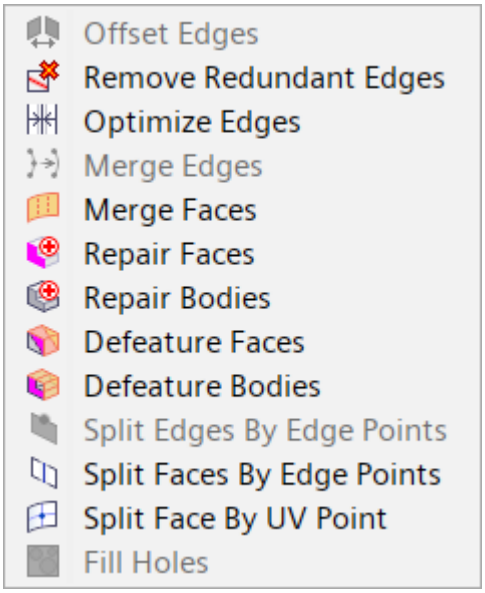


- **Duplicate Body group**
  - Enhanced the duplicate Body group feature to support:
    - Duplicate Hierarchy (ON by default)
    - Duplicate Bodies in Place
    - Place Duplicates in New Body Group
- **Rearranged right-click menu for Repair Tools**
  - Provided separate options for Body and Face specific features
  - Arranged the features in the order of edge, surface, and body

Progress bar update

- Clear status while executing time-consuming features through a progress bar oscillating from left to right
- Feature index and number of features involved in the operation displayed
- Progress message indicates individual activity being performed
- Interaction with model during feature execution now possible

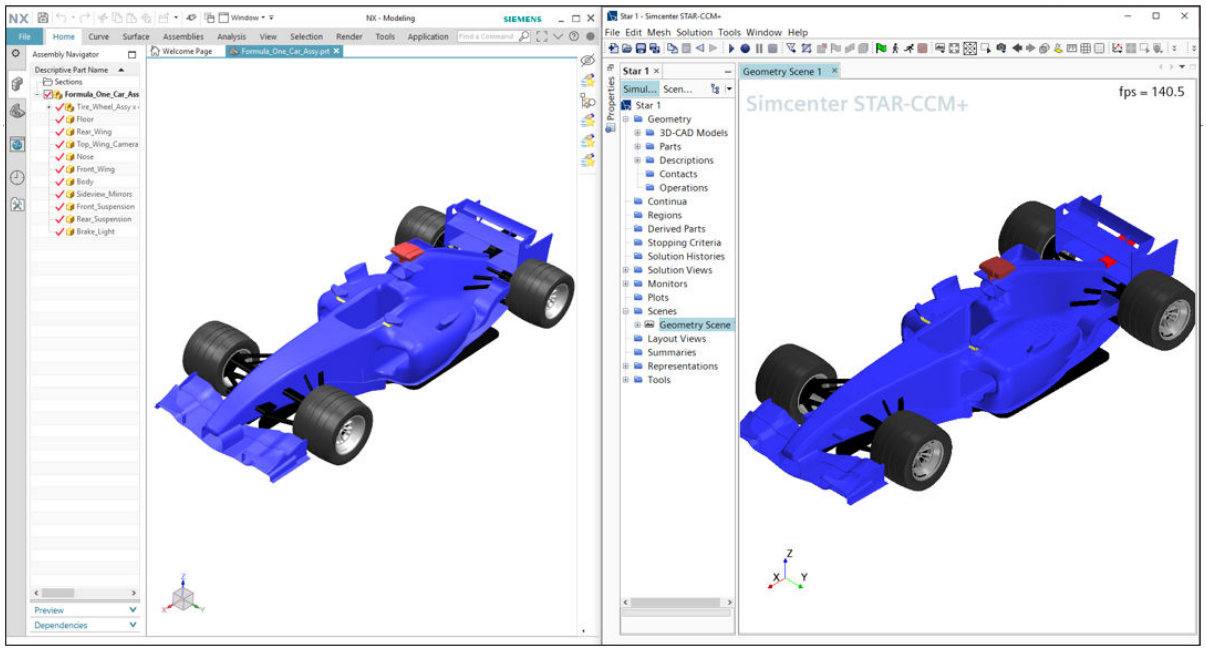
- **Sew operation**
  - Default tolerance value changed to 1.0e-6
- **Freeform**
  - Stiff edges constraint for internal edges for better shape change



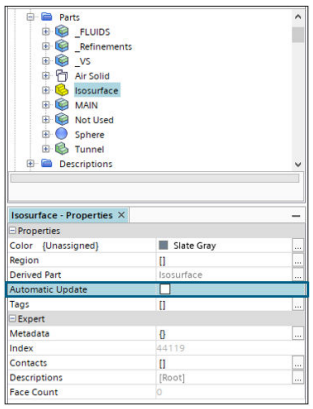
Parts

- **Imported CAD colors**
  - Better communication between design and simulation engineers through importing colors from CAD to Parts
    - Visual cues, like color, enables faster organization of models

- Import CAD colors or assign Colors for Composites, Parts, and Part Surfaces
- Applicable to neutral and native CAD format
- To display CAD Color, switch to “Geometry Part” option in "Color Mode" for a scene



- **Parts extracted from Derived Parts**
  - Automatically update Parts from Derived Parts when mesh pipeline changes
  - Update triggered only when the extracted part is used in a mesh operation
  - Off by default



Mesh

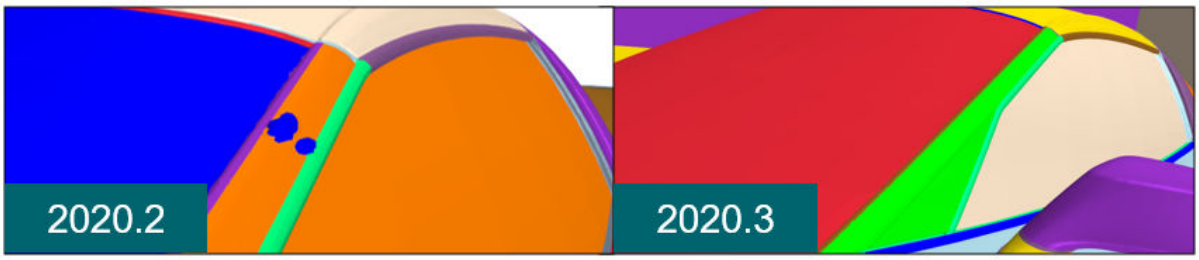
Surface Repair

- **Measure angle D3686**
  - Measure angle between two selected edges through new Query Tool
  - Both connected and unconnected edges supported
- **Calculate and review part volume**
  - New Global Tool **Calculate and review volume topology**

    - Provides part volume, face area, and edge length data for surface used in repair
    - Data presented in interactive table
- **Surface repair warning message**
  - Warning message provided while launching Surface Repair to hint possibility of meshing pipeline overwriting changes
    - Warning triggered for two specific situations
      - Repairing a mesh part description created by a mesh operation which is currently out of date
      - Repairing a part that is in contact with another part

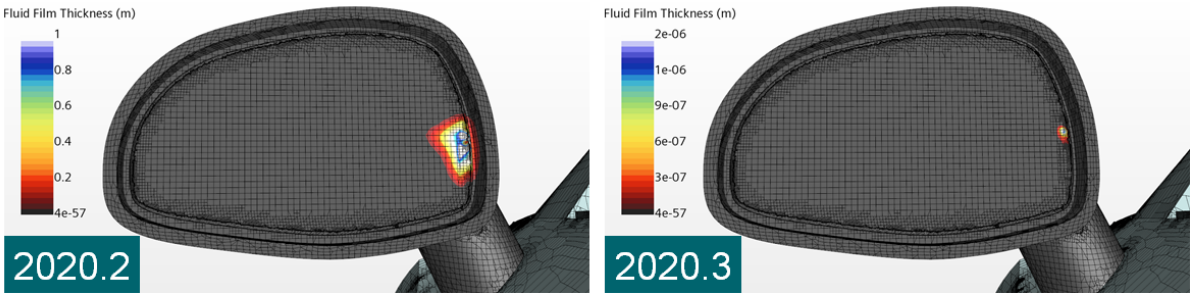
Surface Mesh

- **Surface Remesher Wake Refinement performance improvement**
  - Faster surface remeshing when using wake refinement
  - 1.3x faster for a vehicle external aero case
- **Enhanced Quality Triangle Surface Remeshing performance improvement**
  - Faster Enhanced Quality Triangle Surface Remeshing
  - 1.06-1.4x faster for cases tested
- **Surface Wrapper quality**
  - Avoid unwanted dents created by gap closure option in surface wrapper



Volume Mesh

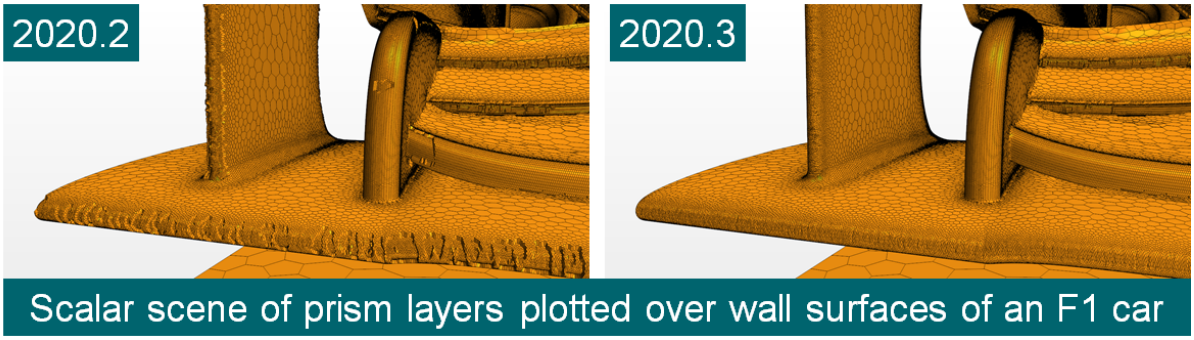
- **More robust fluid film simulations on trimmed meshes**
  - Previously small faces at the boundary or folded/flipped cells could cause divergence and/or unphysical fluid film thickness
  - Now the mesh optimizer searches for small faces to improve the face area of the prism cells
  - Note: “Optimize Boundary Vertices” under “Post Mesh Optimization” must be enabled



- **Robustness and performance improvements for the Advancing Layer Mesher (ALM)**
  - Improvements of ALM for complex geometries
    - Reduced prism layer retraction

  - Note: Reduced prism layer retraction will lead to increased total cell count
- Volume meshing up to 2.5x faster depending on application
- Less meshing failures for complex geometries

- Note: Not available for region based meshing

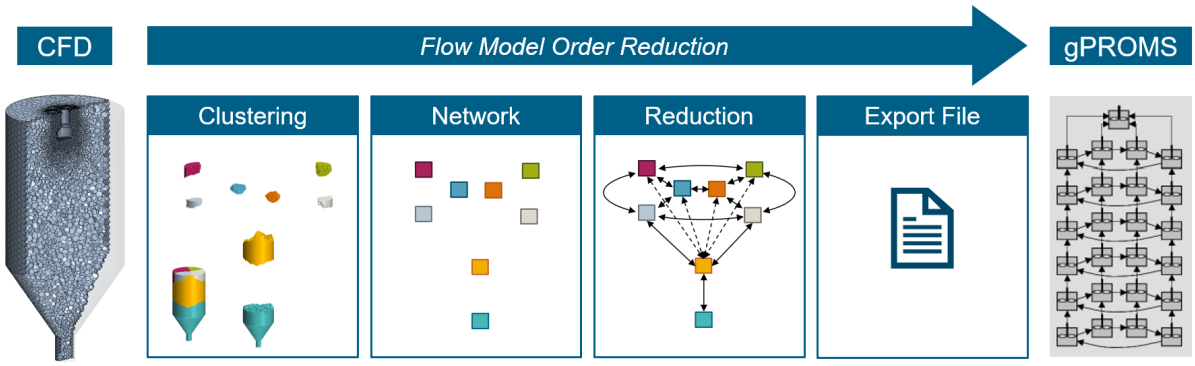


- **Improved cell distribution for trimmer anisotropic volumetric refinement**
  - Previously highly anisotropic trimmed meshes could not be created with faces that had area ratios smaller than 1:100 due to automatic merging of neighboring faces
  - Now automatic merging is not performed, and user-defined custom anisotropic sizes are preserved
- **Surface mesh deformation via a field function**
  - Morph surface mesh via an automated mesh operation: "Surface Preparation > Morph Surface Mesh"
    - Avoids tangling of mesh due to volume mesh morphing for cases with large displacements
    - Up to 1.15x faster solution time observed for a shape optimization study of an s-bend geometry due to fewer inner iterations
    - Limitation: Deformation at junction between a symmetry plane and deformed wall surface is not currently supported
- **Important note: MPI architecture change**
  - Change to Open MPI instead of Platform MPI may increase memory consumption for meshing by 15% on ~120 cores and below
- **Important note: Planned end of Region-Based Meshing**
  - Region-based meshing has been deprecated starting in Simcenter STAR-CCM+ 2020.1 and is planned to be removed in version 2021.2
  - The recommended practice is to use parts-based meshing. See the section **Simcenter STAR-CCM+ > Pre-Processing > Meshing > Parts-Based Meshing** in the Simcenter STAR-CCM+ User Guide

CAE Integration

- **gPROMS Multizonal Coupling D4907**
  - gPROMS by "Process Systems Enterprise Limited - A Siemens Business" is a powerful flow sheeting and process modeling application that allows you to model processes such as spray drying, crystallization, mixing, fluidized beds and more
  - Leverage high fidelity Simcenter STAR-CCM+ flow simulation results to calibrate a Multizonal gPROMS model using a new workflow:
    - Clustering: Sub-divides the CFD domain into clusters
    - Network: Creates a matching network representation

    - Reduction: Computes fluxes between the clusters
    - Export File: Exports the flow data on the network model
  - This integration enables more accurate modeling of complex physical and chemical phenomena, delivering more value from both CFD and process simulations
  - For a tutorial, see: Tutorials > Coupling with CAE Codes > gPROMS File Export: Spray Dryer



- **CGNS Import D4628,D1157**
  - Import CAE simulation results from CGNS files
  - The CGNS files can be used for:
    - Data exchange between sim files, other Simcenter tools, or even 3rd party tools
    - File-based coupling, especially for transient data
    - To collaborate with others, for example to pass on simulation results
  - To access the CGNS import, create a new Physics Continuum and select the following models in the following sequence:
    - External Continuum
    - File Import/Export
    - Surface Three Dimensional or Three Dimensional
    - Explicit Unsteady or Steady
- **Abaqus Co-Simulation: Support for Abaqus 2018**
  - The Simcenter STAR-CCM+ to Abaqus Co-Simulation now supports Abaqus version 2018

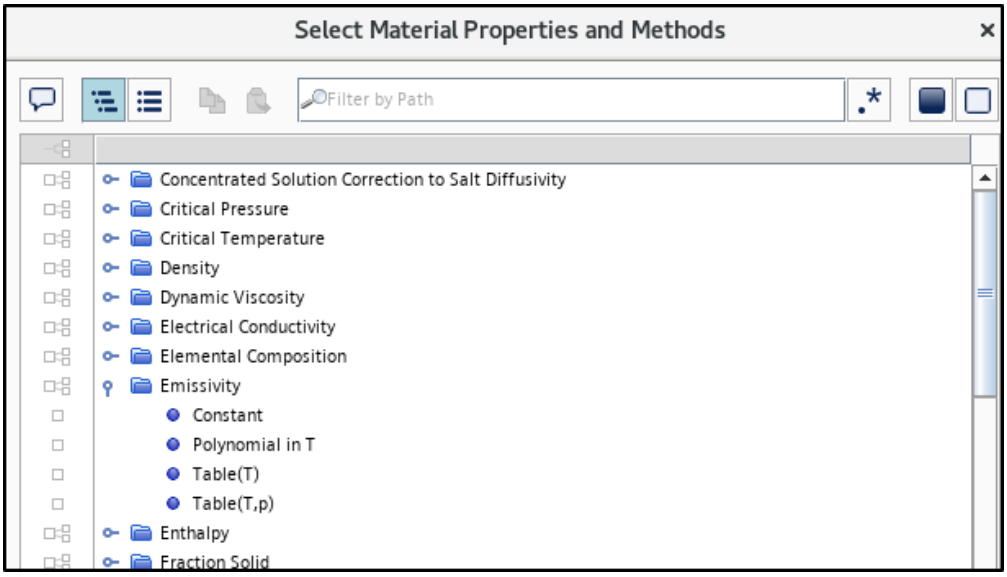
Physics

CFD
Multiphase Flow
Computational Rheology
Computational Solid Mechanics
Electromagnetics and Electrochemistry
Aeroacoustics
Harmonic Balance
Motion, Mesh Adaption, and Mapping

CFD

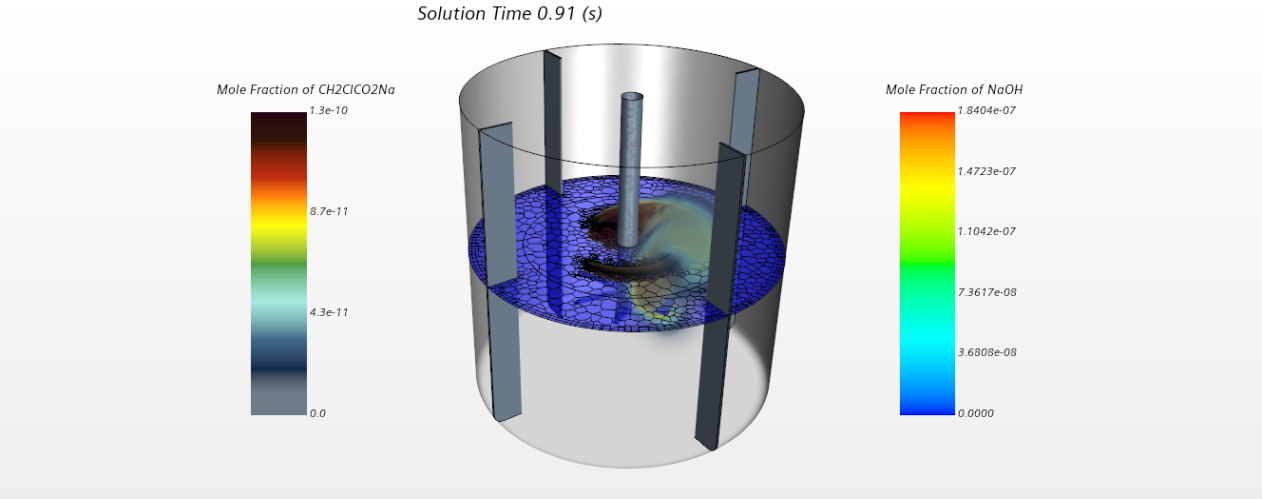
Flow

- **Simulation Operations simplification with Solve Physics**
  - Improves usability of Simulation Operations with a consolidation of the Solve Time Scale & Solve Continua operations into Solve Physics
    - Any Continuum or Time Scale can be selected with an object selector
  - Streamlines workflow with a single operation
    - A single Solve Physics operation supports multiple Continua but only a single Time Scale
    - Continua and Time Scales cannot be mixed in a single operation
- **Custom Material Databases: New workflow and options D5441**
  - Streamlined material library workflow with a new look and feel
    - Object selector instead of long list
    - Deletion now possible
  - Allows easier integration with external libraries or importing from Teamcenter thanks to extra properties and input methods
    - Constant, Polynomial and Table are now supported for all materials
    - Sutherland Law added for Dynamic Viscosity and Thermal Conductivity



- **Highlight for Material Part Groups**
  - Improves set-up workflow in simulations with large number of parts and multiple materials thanks to a visual cue
    - Provides insight on the setup of Material Part Groups
    - Parts assigned to the selected Material Part Group are now highlighted upon selection
  - Reduces set-up errors, and saves time with less scrolling when searching for part assignment in the simulation
- **Improved convergence of the Explicit Mapped Contact Interface**
  - Improved convergence of multi-timescale conjugate heat transfer cases that use activate/deactivate workflow

    - By updating the fluid side explicit mapped interface boundary heat flux and coefficients before mapping to the solid side
- **Adaptive Mesh Refinement (AMR) compatibility with user-defined Equation of State (EoS)**
  - Improves simulation turnaround times for applications with user-defined EoS
    - Example: marine propeller blade Fluid-Structure Interaction (FSI)
      - User-defined EoS is used to provide fluid compressibility and to improve solution stability as a result
      - AMR used for the blade wake; this previously required an additional preliminary run and usage of refinement table
- **Mixing Plane stability improvements**
  - Improves simulation stability when changing mixing plane treatment
    - Solution on the interface is no longer cleared when the interface is changed between implicit and explicit
    - Eliminates stability issues caused by re-initialization of interface
- **Segregated Solver robustness improvements**
  - Update of the Flux Correction Coefficients at Pressure Outlet boundaries
    - Improves robustness by better resolving the flow behavior on pressure outlet boundaries
  - Improved robustness in cases with constant temperature walls
    - Modified the gradient limiting in the higher order convection terms of the energy solver
    - Beneficial where there are events such as valve opening and droplets evaporating in the vicinity such as for in-cylinder calculations
- **Adaptive Mesh Refinement (AMR) compatibility with Multi-Component Liquids**



  - Improved speed and accuracy of multi-component liquid reactions
    - Important for cases with 2 competing reactions and micro-mixing effects
    - Eliminate unphysical pulsation effects where the reactant is injected—caused by reactant being consumed by faster reaction in larger cells

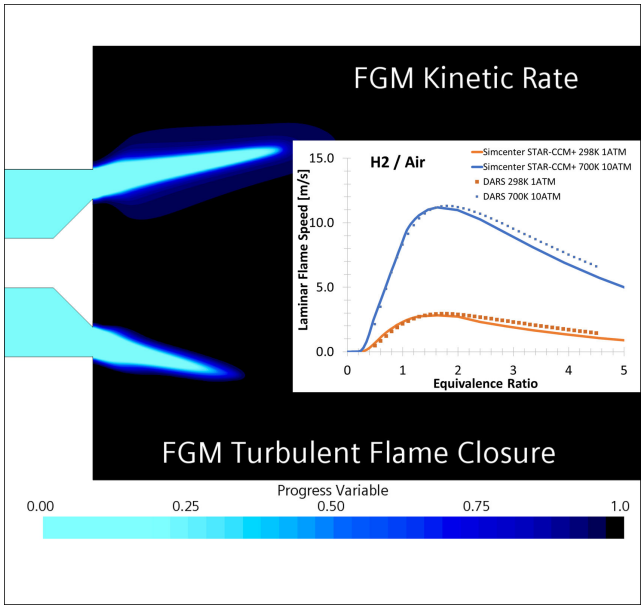
Energy

- **Collimated/Laser Sources D4320**
  - Improved physical realism of cases with collimated source of light
    - Directional Source option for radiation physics condition on boundaries

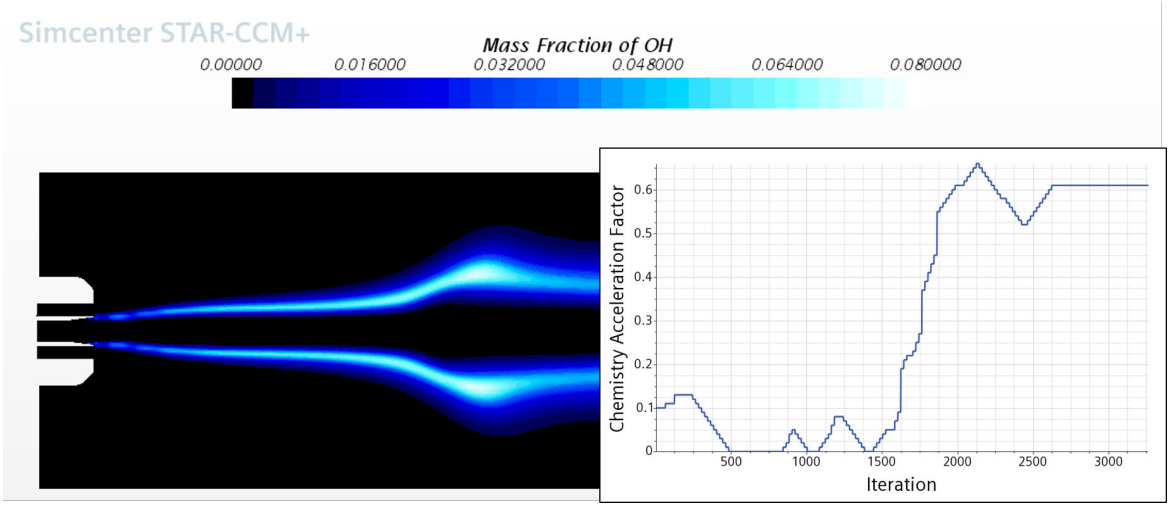
        - Applies to Surface-to-Surface (S2S) and Surface Photon Monte Carlo (SPMC) radiation models
      - Provides ability to apply converging, diverging, collimated (parallel) beam source at boundaries
    - Easy workflow that requires no detailed modeling of source mechanism
      - Input of both heat flux profile and direction profile
      - Specification for entire spectrum or per spectral band
      - Gaussian beam (e.g. lasers) input method
- **Surface Photon Monte Carlo (SPMC) compatibility with Mixing Planes**
    - Simpler simulation workflow
    - Now possible to use mixing plane to simulate, for example, fans and rotating parts in headlamps, where the use of SPMC is essential for the simulation of refraction
- **Participating media radiation compatibility with Adaptive Mesh Refinement (AMR)**
    - Improved simulation speed and accuracy of applications such as large gas turbines with combustion or glass formation, where radiation is necessary for thermal prediction
    - Applicable for both Discrete Ordinate Method (DOM) and Spherical Harmonics lowest order P1.
- **Expression-based material properties for particle radiation D4752**
    - Improved physical realism by allowing expression-based particle absorption and scattering efficiency for the Lagrangian phase Particle Radiation model
      - Useful in applications where absorption and scattering coefficients change meaningfully due to the composition changing
      - For example: coal combustion where combustible coal, char, and ash have very different surface properties, and therefore the radiation properties vary
- **Advanced Ray Tracing as default**
    - Improves speed of ray tracing
      - The Advanced Ray Tracing option was made the default option
      - No support for simulations with shells where Advanced Ray Tracing is not compatible

Reacting Flows

- **Laminar Flame Speed (LFS) tabulation for Flamelet Generated Manifold (FGM)**
    - Provides support for next generation fuels which have substantial flame propagation effects
      - New reactor type for FGM '1D Premixed Freely Propagating'
      - Automatic Laminar Flame Speed Calculation with this reactor type
      - LFS is tabulated as a function of Heat Loss Ratio and Mixture Fraction
      - Available for FGM closure models Turbulent Flame Closure (TFC) and Coherent Flame Model (CFM)
    - Reduced computational expense with a novel calculation method, robust particularly near flammability conditions
    - Improved workflow, no longer requiring the use of external tools such as DARS
    - Limitation: currently laminar flame speeds are generated at a single pressure



- **Fast flame propagation with Complex Chemistry (CC) – CC-TFC-RTCE**
  - Turnaround time significantly reduced for pure flame propagation simulations by combining Complex Chemistry, Turbulent Flame Closure, and the Relax To Chemical Equilibrium model (CC-TFC-RTCE).
    - Typical 3-4x speed up over a full mechanism
    - Transports only the major chemical species
  - Better user experience by automating the propagation in the flame region
    - Only required user input is the Turbulent Flame Speed (TFS)
    - RTCE used in the flame brush
- **Auto Acceleration Factor for chemistry**
  - Significant robustness improvements through automatically and optimally setting the acceleration factor
    - No need for trial and error to find optimal setup
    - Driven by intelligent algorithm
  - Computational expense is reduced by ensuring that the acceleration factor starts low and ends as high as possible
    - Acceleration Factor determines the stability of the chemical source terms
  - Easy monitoring of the case stability through an Acceleration Factor report, monitor and plot



- **Liquids with surface chemistry D4956**
  - Enables many chemical process industry applications through liquid surface chemistry
    - Liquid catalytic beds
    - Common liquid surface reactions supported
  - Fast calculations through use of in-built chemistry solver, which can also be combined with auto acceleration factor
  - Improved user experience through built-in capability
    - No longer necessary to change gas properties to liquid
  - Compatible with surface chemistry in porous media and on flat surfaces
- **Soot Moment Model improvements**
  - Improved convergence, stability and accuracy for the soot moment model
  - Addition of the Hydrogen Abstraction Carbon Addition-Ring Closure (HACA-RC) surface oxidation chemistry model for diesel engines
  - OH and O_2 fields can now be viewed as fields when an Extended Coherent Flame Model (ECFM) soot table is used
  - Important notes:
    - Changes in soot volume fraction from previous versions are expected
    - Soot moment residuals will experience a jump
    - Soot moment residual normalization has been turned off by default
    - Cold Flow Temperature property has been removed from the user interface
- **Flamelet Table Generation Improvements**
  - Up to 8x speed up for Flamelet Generated Manifold (FGM) and Chemical Equilibrium table generation for large mechanisms
    - For mechanisms greater than 200 species a new chemical equilibrium solver is used
  - Up to 3x speed up for 1D FGM and Steady Laminar Flamelet (SLF) table generation
    - A novel algorithm is now employed to decrease table generation time for 1D FGM and SLF tables
    - Speed up increases as the size of the mechanism increases
  - Improved user experience through retaining flamelet table generation settings when a new chemical kinetic mechanism is imported
    - Common settings such as fuel and oxidation composition, and absolute pressure, that are not normally simulation dependent, are now kept when a new mechanism is performed

  - Rename of 'Read FGM Library and Construct Table' option:
    - The word 'DARS' has been removed from this right click option to reflect the fact that any FGM table in the right format can be imported
    - This change has also been made for SLF and Chemical Equilibrium tables
- **Hot surface ignitors**
  - Hot surface ignitors can now be modeled using the 'Subgrid Spark Ignitor'
    - It is now possible to disable the convection of the flame kernel
    - This mimics a hot surface ignition process as opposed to a spark plug where the flame kernel is convected away with the flow.
    - This change has also been made for Steady Laminar Flamelet and Chemical Equilibrium models
- **Minor usability improvements**
  - Spark Ignitors are now highlighted in scenes when selected in the tree
  - Removal of the CFL Scaling Factor parameter

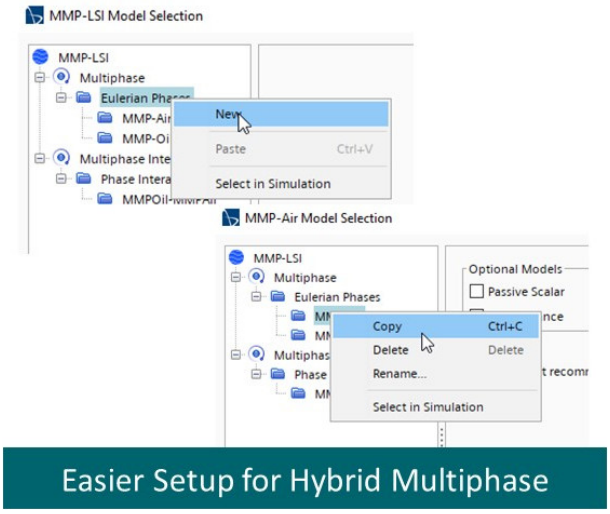
Turbulence

- **Scale-Resolving Hybrid (SRH) turbulence model D2314**
  - Increased fidelity Unsteady RANS simulations through enhanced temporal modeling approach
  - Enables potential accuracy improvements to existing URANS simulations with minimal additional computational cost
- **Sub-Grid Scale (SGS) Turbulent Kinetic Energy Field Function D4795**
  - The SGS Turbulent Kinetic Energy field function indicates levels of modeled turbulence, and can be used to assess mesh resolution in Large Eddy Simulation (LES) calculations
  - Previously this field function was only available when combustion or aeroacoustics models were activated
  - It is now available as default with any LES model, as long as *Temporary Storage Retained* is activated

Multiphase Flow

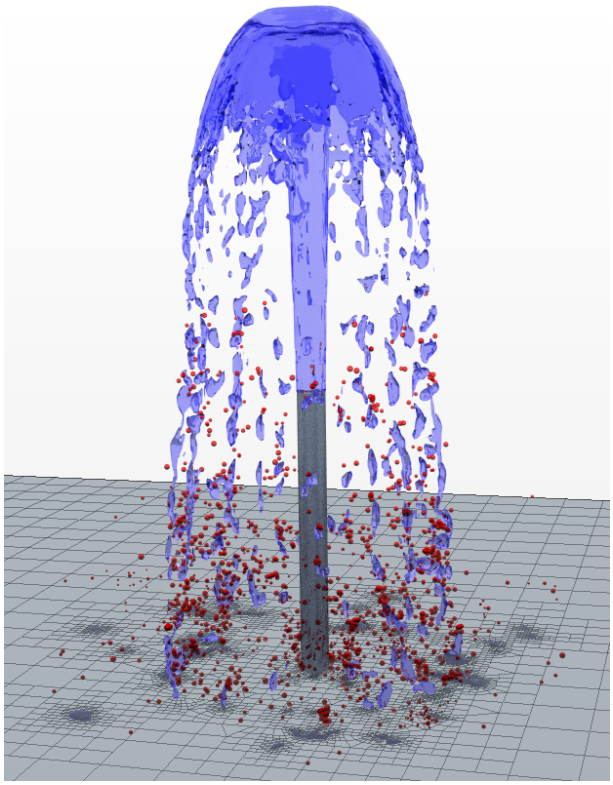
General User Experience Improvements for Multiphase

- **Actionable model selection panel**
  - Reduce simulation set-up time and user error
    - Possible to create new phases and phase interactions in the panel
    - Users can copy/paste, delete or rename phases/phase interactions
  - More easily relate model selection panel to corresponding entities in simulation tree
    - **Select in Simulation** jumps to the main tree with the relevant node selected

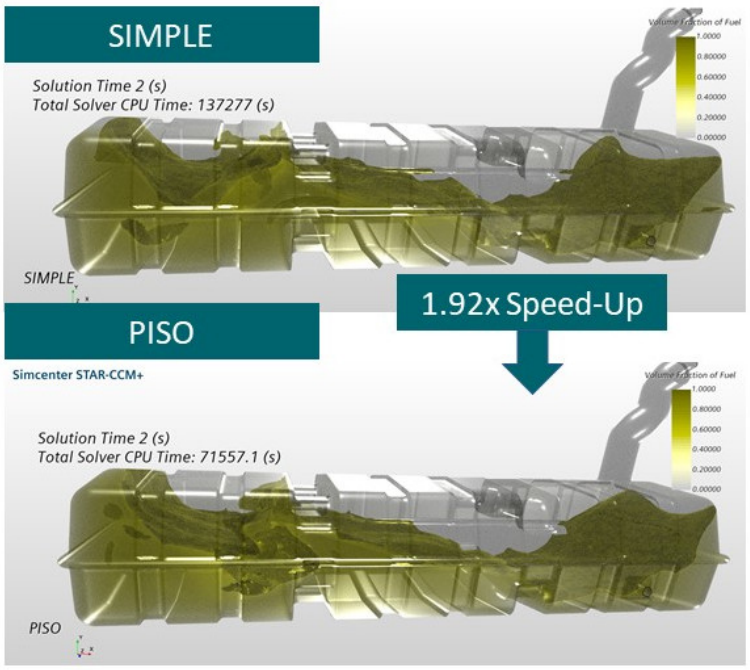


Volume Of Fluid (VOF)

- **Volume Of Fluid (VOF) to Lagrangian (LMP) resolved transition D1541**
  - Reduced computational expense for applications such as fuel spray break-up or vehicle water management compared to fully resolved VOF
    - Hybrid approach:
      - Formation of droplets is resolved by VOF and a fine mesh
      - Subsequent tracking of droplets is done with LMP in a coarser mesh
  - Maintain accuracy at lower computational cost
    - Use VOF to resolve important physics locally without the need to track every droplet and bubble as VOF
  - Works alongside Adaptive Mesh Refinement (AMR) and cell clustering to ensure best use of resources and models
    - Formation of droplets is resolved by VOF and a fine mesh
      - Typically AMR is used to track VOF blobs locally without expense of a globally dense mesh (not mandatory)
    - Transitioning is based on user criteria for blob size
      - Can be a field function to allow zonal or other criteria to be used
      - Built on the VOF blob detection model
    - Subsequent tracking of droplets is done with LMP
      - Cell clustering is used to allow the LMP droplets to exist on the same mesh as resolved VOF
      - If AMR is used this will then coarsen the mesh over subsequent timesteps
  - The above workflow describes droplets, but modeling bubbles is also possible, subject to the normal limitations of LMP in terms of available equations of state (no independent bubble pressure field)



- **VOF compatibility with PISO**
  - Reduced time to solution for free surface simulations
    - For suitable cases PISO can be 1.5-3x faster than SIMPLE
    - PISO is suited to cases with a small timestep which is typically the case for VOF as the CFL number is less than one
  - Can be used with single or multi-step versions of VOF
  - Volume fraction update options for PISO allow for reduced updates
    - First and Last Iteration (default)
    - Last Iteration
    - Every Iteration (same as SIMPLE)
  - May not be suitable for quasi-steady type simulations
    - Here time accuracy is not required and SIMPLE may be more efficient (via a larger stable time-step size)

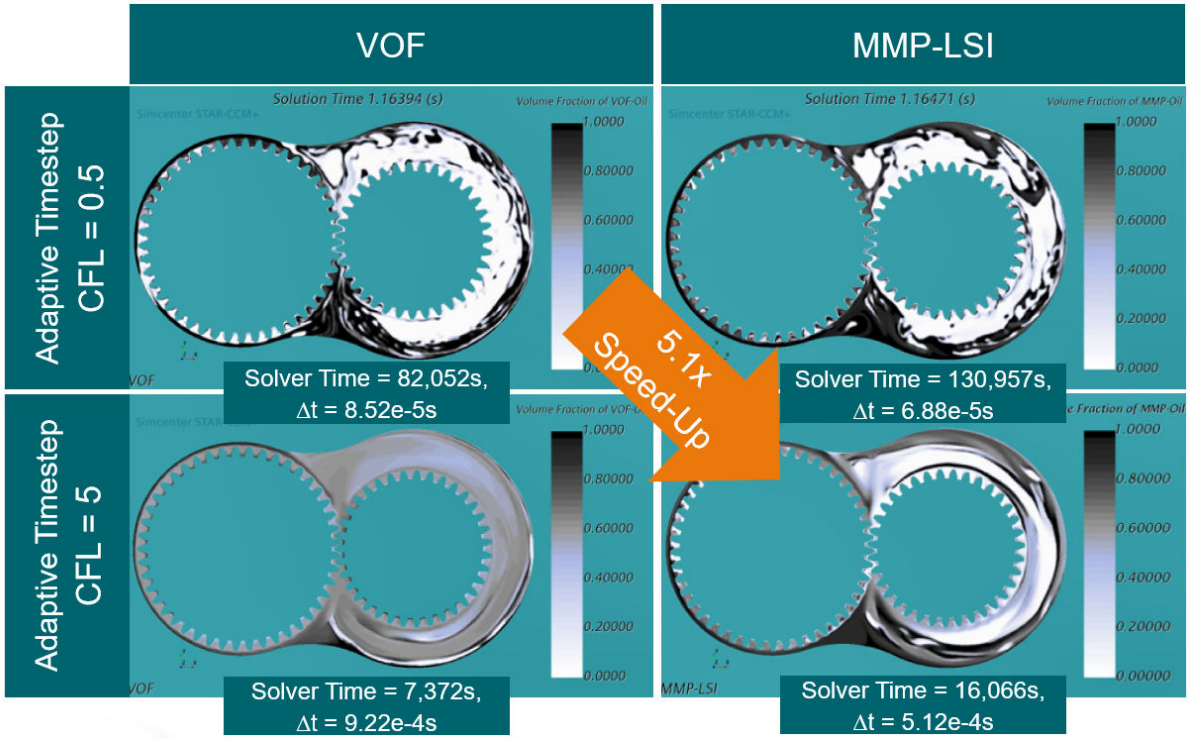


- **Wave Damping/Forcing by boundary D4305**
  - Reduced computational expense for marine simulations
    - Use wave damping at outlets to prevent reflections in simulations where wave forcing also used
      - Users can specify wave damping or wave forcing on a per boundary basis
        * Previously wave damping could not be used if forcing was used
        * Forcing is used to enforce desired wave shapes in the vicinity of inlets
      - Reduces computational domain required thus making applications practical that are not otherwise so

Mixture Multiphase (MMP)

- **Mixture Multiphase - Large Scale Interface Model (MMP-LSI)**
  - Reduced computational expense compared to fully resolved Volume of Fluid (VOF)
    - Combines the benefits of VOF and MMP
      - Models both mixtures and free surface flows in the same simulation
      - Combines characteristics from both VOF and MMP approaches
        * VOF captures free surfaces without slip between phases in a mixtures
        * MMP models slip between phases in a mixtures without free surfaces
        * MMP-LSI can do both
      - Uses Adaptive Interface Sharpening (ADIS) convection scheme
    - Allows lower resolution than VOF by including appropriate modeling of mixtures whilst capturing free surfaces
    - Computationally cheaper than the Eulerian Multiphase Large Scale Interface (EMP-LSI) equivalent
      - Solves only one set of transport equations for all phases (like VOF)
  - Improved physical realism and accuracy

- Compared to VOF for under-resolved mixtures
  - Under-resolved VOF results in numerical mixtures that are not physical
  - MMP-LSI models the mixture correctly with appropriate drag laws



- **Film Stripping into Mixture Multiphase**
  - Reduced computational expense for mixtures with Fluid Film
    - Allows Mixture Multiphase (MMP) to be used with Fluid Film in place of Eulerian Multiphase (EMP)
    - Builds on existing MMP impingement/deposition into Fluid Film phases
  - Model complex multiphase applications more efficiently
    - Allows modeling of applications with thin films on surfaces including annular flows
    - MMP well suited to modeling fine mists in gearbox lubrication and e-machine cooling

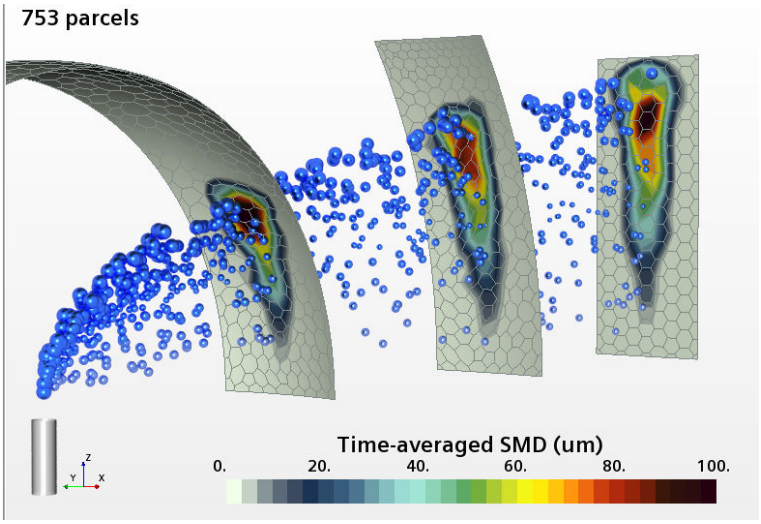
Eulerian Multiphase (EMP)

- **S-Gamma improvements**
  - User control over accuracy/cost trade-off
    - Discrete Quadrature S-Gamma now allows user-defined discretization of population
    - Allows user to increase the number of quadrature points to increase accuracy at increased cost
    - Alternatively use reduced number of quadrature points for rapid solution
  - Improved stability
    - S-Gamma now solves for specific moments rather than moments directly as in past versions
    - Specific moments are given by dividing moments by the dispersed phase volume fraction
- **Interface Momentum Dissipation for Eulerian Multiphase-Large Scale Interface (EMP-LSI) Surface Tension**
  - Improved accuracy and robustness for cases with surface tension
    - The Interface Momentum Dissipation model previously available for Volume of Fluid (VOF) is made available for EMP-LSI

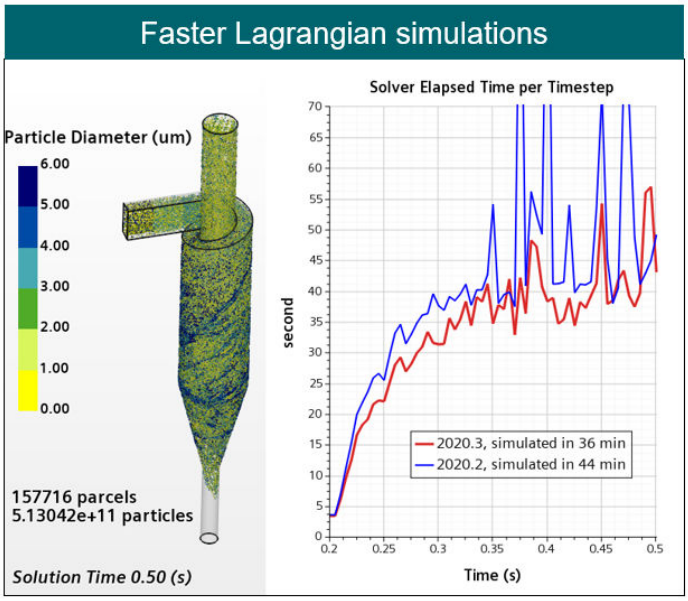
- This model reduces the spurious currents that can be created around free surfaces due to surface tension

Lagrangian Multiphase (LMP)

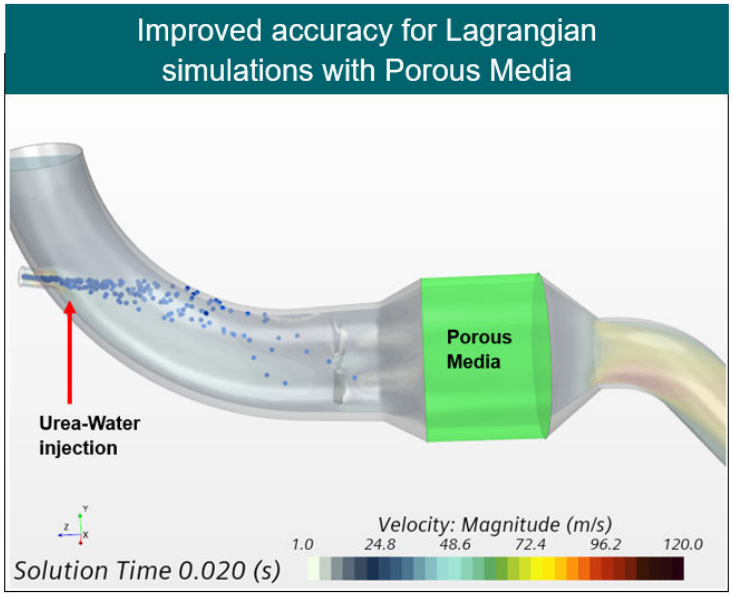
- **Boundary Samples Data Mapper D4380**
  - Improvement in Lagrangian data analysis with support for mapping of Lagrangian data on boundaries
    - Useful for time-averaging spatial distribution of Sauter Mean Diameter of evaporating or reacting droplets at fixed distance from injector
  - Improved workflow for time averaging of particles interacting with boundaries
    - Maps Particle Tracks generated by Boundary Sampling model on faces of boundary
    - Result of mapping can be used in Field monitors to time-integrate or time-average Lagrangian data
    - Mapping operators: Average, Volume-Weighted Average, Mass-Weighted Average, Moment(3,2), Variance, Sum, Minimum, Maximum



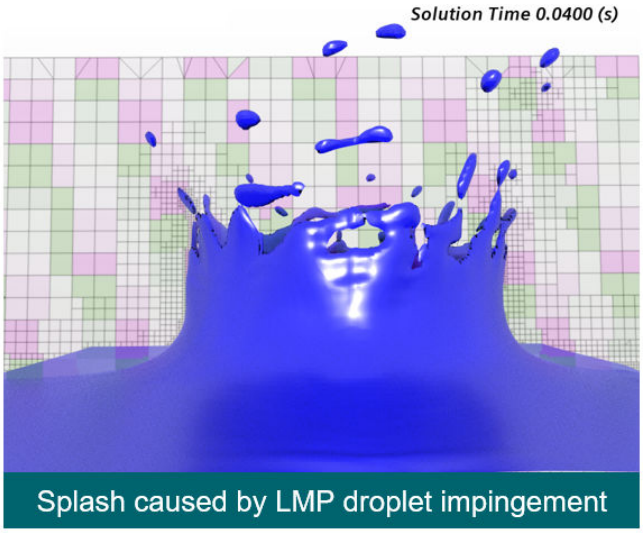
- **Speedup of Schiller Neumann Drag model**
  - Faster Lagrangian simulations through optimization of the Schiller Neumann drag force model without loss of accuracy
    - Up to 1.2x speedup
  - No new settings: this change is applied automatically to all simulations with Schiller Neumann drag model
    - Speedup factor is highest for applications with large number of parcels, fine mesh, and absence of complex particle-wall interaction models



- **LMP compatibility with Porous Media model D5518**
  - Improved accuracy and ease of use when using Porous Media model instead of Porous region approach
  - Typical applications include catalytic convertors, selective catalytic reduction, plenum chambers in fluidized bed reactors



- **Source smoothing for Volume of Fluid (VOF)-Lagrangian impingement model D5316**
  - Improved accuracy of VOF-Lagrangian impingement model
    - LMP mass is now contributing into all cells of the cluster hosting impingement event
      - Eliminates or reduces momentum loss when impinging parcel size is larger than cell size
  - No changes in the user interface



- **Parametrization of input values of Rosin-Rammler distribution**
  - Improves the workflow for calibrating the particle size distribution
  - Exponent of Rosin-Rammler distribution can be set using global parameters
- **Particle Enthalpy field function**
  - Useful for evaluating energy component of Lagrangian impingement source when specific heat of liquid or gas material is specified through polynomial function method
- **Improved handling of warning messages in evaporation models**
  - Frequency of warning messages reduced when mismatch between components in LMP and continuous phase is detected

Discrete Element Method (DEM)

- **Flux based choices for DEM flow rate specification**
  - Added Mass Flux, Volume Flux, Particle Flux options to the list of Flow Rate Specification methods in injectors

Computational Rheology

- **Conjugate Heat Transfer (CHT) for Rheology: Convection and radiation boundary conditions**
  - Improved accuracy and realism for CHT applications
    - Include effects of convection and radiation at wall and freestream boundaries
    - Mirrors Finite Volume boundary conditions options
  - Convection allows reference temperature and Heat Transfer Coefficient to be set
    - Accounts for external heat transfer
  - Radiation allows ambient temperature and surface emissivity to be set
    - Accounts for radiation from environment
    - Equivalent to environment option for Finite Volume flow
    - Requires selection of new *Viscous Radiation Model*
- **Pressure Inlet for Rheology D4727**
  - For cases where flow rate data not available
    - Calculates velocity profile based on imposed pressure
    - Thixotropy, level-set function, and temperature data can be set.

- Note: Pressure inlet inherits 'Stagnation Inlet' name from Finite Volume CFD
  - In reality this is a static pressure inlet
  - Dynamic head not meaningful for viscous flows
- **Thermocapillary Marangoni Convection for Surface Tension**
  - Model Marangoni flow introduced by gradients in surface tension caused by non-uniform temperature fields
  - Available as a sub model of Surface Tension: *Thermo-capillary Marangoni*

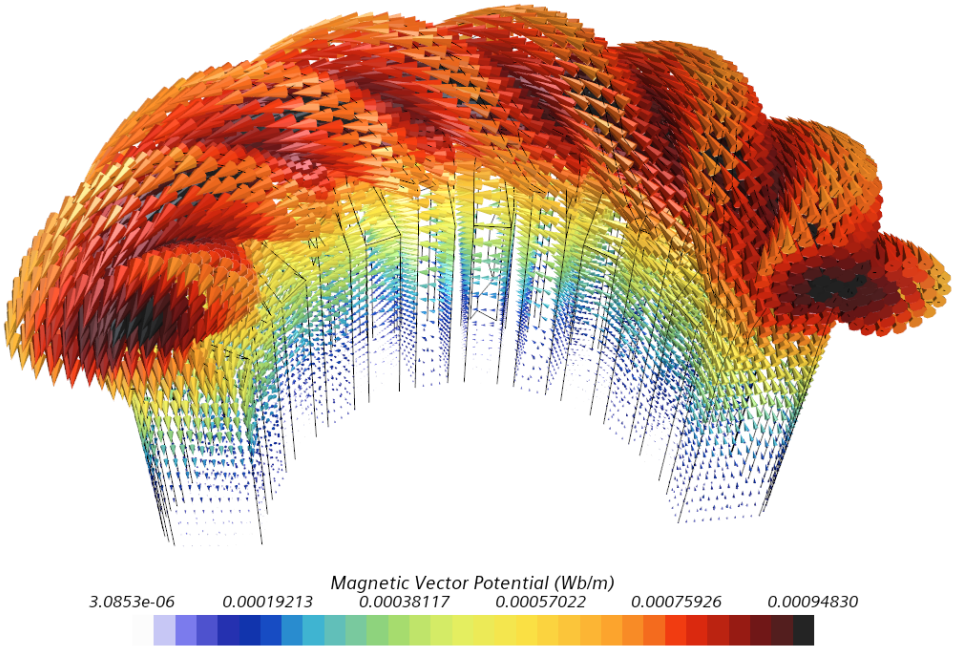
Computational Solid Mechanics

- **Flexible Dynamic Fluid-Body Interaction (DFBI): Account for structure deformation**
  - Get Fluid-Structure Interaction (FSI) results faster by maintaining linearity despite large rigid body motion
    - 6-DOF (Degrees of Freedom) model is used to solve for the rigid body motion
    - Continuum Finite Element (FE) model is used to compute the deformations
    - Separation of rigid body motion and deformation makes it possible to maintain linearity of the FE model
  - 3 options are available:
    - DFBI – Deformable body: entire body deforms, for example hull and frame
    - DFBI – Partially deformable body: only some parts of the body deform, for example only the hull
    - DFBI – Deformable moving attachment: a moving attachment deforms, for example the propeller
  - See also: Tutorials > Solid Stress > FSI and 6-DOF Motion: Stress Analysis on Boat Propeller

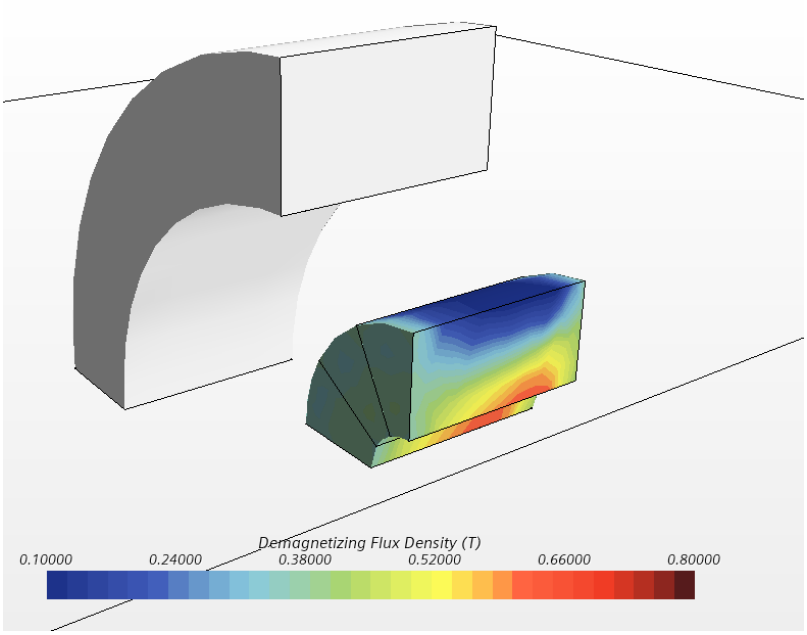
Electromagnetics and Electrochemistry

Electromagnetics

- **Finite Element (FE) Excitation Coil model - Boundary method**
  - Enables simulation of excitation coils with stranded windings in 3D FE
    - Supports complex geometries (non-uniform cross-sectional areas)
    - By ensuring divergence-free electric current density distributions, the new model improves convergence of FE iterative solver and computes smoother Magnetic Vector Potential distributions
  - Beneficial for realistic e-machine simulations in 3D
  - The coil inflow current is specified as boundary condition

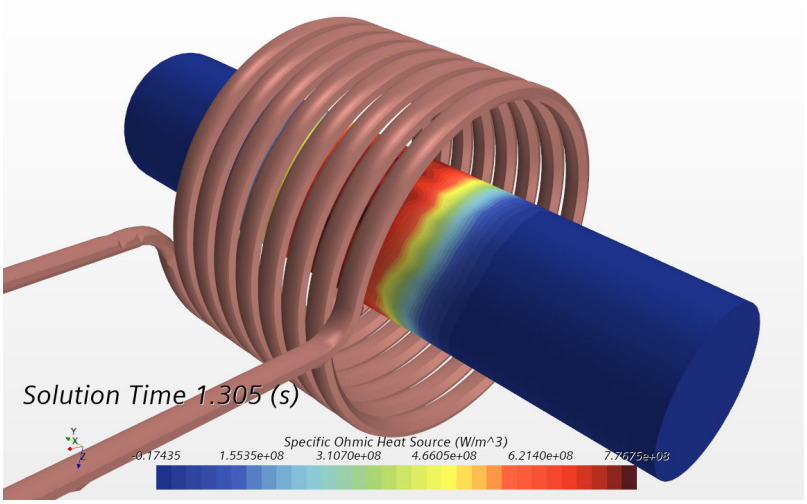


- **Demagnetization Indication model**
  - Highlights areas within permanent magnets at high risk of irreversible demagnetization
    - The Demagnetization Flux Density measures by which amount the magnetic flux density is temporarily below the critical value in each cell
      - A large value indicates high risk of irreversible demagnetization
    - Beneficial for applications where it is desirable that the magnet strength stays constant, for instance: e-machines



- **Finite Element (FE) direct solver in frequency domain**
  - Enables frequency domain (Harmonic Balance) simulations of applications requiring a FE discretization
    - Extends Harmonic Balance offering to linearized ferromagnetic materials

        - Previous versions of frequency domain electromagnetic solvers could simulate only dia- and paramagnetic materials.
    - Beneficial for applications like induction heating, magnetic stirring, wireless energy transfer, transformers, busbars
    - The new FE frequency domain solver is direct



- **Ohmic Heating for Finite Element (FE) Solid Energy**
    - Improved accuracy of temperature distributions in transient simulations with eddy-currents
        - The Ohmic Heating model, the FE Solid Energy model and the FE Magnetic Vector Potential model are now compatible
        - The Ohmic Heating model provides an energy source to the FE Solid Energy model
        - Higher fidelity thanks to higher-order internal representation of FE discretization
    - Beneficial for multi-physics simulations in time domain where ohmic heating is relevant (induction heating, wireless energy transfer)
- **Ohmic Heating in frequency domain for Finite Element (FE) Solid Energy**
    - Improved accuracy of temperature distributions in frequency domain simulations with eddy-currents
        - The Ohmic Heating model, the FE Solid Energy model and the Harmonic Balance FE Magnetic Vector Potential model are now compatible
        - The Ohmic Heating model provides an energy source to the FE Solid Energy model
        - Higher fidelity thanks to higher-order internal representation of FE discretization
    - Beneficial for multi-physics simulations in frequency domain where ohmic heating is relevant (induction heating, wireless energy transfer)
- **More accurate Magnetic Energy, Co-Energy, and Stress Tensor in Finite Element (FE)**
    - Those field functions are now evaluated by accounting for FE shape functions
        - Results in smoother and more accurate distributions
- **Motion-Induced Voltage Report for non-magnetostatic cases**
    - The Magnetic Motion Induced Voltage Report and the Motion Induced Voltage Source in the Excitation Coil Current Element are now exposed in simulations including eddy currents
- **Removed temperature range check in electromagnetic table(T)**
    - The range check and the warning message was removed
    - New behavior consistent with other models
- **Combine Excitation Coil Circuit Element with Finite Element Excitation Coil Circuit Element**

- Simplifies workflow by removing the distinction between those two circuit element types
- The two circuit elements have been merged into a unique Excitation Coil Circuit Element

Electrochemistry

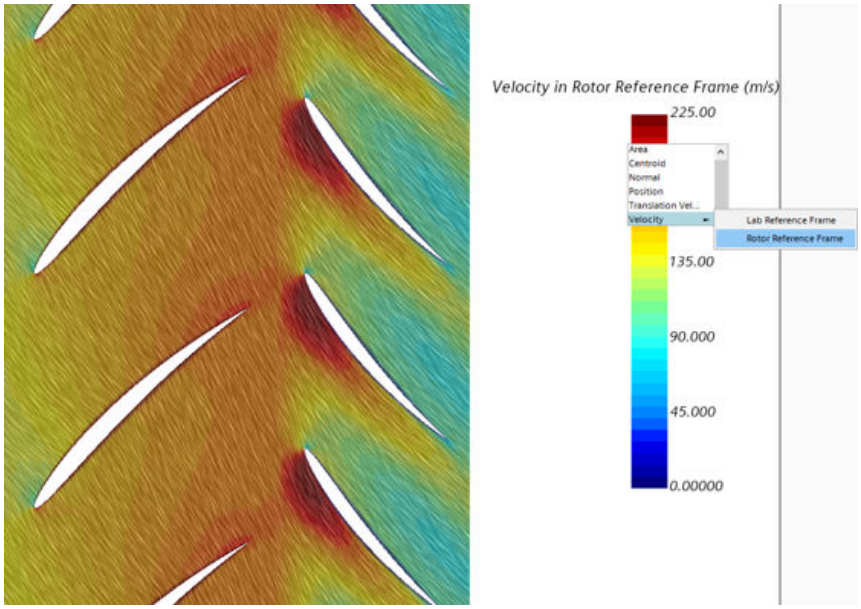
- **Charge deposition on dielectric surfaces**
  - Enables the simulation of capacitively coupled plasma (CCP)
    - A new physics model 'Charge Accumulation Model' is available
    - Interfaces between two continua can now accumulate charge
- **Reference concentration for electrochemical species**
  - Simpler workflow by removing the need to define a field function to alter the electrochemical reaction rates to account for deviations from unity reference concentration
  - Used when calculating electrochemical reaction rates in, for example, the Butler Volmer equation, and in the Nernst equilibrium potential.

Aeroacoustics

- **Two properties of Acoustic Wave Model deprecated**
  - Low-pass space filtering option has been deprecated starting in Simcenter STAR-CCM+ 2020.3 and is planned to be removed in 2021.1
  - High order APE sources option has been deprecated starting in Simcenter STAR-CCM+ 2020.3 and is planned to be removed in 2021.1.
  - Please contact your Simcenter customer support representative if you have any questions or concerns.

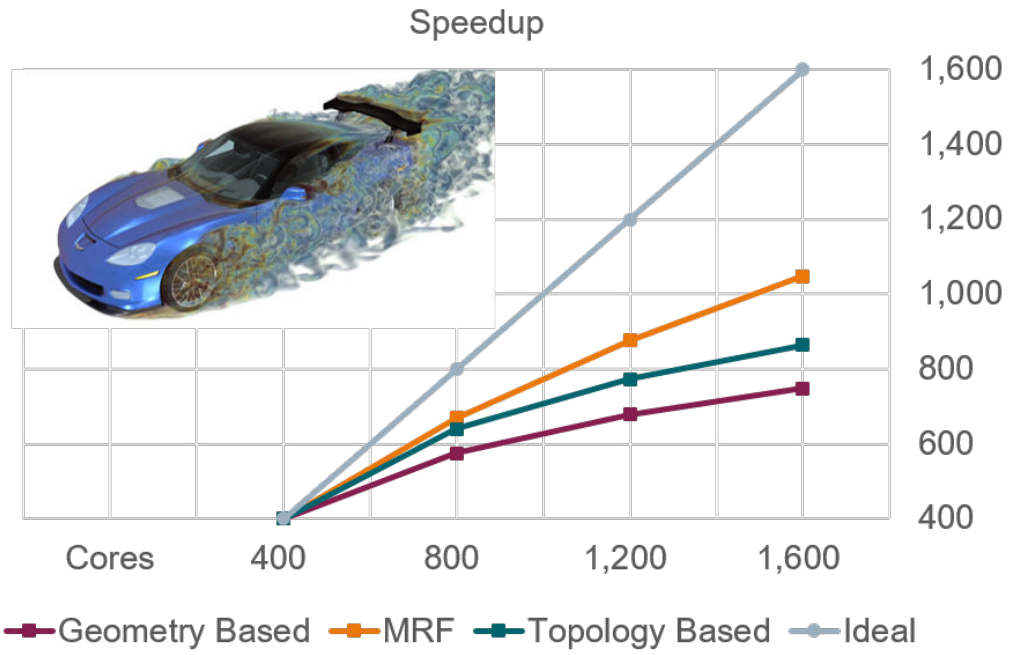
Harmonic Balance

- **Deprecated Harmonic Balance Field Functions now hidden**
  - Simpler workflow with the use of standard field functions in scenes, plots, and reports
    - The field functions used by the legacy visualization method have been removed from the field function selection list and the legend right-click menu in scenes
  - The macros using deprecated field functions will continue to work as expected
  - User-defined field functions can continue to be defined using the deprecated field functions
- **Reference Frame support for Harmonic Balance Field Functions**
  - Improved workflow for Harmonic Balance solution visualization, consistent with the steady solver
    - Previously, harmonic balance field functions such as total pressure or velocity were only available to be visualized or reported in the Lab reference frame
    - Now it is possible to plot, visualize, and report these quantities in any reference frame present in the simulation



Motion, Mesh Adaption, and Mapping

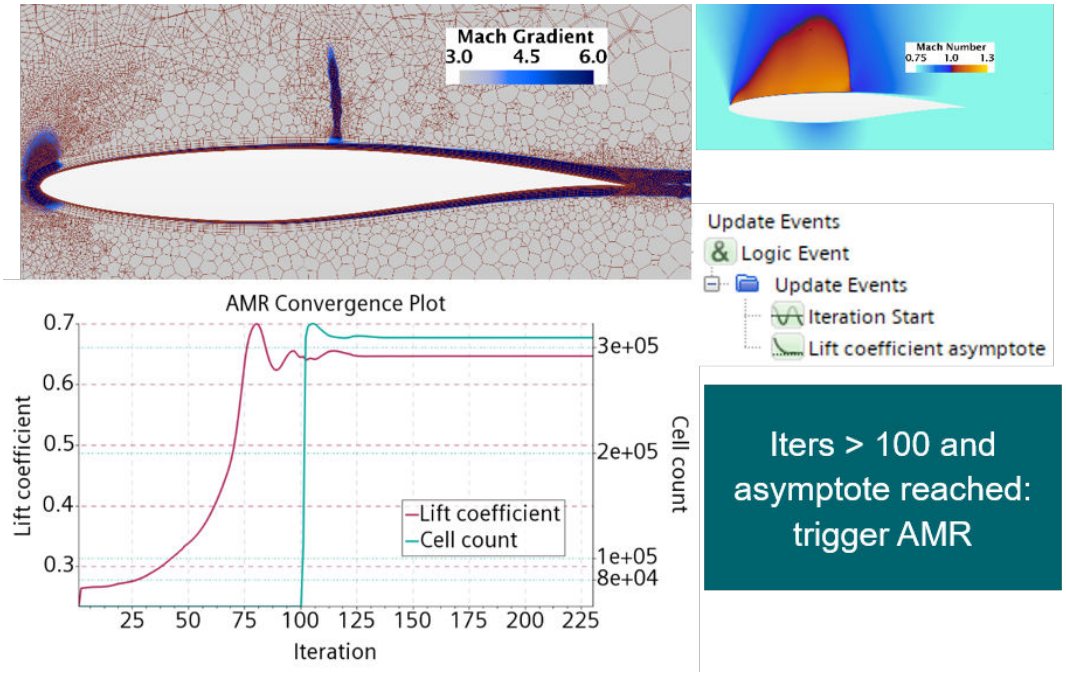
- **Enhanced performance of sliding mesh interfaces on high core counts D916**
  - 1.3x faster transient drag prediction for a passenger car on 1600 cores, using rigid body motion and sliding mesh to model the wheels
  - Previous legacy **Geometry-based** sliding mesh interface was serial
    - Restricted scaling and performance on high core counts
  - New **Topology-based** sliding mesh interface utilizes multi-threading
    - Shared memory parallelism improves scaling on high core counts
  - Applications: Unsteady drag prediction of rotating automotive wheels, electric motor cooling, VTM fan, gear boxes etc.



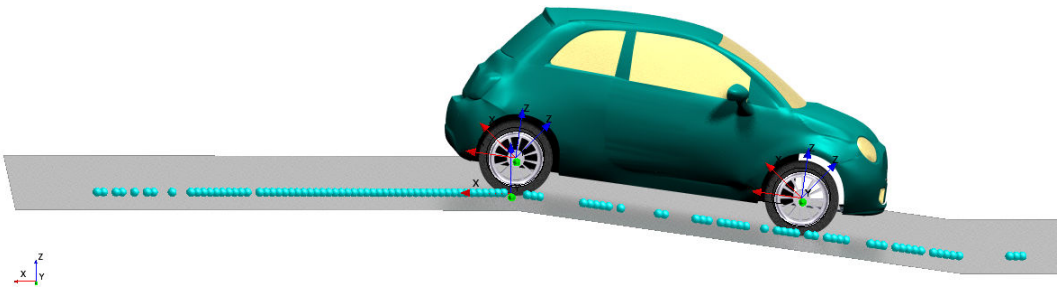
- Speedup dependent on ratio of number of cells in the core volume mesh to number of faces on the interfaces

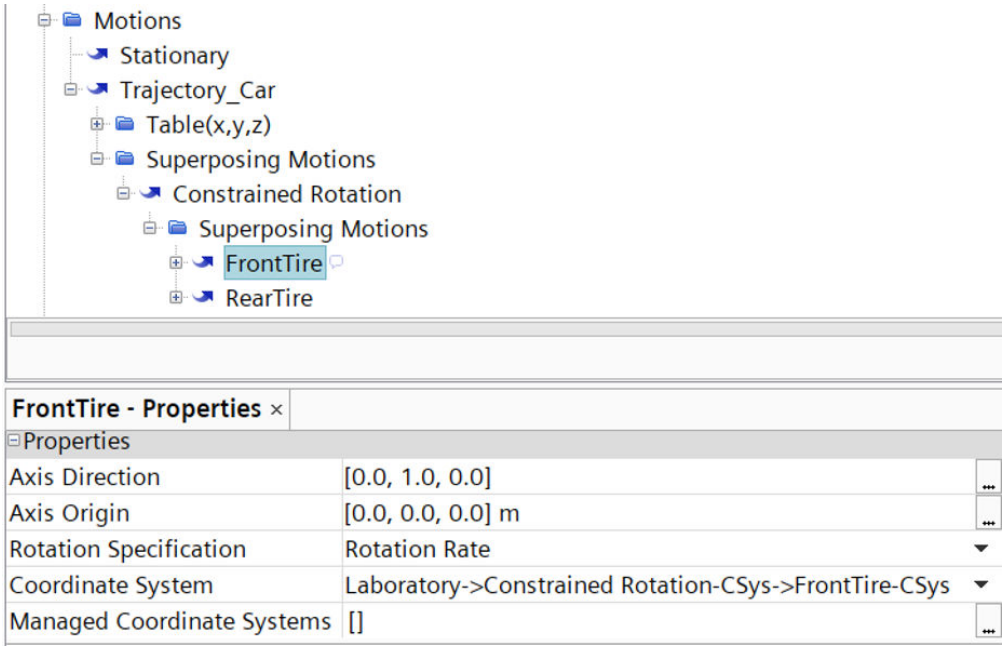
Case	Cells in core volume (Nvol)	Faces on the interface (Nint)	Ratio (Nvol/ Nint)	Cores	Speedup
Passenger car A	172 M	200 K	860	1600	1.22x
Passenger car B (1/2 model)	73 M	400 K	183	1600	1.33x
Electric motor oil cooling	7.2 M	1.78 M	4	160	1.34x

- **Trigger Adaptive Mesh Refinement (AMR) through Update Events** D5467
  - On-demand, flexible, triggering allows for workflows where flow must develop prior to mesh adaptation starting
    - Previously only constant AMR periods were supported
    - Now **Tools > Update Events** framework can be used to trigger AMR
      - Example: Trigger based on minimum number of iterations, asymptote reached, etc.
    - Note: Only Delta-time trigger is available for free-surface based AMR
    - Applications: Shocks, wake dominated flows, combustion

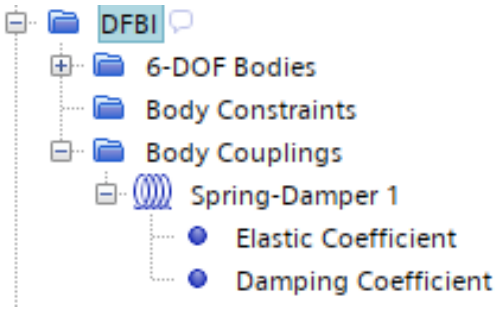


- **Extended capabilities for Adaptive Mesh Refinement**
  - AMR now compatible with:
    - Multi-component liquid and reactions
    - Participating media radiation (DOM and P1)
    - User-defined equations of state (EoS)
    - See Physics section for more details
  - Seamless propagation of AMR transition zones across internal interfaces
  - "Clear Adaption" checkbox added to "Clear Solution" dialog box
    - In previous versions this was only accessible via ***Solvers > Adaptive Mesh***
- **Superimpose rigid body rotation on trajectory motion**
  - Easily set up superposing rigid body rotation on trajectory motion
    - Now possible to superpose rigid body rotation on constrained, tangential, and vantage point rotations
      - Coordinate system for superposing rigid motions created automatically
    - Enables accurate vehicle wading simulations with rotating wheels

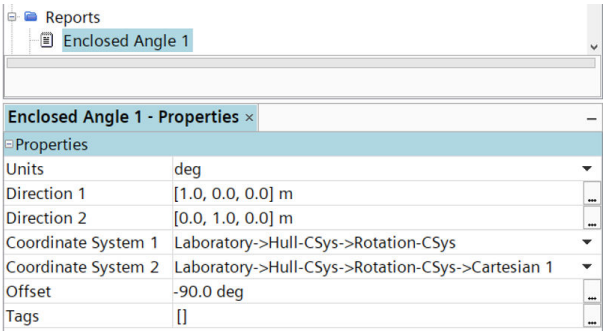




- **Specify field functions as input to trajectory motion**
  - Previously tables were used
  - Now analytical functions can be defined via field functions
- **Multi-body coupling: Spring and Damper**
  - Model spring-damper couplings in Dynamic Fluid Body Interactions (DFBI)
    - Previously linear spring coupling only
    - Now one can specify both elastic and damping coefficients
    - Useful for realistic simulation of spring-damper sub-systems
      - Applications: Realistic vehicle wading
    - Report to monitor spring-elongation available via **Reports > New Report > DFBI > Spring Elongation**

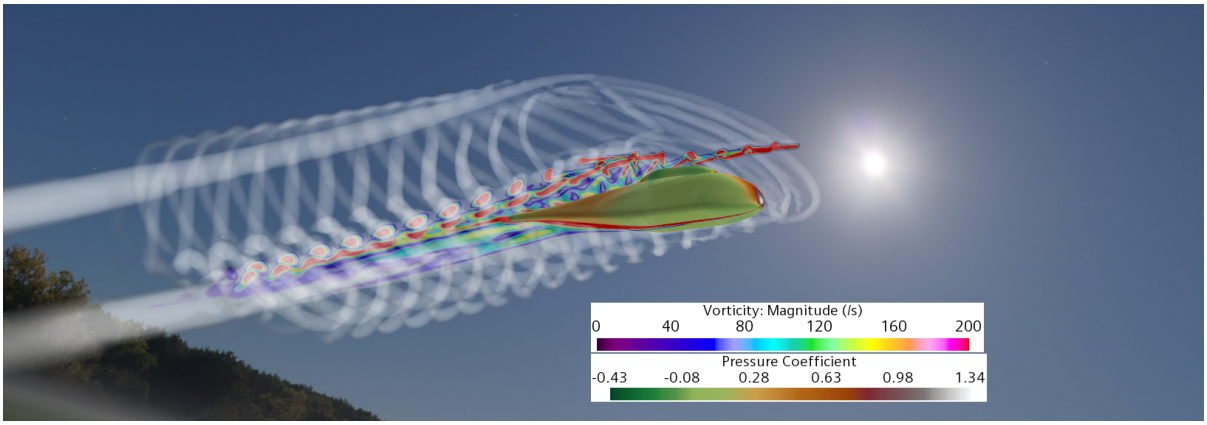


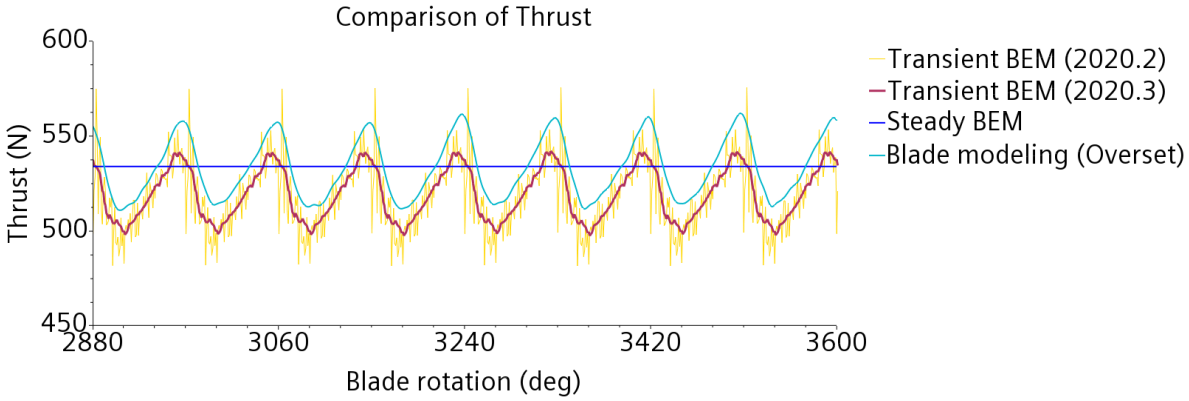
- **Enclosed angle between two vectors in relative motion D3448, D1983**
  - Easily measure angle between two vectors defined in different coordinate systems
    - Example: Measure the angular position of a rudder attached to a ship in maneuvering simulations
    - Offset value can be used to obtain a signed angle
      - Example: An offset value of -90 returns (-90, +90) instead of (0,180)
    - Report can be created via **Reports > New Report > User > Enclosed Angle**



- **Smoother source terms for transient Blade Element Method**
  - Reduced high frequency noise in unsteady blade element method
    - Previously uniform shape distribution of source terms was used

- Now shape functions are used which provide a smoother distribution
  - More accurate flow field
  - Smoother thrust, rolling, and pitching moments





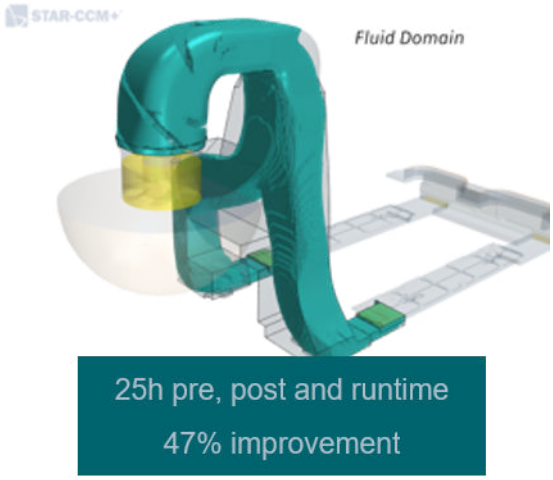
- **Improved accuracy for overset meshes at Symmetry Boundaries**
  - Previously linear interpolation was not always possible on symmetry boundaries, causing flow inaccuracies
  - Now, linear interpolation is applied
- **Resolved islands of active cells in overset zero gap zone**
  - Reduced risk of solver divergence by turning islands of active cells in the zero gap zone as inactive
  - Example: Car wheels in contact with the ground
- **Use Parts to define volume control Point Sets for morphing**
  - Example: Protect mesh surrounding spark plug from being morphed in in-cylinder combustion engine simulation

Design Exploration

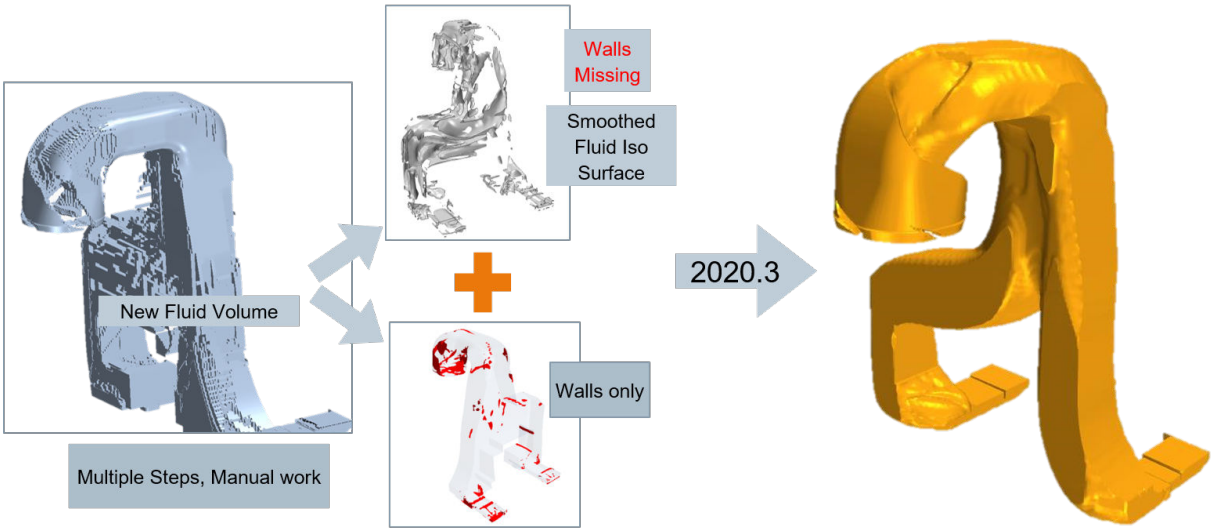
Adjoint

- **Adjoint Topology Optimization D4350**
  - End-to-end topology optimization workflow for generative engineering with new design exploration approach
    - Supports both flow and thermal optimization problems

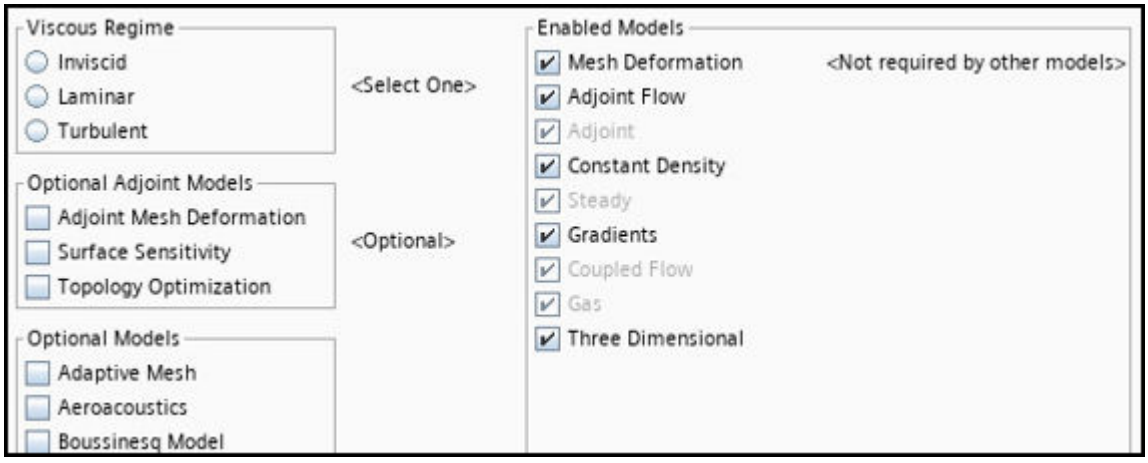
        - Automatically finds best performing designs based on a given design space
        - Create non-intuitive designs that traditional approaches may not discover



    - Built-in constrained optimization method
        - One objective — multiple constraints including volumetric constraint
    - Stable method with level set approach for cold flows and Conjugate Heat Transfer (CHT)
        - Clean designs with fewer kinks and folds
        - Solid represents areas where flow is blocked; but the the conduction, temperature, is solved
- **Closed Isosurface for Topology Optimization**
    - New "closed" option for isosurfaces to provide easy workflow to export the resulting design from topology optimization
        - Automatically creates smooth and closed surface that requires minimal cleaning

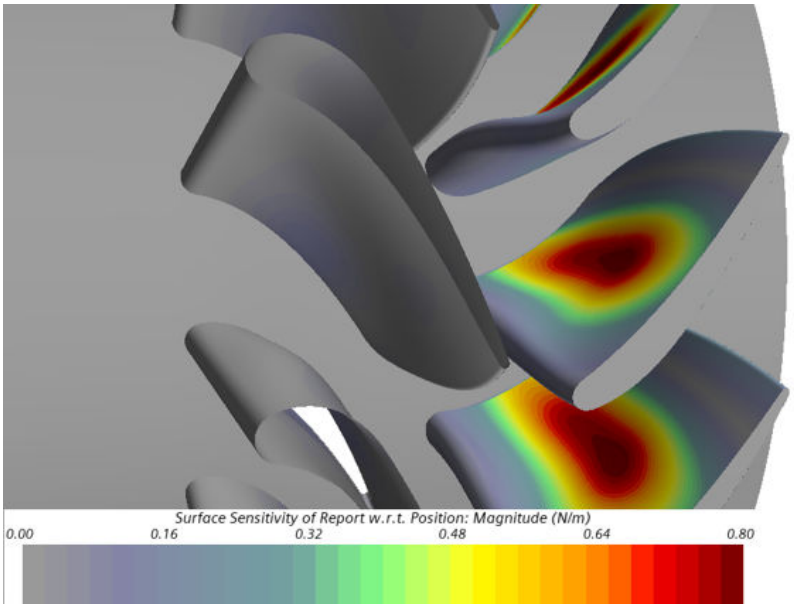


    - Save time extracting the watertight and smooth derived part formed using the material indicator field function
    - May be exported to Geometry Parts for further manipulation and analysis
- **Selection Panel for optional Adjoint models**
    - Improves the user experience to set up adjoint applications with a dedicated optional adjoint models section



- No need to search in the long list of optional models
- Options for Adjoint Mesh Deformation, Surface Sensitivity, and Topology Optimization

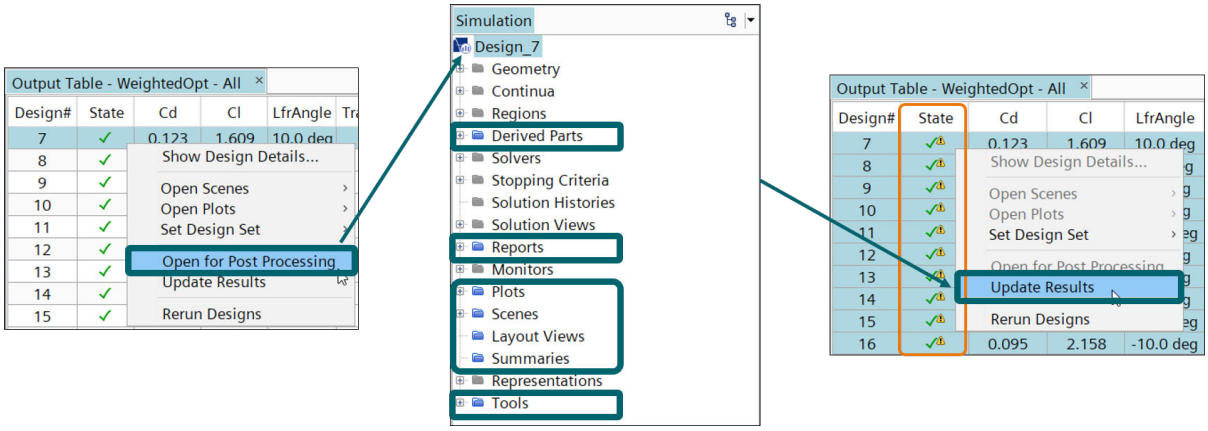
- **Implicit Mixing Plane compatibility with the Adjoint solver**
  - Enables adjoint sensitivity analysis for multi-row turbomachinery simulation
    - Increases accuracy of primal simulation as compared with simulation of an isolated blade-row



  - Mixing plane is widely used in turbomachinery but also headlamp applications and others cases with rotating parts

Design Manager

- **Asynchronous post-processing D4622**
  - Increase product insights and knowledge by extracting additional analysis data after study execution
    - Refine and adapt existing data analysis
    - Execute additional post-processing to investigate new trends
  - Create once and apply to hundreds after study execution
    - Create new or modify existing post-processing objects for any design
    - Automatically propagate modified or new objects to all designs or only selected designs



- **Additional information printed in log file D4859**
  - Increase information accessibility in log file to better identify the associated design run
    - Parameter name and value, report value, .sce and hardcopy file names

```
---> Executing Design Operation : Set Parameters

List of parameters to be set ...
    Simulation Parameters
        Vector : [-0.500000, 0.000000, -0.500000]  (in)
    3D-CAD Model 1
        depth : 0.075000  (m)
        extrude : 0.450000  (m)
        thickness : 0.007500  (m)
```

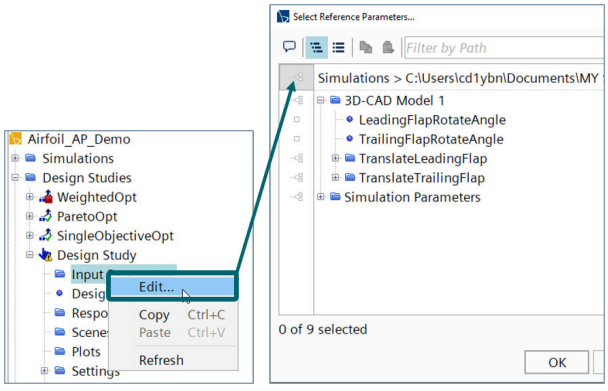
```
---> Executing Design Operation : Get Report
    ShearStress : 0.01903719881638014 (Pa)

---> Executing Design Operation : Export Scene
    Export to scene file "Geometry Scene 1.sce"

---> Executing Design Operation : Export Scene
    Export to scene file "Mesh Scene 1.sce"

---> Executing Design Operation : Export Hardcopy
    Export to hardcopy "Velocity.png"
```

- **Edit shortcut for Input parameters of Manual Studies**
  - Easily select parameters directly from the Input Parameters node

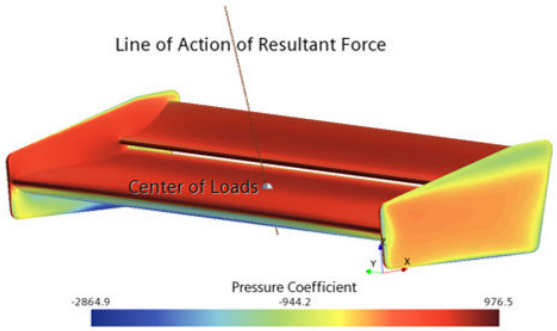


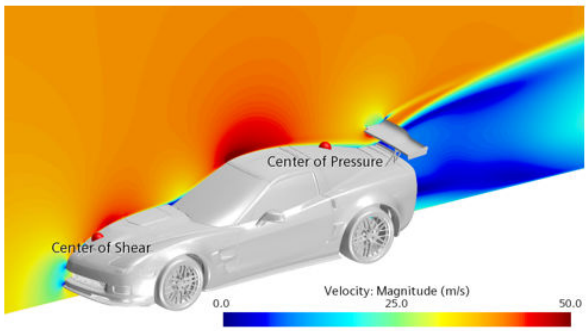
- **HEEDS|post access from Design Manager change notice**
  - Starting 2021.1

- HEEDS|post will not be packaged with the Simcenter STAR-CCM+ Intelligent Design Exploration license anymore
- HEEDS|post will still be accessible from the Design Manager interface with:
  - a standalone HEEDS|post license (one license checked out per session)
  - a separate HEEDS|post installation

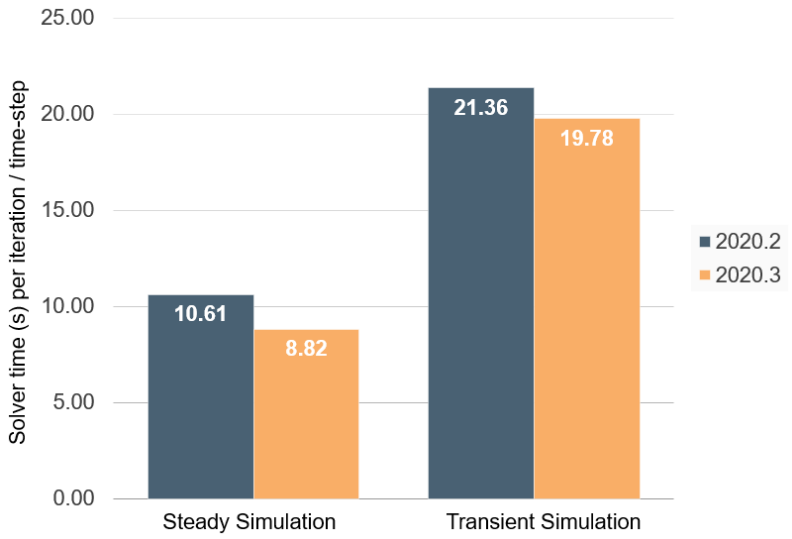
Data Analysis

- **Center of Loads D2445**



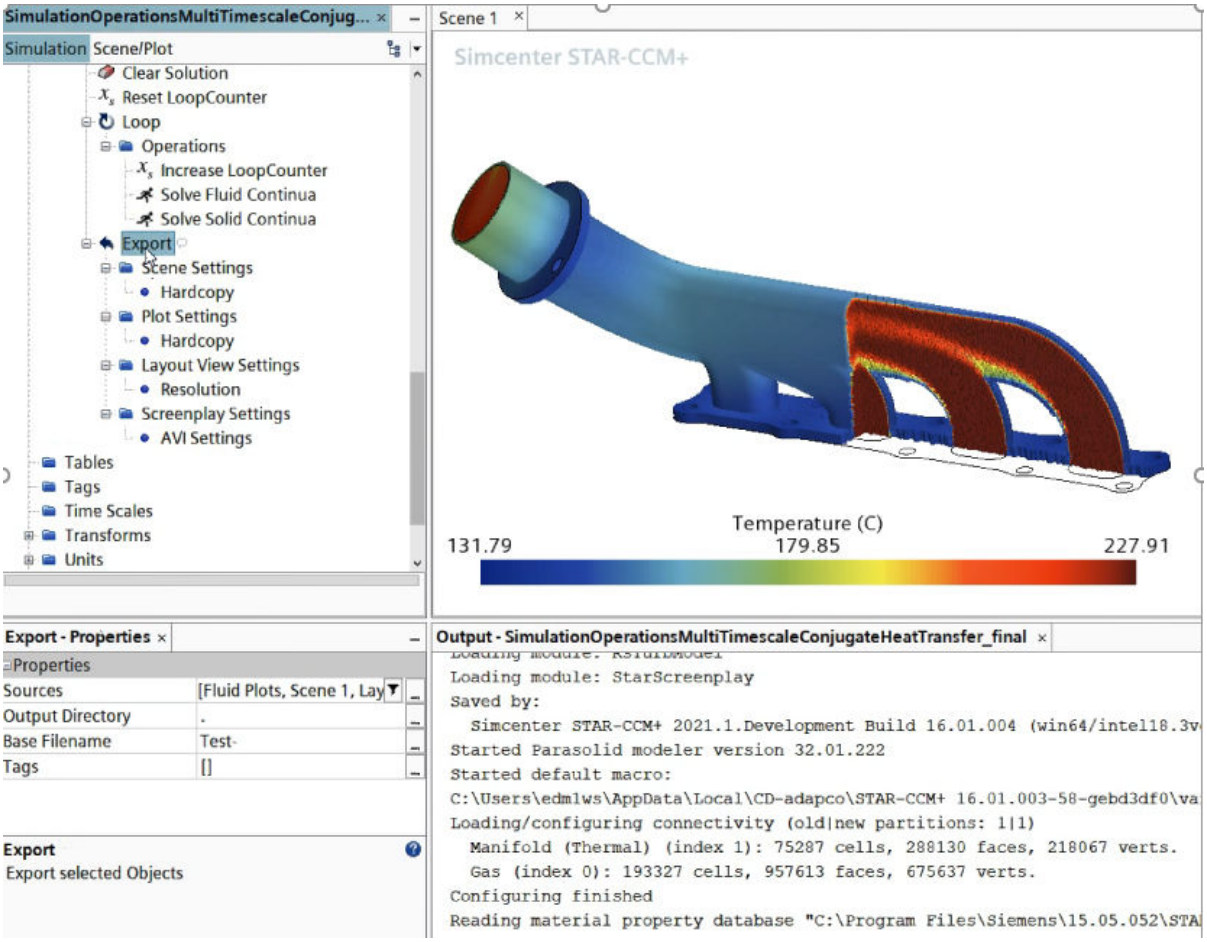


  - Quickly calculate Center of Loads to assess aerodynamic balance
    - Avoid complex workflow setup or manual calculations
    - Obtain intersection point of line of action of resultant force due to fluid flow (top-left image)
    - Specify type of loads to include - pressure, shear or both (top-right image)
    - Specify location option i.e. part geometry surface or reference plane
  - Computes the Center(s) of Loads of parts individually and returns the total resultant load
  - Current limitation
    - Calculation on boundaries intersecting a symmetry plane does not account for symmetric nature of the loads
      - In such circumstances results may be non-intuitive
- **Improved Frontal Area Report performance**
  - Obtain between 1.08X to 1.2X speedup in simulations utilizing Frontal Area Reports compared to the previous release



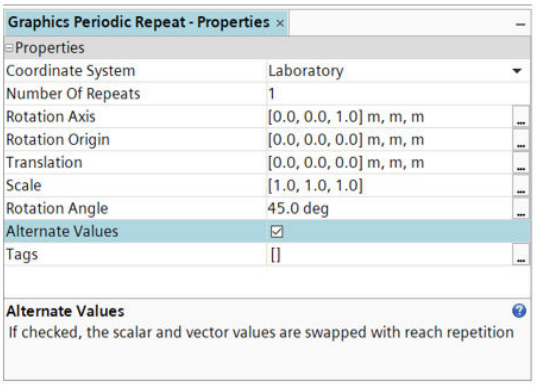
- Performance gains are returned for both steady and transient analysis
  - Benchmark results are shown above for a vehicle external aerodynamics simulation (5 million cells on 4 parallel processes)
- Notes on performance
  - The performance gain can decrease with increasing core counts
  - Changes to the data source while the solver is running will automatically force a report recalculation

- **Solution History support for Statistical Reports**
  - Gain deeper insight into quantitative transient analyses
    - Avoid having to re-run transient simulations in order to do quantitative data analysis
    - Quickly interrogate transient results using Screenplay in combination with Solution History (.simh) data
  - Leverage previously created Statistical Reports
    - Most reports can be computed from a simh representation
    - Statistical reports use the report monitor as the data source
    - No explicit setting for the representation is needed
- **Export Simulation Operations for data analysis artifacts**

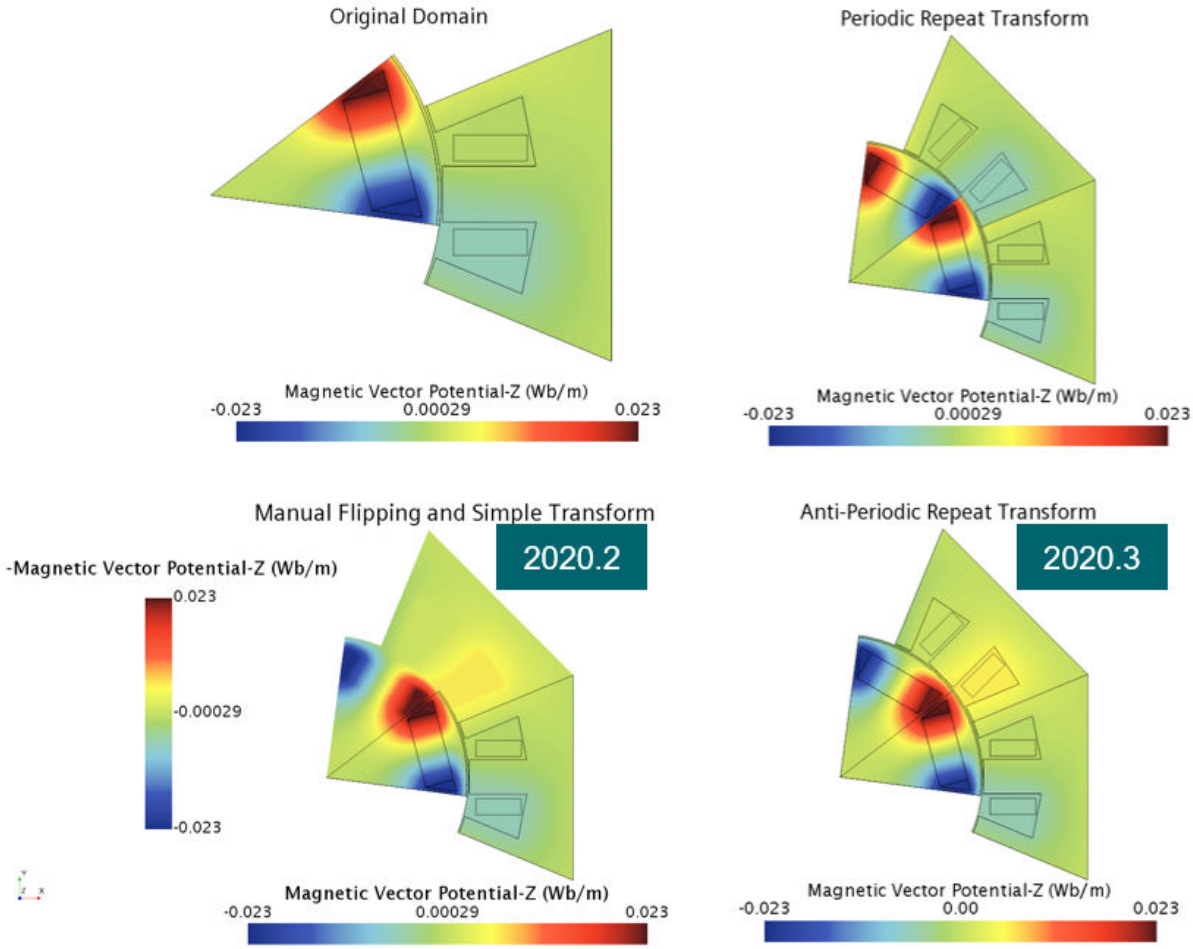


- Enhance productivity by exporting data analysis artifacts within Simulation Operations workflow
  - Easily automate data analysis and creation of artifacts
  - Customize export object properties
- Explicit support for
  - Scenes - Hardcopy creation and Scene file export
  - Plots - Hardcopy creation and Scene files or `.csv` export
  - Layout Views - Hardcopy creation
  - Screenplays - Animations through `.avi` or `.png` images

- **Visualization of anti-periodic Geometry**
  - Easily reconstruct and visualize full domain from anti-periodic interfaces



- In a series of images below

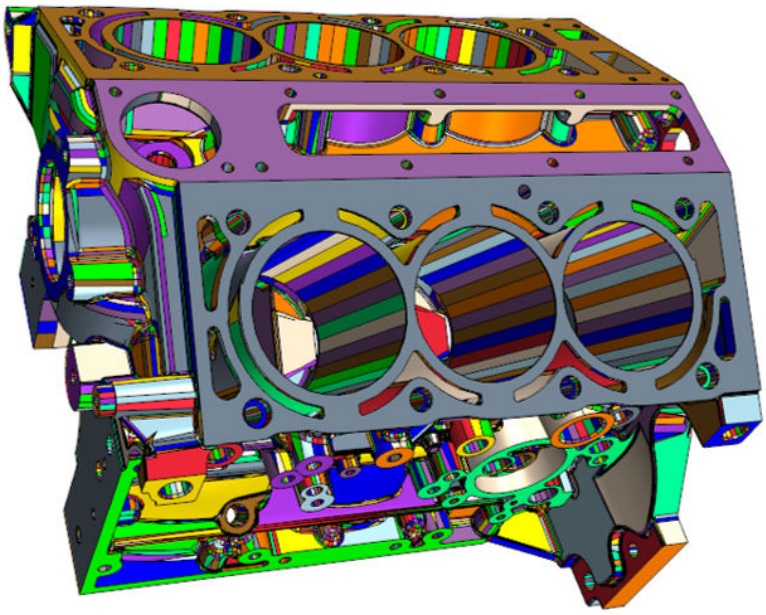


  - Top-left: Original domain Magnetic Vector Potential-Z variation
  - Top-right: Periodic displayer transform applied
  - Bottom-right: Anti-periodic displayer transform applied using simple checkbox option in 2020.3
  - Bottom-left: The previous need to define a user field function to negate values is now obsolete
- Important note: Support for Anti-periodic transforms for reporting are planned for a later release

- **Faster animation exports and hardcopies**
  - Gain up to a 1.32X speed-up in exporting single hardcopies or Screenplay animation exports
  - Leverage increased performance gains relative to applied image resolution (see table below)

Applied image resolution	Speedup of Screenplay animation (451 PNG files) creation	
	With Anti-Aliasing	Without Anti-Aliasing
960 x 540	1.12X	1.12X
1920 x 1080	1.14X	1.18X
3840 x 2160	1.14X	1.32X

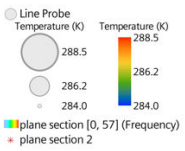
- **Improved Scene rendering with many-part surfaces**

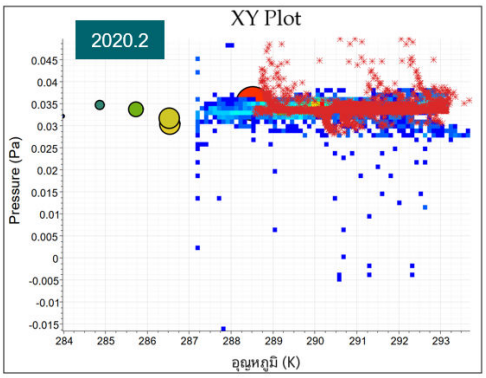


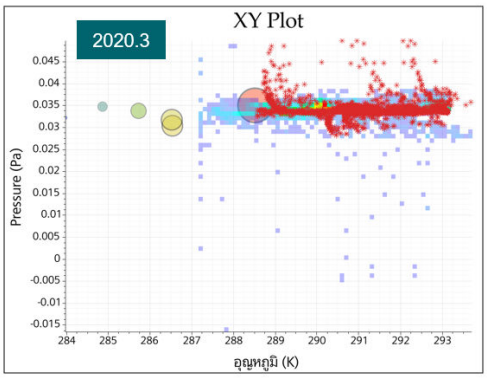
- Speedup of rendering of scenes with high numbers of part surfaces
  - Lowered memory usage for client and server processes
  - Reduced geometry Scene rendering time
- Table below shows performance benefits for rendering geometry Scene shown above containing 37614 part surfaces

Version	Client Process		Server Process
	Memory Usage	Render Time	Memory Usage
2020.2	5.1 GB	0.075s	1.7 GB
2020.3	3.1 GB (1.7X)	0.06s (1.2X)	1.2 GB (1.4X)

- **Enhanced PDF Plot Export**
  - Improve your plot hardcopy appearance with an improved PDF export capability to support scalable Vector graphics
    - Text and symbols will look sharper
    - Transparency for plot symbols is now supported for bubble plots and heatmaps
    - Plot hardcopies can be scaled without loss of resolution

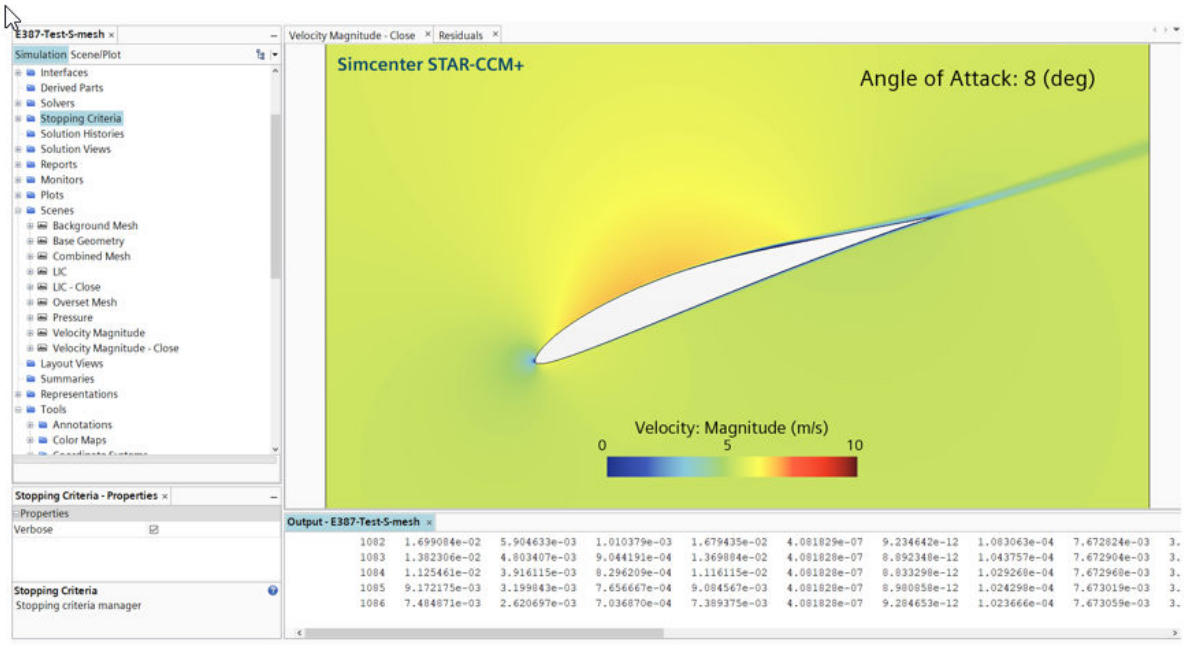






- Important notes:
  - PostScript support is discontinued in 2020.3 for plot hardcopies
  - Support for annotations in exported PDF files is planned for a later release

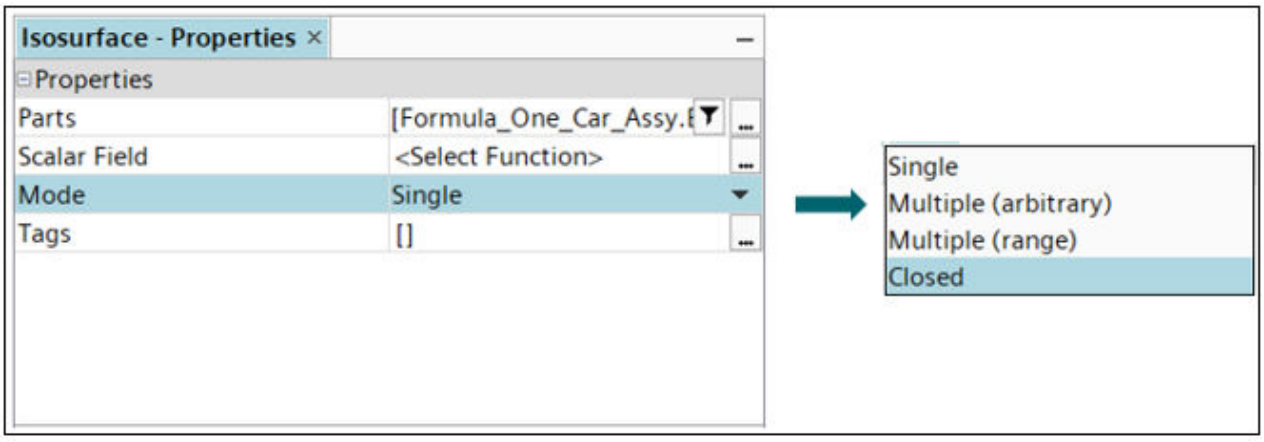
- **Reduced Scene rendering time during solver runs**



  - 2.3X speed-up in Scene rendering time during solver runs compared to previous versions
    - Time saving realized when rendering a scene is longer than a solver iteration
    - Less instances of 'Waiting for Client catch up' messages in console expected
  - Table below shows performance benefit for a static geometry scene shown above with only scalar displayed data change

Version	Render running time per iteration (with Nvidia P4000 graphics card)
2020.2	0.7s
2020.3	0.3s (2.3X)

- Current limitation
  - Performance benefit will not be realized when render time is already faster than the solver or render/server processes are not sharing resources
- **Finite Element driven interpolation for all Derived Parts**
  - Improves the data analysis accuracy when working with Finite Element models
  - Previously only Point Probes, Line Probes, and the Presentation Grid were supported
- **Closed Isosurface option for Derived Parts**

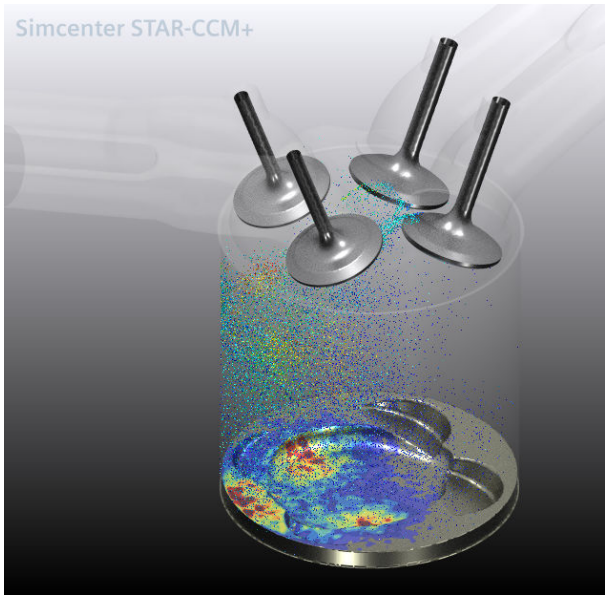


- Create closed isosurface with the new **Closed Mode** option
  - Creates a closed surface bounded by the iso-surface and the input boundaries or part surfaces
- Can be used to easily visualize the wetted surfaces for filling processes
- Specifically developed for the extraction of shapes from adjoint topology optimization analysis
  - See Design Exploration section for more detail

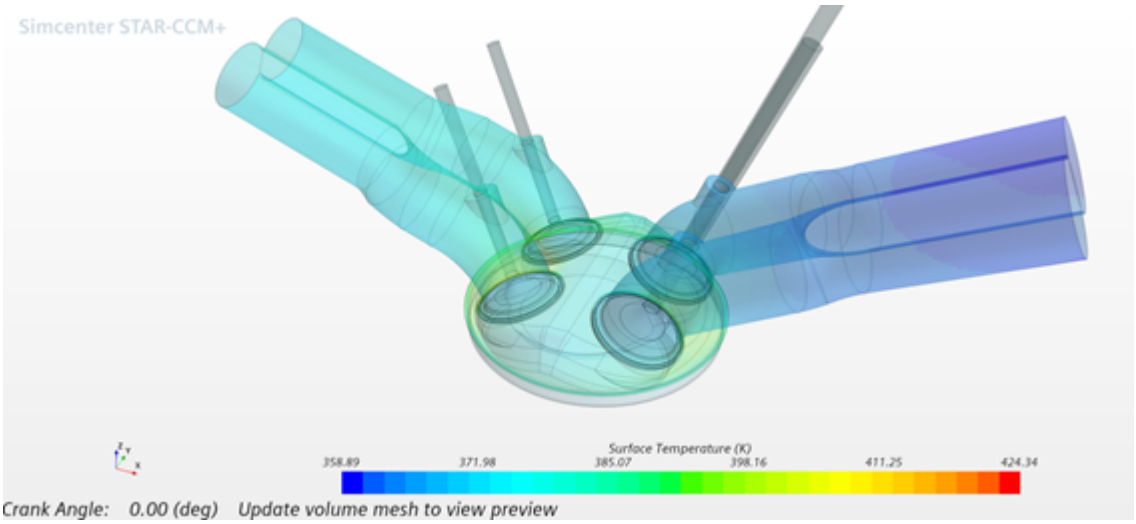
Application Specific Tools

In-cylinder Solution

- **Setup Liquid Film Model**
  - Add more realism/fidelity to simulations where high fuel loading means films/pools form on surfaces (ports/piston/liner/valves)
    - Improve prediction of cylinder pressure due to less vapor fuel to burn
    - Improve prediction of emissions due to locality of fuel species
    - Provide insight into onset of “pool fires” on piston
- **Import/Apply Spatially-Resolved Temperature Boundary Conditions**
  - Use detailed temperature boundary conditions for all walls with spatially-resolved temperatures
    - Improved accuracy for heat transfer predictions
  - Quick to setup in the In-cylinder Solution workflow
    - Table XYZ static temperature option in the drop-down selector for each surface's Edit panel



- Requires update of volume mesh in order to preview the applied field



- **Multi-pulse Injection Support for Huh Break-up Model Setup**
  - Minimize time consuming manual setup of Huh model
    - Save significant time in manual calculations and re-calculations when updating relevant injector properties for Huh break-up model
    - Automatically detects the number of pulses in the imported injection profile
    - Calculates the mean injection velocity for each pulse, individually, discarding dwell periods between pulses; stores the profile of time vs. mean injection velocity in an internal table
    - Recalculates whenever a relevant input is updated (e.g. the table, nozzle diameter, fuel density, etc.)
- **Apply Rigid Prism Boundary Layer on all Wall Boundaries**
  - Improves consistency of y+ value in boundary layer of walls by not allowing morphing to compress/ stretch the prism layer
    - Cells will morph only when wall contact is imminent, similar to overset approach (e.g. valve closing)
    - Automatically applied to all wall boundaries in the simulation setup by In-cylinder Solution; can be de-activated manually in the Simulation tree if required
- **Assisted Setup of 2-Stroke Simulations with External CAD Bodies**
  - Setup 2-stroke simulations using external geometry (CAD bodies) for an intake manifold faster, with less input error
    - Supports import of plenum for intake (2-stroke only)
      - Requires body to be named with either “Intake” or “Plenum” with an “Inlet” boundary
      - Creates an additional region under “Regions” in the Simulation tree (e.g. “Gases 2”)
      - Multiple plenums allowed
- **Up to 4 exhaust valves now allowed for In-cylinder configuration**

Electronics Cooling

- **Support added for saving/loading imported IDF/ODB quickparts**
  - Supports:
    - IDF chip
    - IDF generic part
    - IDF/ODB PCB

User Guide

- **New Tutorials**
  - Incompressible Flow
    - Adjoint Topology Optimization: Channel Flow with Minimized Pressure Drop
    - Anisotropic Flow: Cyclone Separator
  - Coupling with CAE Codes
    - gPROMS File Export: Spray Dryer
- Modified Tutorials
  - Introduction Tutorial – Minor change in the order of setting boundary types
  - 3D-CAD: Cyclone Separator – Update to steps for creating the inlet
  - Co-Simulation API: Spindle Valve – Partner program now takes time-step from the Simcenter STAR-CCM+ simulation
  - Volume Rendering: Steckler Room – Updated to use custom unit in scalar scene
  - Eddy Break-Up: Coal Combustion – Velocity URF set to 0.5
  - Abaqus Co-Simulation: Mechanical Coupling – Updated to use Abaqus 2018
  - Abaqus Co-Simulation: Thermal Coupling – Updated to use Abaqus 2018
  - GT-SUITE Co-Simulation: 1D Coupling – Now uses `.gtm` input file rather than `.dat`
  - Simcenter Nastran Co-Simulation: Disc Valve – Added steps for creating data mappers
  - Flamelet Generated Manifold: Perfectly Premixed Combustion with Adaptive Meshing – Removed the Bulk Outlet Temperature monitor plot
  - Rigid Body Motion: Rotating Fan – Updates to under-relaxation factors
- New Cases in Verification Suite
  - Flow and Energy
    - Unsteady Compression Molding of a Newtonian Fluid
  - Electromagnetism
    - TEAM Problem 23: 3D Static Force Problem in a 36-Degree Body
    - TEAM Problem 30a: 2D Induction Motor Analysis

Siemens Digital Industries Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific
Suites 4301-4302, 43/F
AIA Kowloon Tower,
Landmark East
100 How Ming Street
Kwun Tong, Kowloon
Hong Kong
+852 2230 3308

About Siemens Digital Industries Software

Siemens Digital Industries Software is driving transformation to enable a digital enterprise where engineering, manufacturing and electronics design meet tomorrow. Our solutions help companies of all sizes create and leverage digital twins that provide organizations with new insights, opportunities and levels of automation to drive innovation. For more information on Siemens Digital Industries Software products and services, visit siemens.com/software or follow us on LinkedIn, Twitter, Facebook and Instagram.

Siemens Digital Industries Software – Where today meets tomorrow.