CFD EXPERTS

Simulate the Future

WWW.CFDEXPERTS.NET

Ansys Fluent Migration Manual

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2021 R2
July 2021

Published in the U.S.A.

Table of Contents

List of Figures

Chapter 1: Migrating to Ansys Fluent 2021 R2

The purpose of the Ansys Fluent Migration Manual is to help you transition from the previous version for the meshing mode and solution mode of Ansys Fluent. Read through the entire document to understand the changes that have taken place. The information is enclosed in the following chapters:

If you are upgrading across several releases, you may find it helpful to consult the Migration Manuals for the intervening releases as well. You can find these in the Ansys Help on the customer site, by selecting the appropriate release from the drop-down menu on the Home page.

When you write journals using this release, then you should add the text command `/file/set-tui-version "21.2"` at the top of the file. This text command will help when you run the journal in a future release, as it will hide any new text user interface (TUI) prompts that have been added and will restore any prompts that have been removed. It is intended to help with backwards compatibility with future releases (up to two full releases after the release the journal file was created in/for).

For information about past, present, and future operating system and platform support, see the Platform Support section of the Ansys Website.

Note that beta features have not been fully tested and validated. ANSYS, Inc. makes no commitment to resolve defects reported against these prototype features. Changes in the solution behavior are sometimes expected and are not captured in this document. However, if there are solution differences between beta features and the Release 2021 R2 standard Fluent features that concern you, contact technical support for assistance. Your feedback is appreciated and will help us improve the overall quality of the product.

Chapter 2: Fluent Release Notes

The following sections contain release information for Ansys Fluent 2021 R2.

Backwards Compatibility: Ansys Fluent 2021 R2 can generally read case files and data files from all past Fluent releases. Solver and model settings from previous case files are typically respected. However, in some cases due to defect fixes and core improvements to improve robustness and/or performance, convergence behavior and/or results obtained may be different. Such release-to-release changes are documented in the Fluent Migration Manual, along with instructions to recover the previous behavior when possible.

2.1. Supported Platforms for Ansys Fluent 2021 R2

Information about past, present, and future operating system and platform support is viewable via the Ansys website.

2.2. New Features in Ansys Fluent 2021 R2

The following sections list the new features available in Ansys Fluent:

General User Interface

2.2.1. Meshing Mode

New features available in the meshing mode of Ansys Fluent 2021 R2 are listed below.

General Notes

- By default, when reading or writing mesh files, Fluent uses the Common Fluids Format (CFF) which is built on the Hierarchical Data Format (HDF5). Mesh files using CFF can be identified using the `msh.h5` file name extension.

Fluent Guided Meshing Workflows

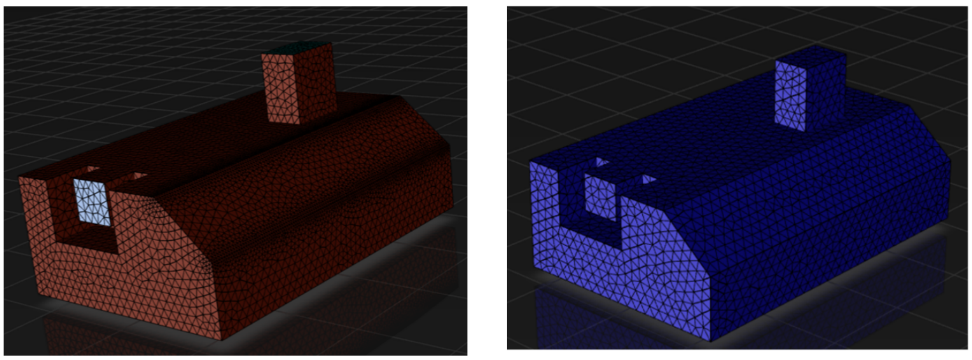
- General

- A better handling of curvature size function in this release has resulted in several improvements:

  1. On rare occasions, curvature size functions were being ignored, however, the problem has now been corrected.

  2. Some models, particularly those with local sizing, have experienced a 5-20% reduction in the surface mesh cell count.

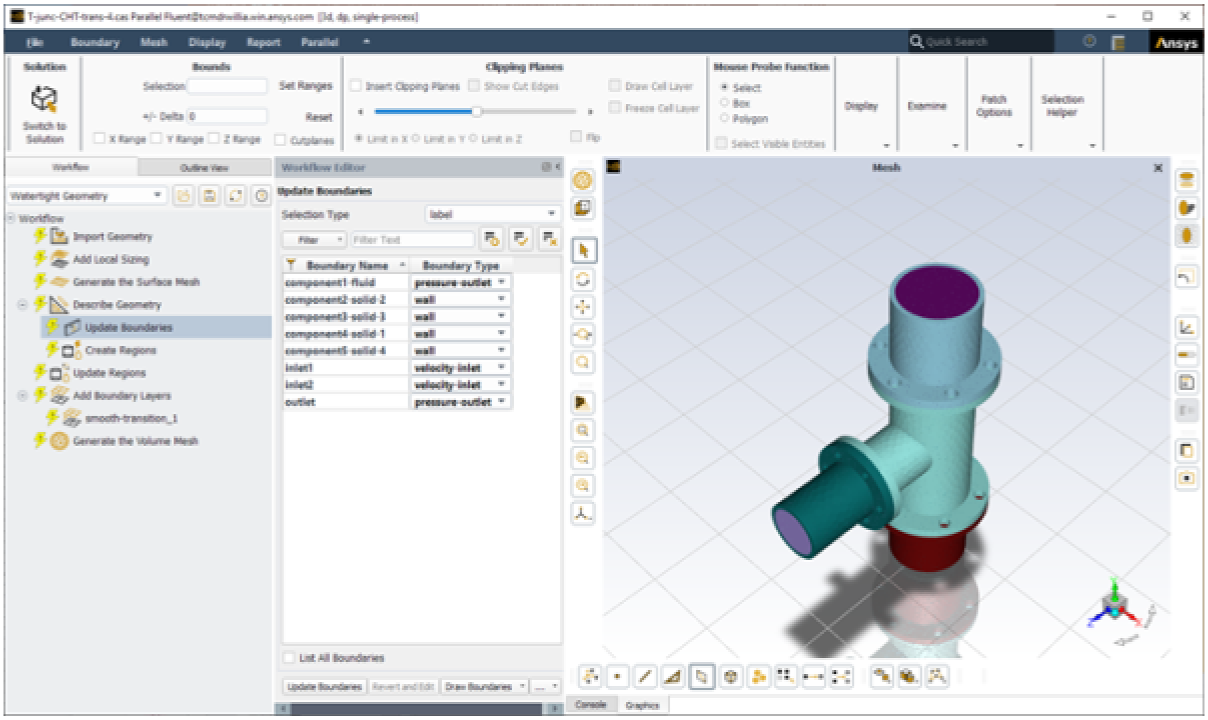
  3. Models that import the mesh and utilize extensive re-meshing, the surface mesh count reduction can be substantial (greater than 50%).



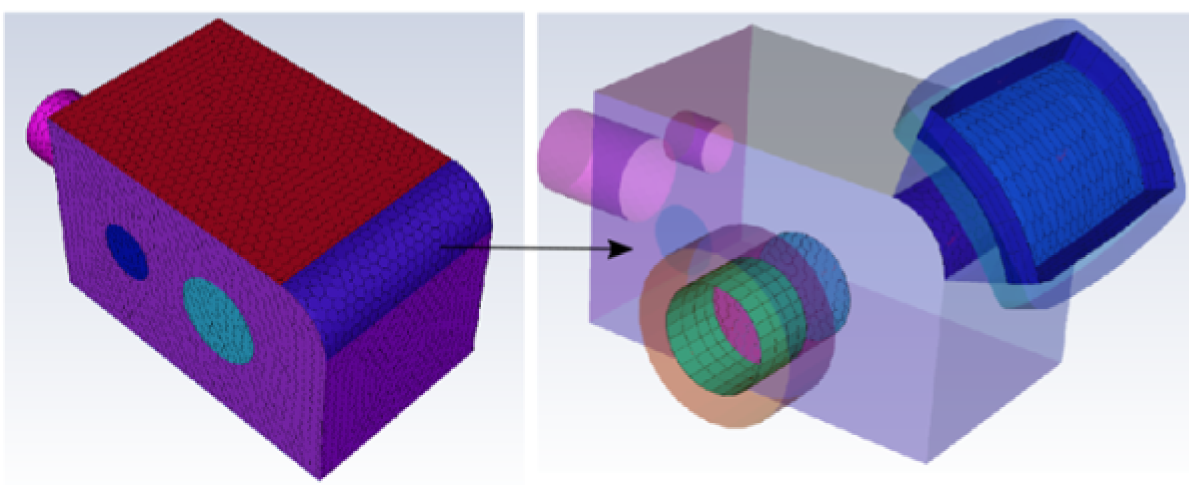
Before After

  In the later two cases, the new meshes should better correspond to provided curvature angle controls.

- For either workflow, under **File > Preferences > Meshing Workflow**, you can now use the **Dockable Workflow Editor** option to be able to separate the guided meshing workflow task steps from their properties (that is, enable the detachable **Workflow Editor**). This allows you to relocate and resize the **Workflow Editor** for easier and convenient access, especially when there is limited space available and several properties in a task. Changing this option will take effect in a new session. See Editing Tasks.



- The **Watertight Geometry** Workflow
  - You can now read boundary (prism) control files into the **Add Boundary Layers** task. See Adding Boundary Layers for details. Likewise, if you have added or read in any boundary layer settings, you can write out a boundary (prism) control file in the **Generate Volume Mesh** task. See Generating the Volume Mesh for details.
  - You can now extrude planar and non-planar volume meshes using the **Extrude Volume Mesh** task. See Extruding the Volume Mesh for details.



  - The **Import Body of Influence Geometry** task is available so that you can import bodies of influence that are defined in separate CAD geometry or mesh files. See Importing Body of Influence Geometries for details.
  - You can now add the **Set Up Periodic Boundaries** task to your workflow *prior* to the **Generate Surface Mesh** task. See Setting Up Periodic Boundaries for details.

- The **Generate the Volume Mesh** task now includes the **Solid Region Growth Rate** advanced option, allowing you to control the growth rate of all solid regions in the volume mesh (for poly-hexcore meshes where solids are to be filled with polyhedra cells). See Generating the Volume Mesh for details.

- Using the **Invoke Custom Numbering or Pattern** field, the **Add Linear Mesh Pattern** task allows you to create and implement your own customized mesh patterns using a specialized scripting technique. See Adding Linear Mesh Patterns for details.

- This release brings a general change in behavior regarding zone separation, giving you more flexibility in deciding how zones are handled downstream in the workflow. Once you have a volume mesh, cell and face zone management operations (such as merging or renaming) are now more readily available using the new **Manage Zones** task. See Managing Zones for details.

  With the availability of zone management, there are additional changes to the surface mesh and the volume mesh generation tasks. In the **Generate Surface Mesh** task (see Generating the Surface Mesh), the zone separation by angle option has been brought more to the forefront. In addition, in the **Generate Volume Mesh** task (see Generating the Volume Mesh), a zone merge option has also been added if excessive zone management is not required.

- The **Fault-tolerant Meshing** Workflow

  - The **Import CAD and Part Management** task now includes several small but useful usability enhancements when working in the CAD Model or Meshing Model trees:

    → In the CAD Model tree, the **Move Selected to New Object** context menu option allows you to transfer the selected CAD model object(s) onto another object in the Meshing Model tree. The **Alt**+**M** hot key combination is also available for this operation.

    → In the Meshing Model tree, the **Move to a New Object** context menu option allows you to relocate the selected component(s). to a new meshing object in the tree.

    → In the Meshing Model tree, the **List Model Operations** context menu option allows you to see a listing in the console window of any meshing operations that have been defined for the selected object. See Listing Meshing Operations for more information.

    → In the Meshing Model tree, additional icons (, , and ) have been added to indicate that the components have associated meshing operations (such as transformations or re-faceting).

  - The **Import CAD and Part Management** task now includes the ability to apply rotational and translational transformation operations prior to creating meshing objects. See Performing Transformation Operations on Meshing Model Objects for details.

  - The **Import CAD and Part Management** task now includes an improved ability to apply refaceting as a separate operation on one or more portions of your imported CAD geometry prior to creating meshing objects. See Performing Refaceting Operations on Meshing Model Objects for details.

  - You can now define your porous regions by reading a text file into the **Create Porous Regions** task. See Creating Porous Regions for details.

- In the context of overset meshing, the **Extraction Mesh** field in the **Update Region Settings** task now supports **wrap** and **existing mesh** options (in addition to **surface mesh**). See Updating Your Region Settings for details.
- The **Choose Mesh Control Options** task now allows you to use existing target and/or wrap size field files (`*.sf`) when refining your mesh controls in the workflow. See Choosing Mesh Control Options for details.
- The **Choose Mesh Control Options** task now provides more flexibility in how your meshing size controls will be calculated via the new **Compute Size Field For** advanced option. This option allows you to choose whether you specify only the target size control, or the wrap size control, or both. See Choosing Mesh Control Options for details.
- The **Create External Flow Boundaries** task now allows you to directly specify the external bounding box location and position values without having to select any object(s) or zone(s). See Creating External Flow Boundaries for details.
- The arrangement of interface elements in the **Describe the Geometry and Flow** task has been enhanced and improved. See Describing the Geometry and the Flow for details.
- You can now extrude planar and non-planar volume meshes using the **Extrude Volume Mesh** task. See Extruding the Volume Mesh for details.
- The **Generate the Surface Mesh** task now includes the **Auto Assign Zone Types?** advanced option, allowing you to automatically assign inlet, outlet, far-field, interface, and symmetry boundary types to zones during surface mesh creation based on the chosen boundary names. See Generating the Surface Mesh for details.
- The **Generate the Surface Mesh** task now includes the **Global Minimum** advanced option, allowing you to set the global minimum surface mesh value. The default value is based on available size controls and bodies of influence. This leads to an improvement when considering polyhedra and hexahedra cells, where previously, the adjusted minimum size was not utilized and resulted in an increased number of cells. See Generating the Surface Mesh for details.
- The **Generate the Volume Mesh** task now includes the **Avoid 1/8 octree transition in hexcore region?** advanced option, allowing you to avoid any potential 1:8 cell transition in the hexcore or poly-hexcore region of the volume mesh, replacing any abrupt change in the cell size with polyhedral cells. See Generating the Volume Mesh for details.
- The **Enable Parallel Meshing for Fluids** field in the **Generate the Volume Mesh** task now supports polyhedral cells in addition to hexcore and poly-hexcore cells. This option applies to parallel volume *and* continuous boundary layer (prism) meshing. See Generating the Volume Mesh for details.

Mesh Generation

- When using the Rapid Octree mesher, the `mesh/rapid-octree/distribute-geometry?` text command is now supported as a full feature and is enabled by default, so that the input geometry is distributed across partitions / compute nodes rather than being copied to each process. This reduces the memory requirements of the mesh generation significantly, especially for geometries with a high number of triangles. Note that this geometric distribution is automatically deactivated if the geometry is not fully enclosed by the defined bounding box. For details about the Rapid Octree mesher, see Generating Rapid Octree Meshes.

2.2.2. Solution Mode

New features available in the solution mode of Ansys Fluent 2021 R2 are listed below. Where appropriate, references to the relevant section in the User's Guide are provided.

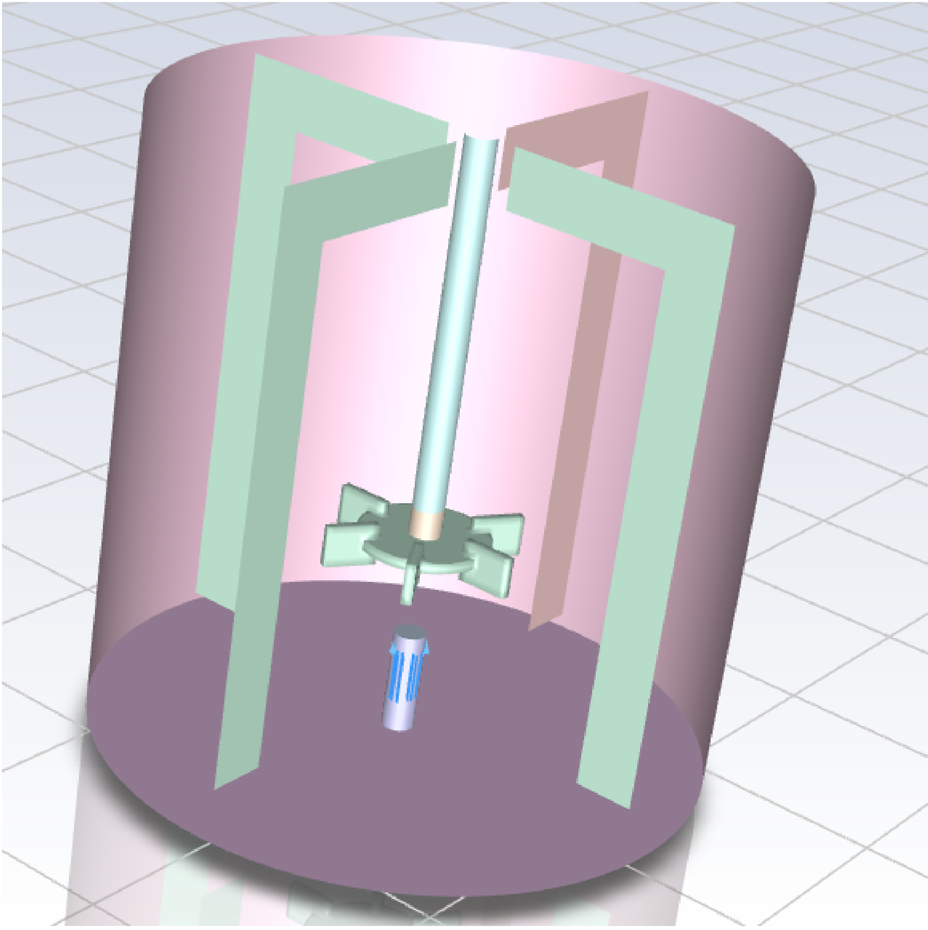
Files

- The speed of case file reading is improved for cases with large numbers of cell zones and/or face zones (10,000+). The observed speed-up is case-dependent, but some industrial cases experience a 3X speed-up.
- Exported Ansys Viewer format (AVZ) files now include any mesh edges which have been included in the display. Refer to Choosing the Picture File Format in the *Fluent User's Guide* for additional information.

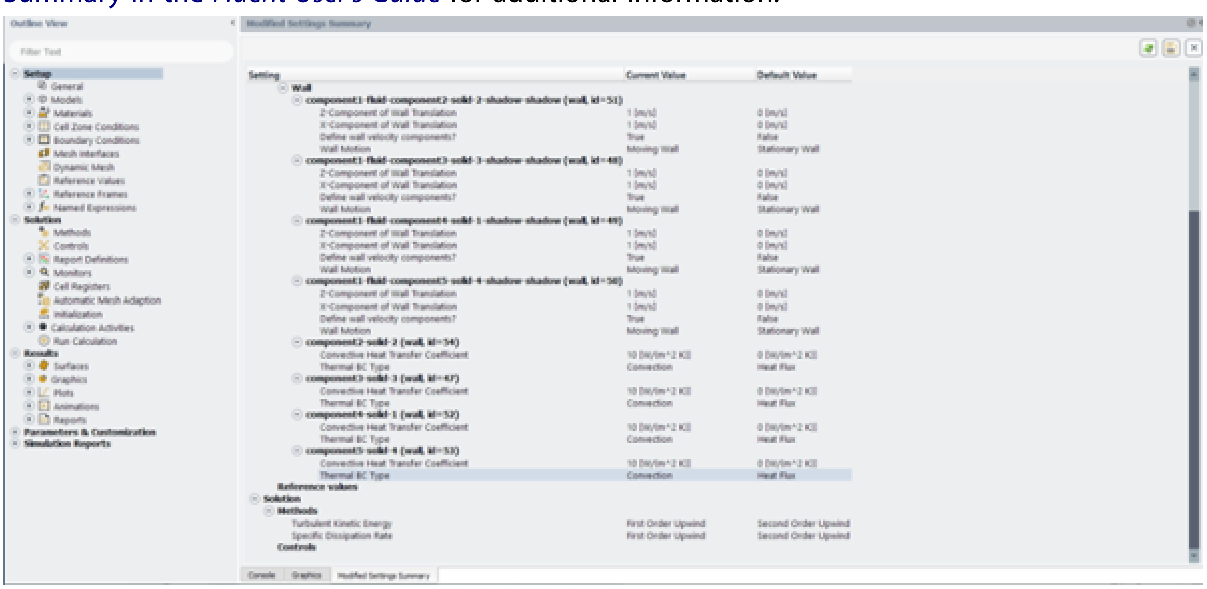
User Experience

- (Large models) Fluent detects when a model's size will cause sluggish interactions and automatically reduces the graphics display during manipulations to speed up the reaction time. How the model is reduced and whether the reduction is done automatically is controlled in Preferences: **Preferences** > **Graphics** > **Performance** > **Fast Interactive Display**.
- You can display the mesh using a pastel color scheme by selecting **Pastel Colors** for **Model Color Scheme** in the **Appearance** branch of **Preferences**.

Figure 2.1: Mesh Display with Pastel Color Scheme



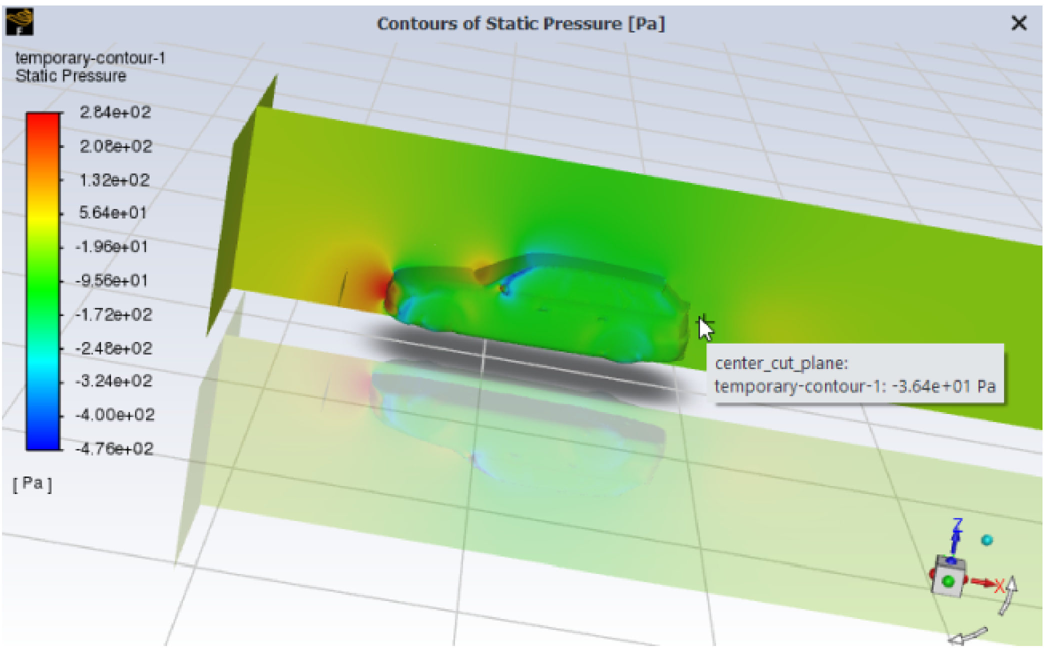
- The new **List Modified Settings** feature lets you easily review the settings that you modified in your current case and compare against Fluent's default values. This applies only to settings which are exposed in the GUI. TUI-only options are not yet included. See Modified Settings Summary in the *Fluent User's Guide* for additional information.



- You can now drag and drop surfaces (such as points, planes, iso-surfaces, and so on) from the Outline View tree to the graphics window to display them. This feature behaves the same as the **Add to** context menu option accessed by right-clicking the relevant branch in the Outline View tree.

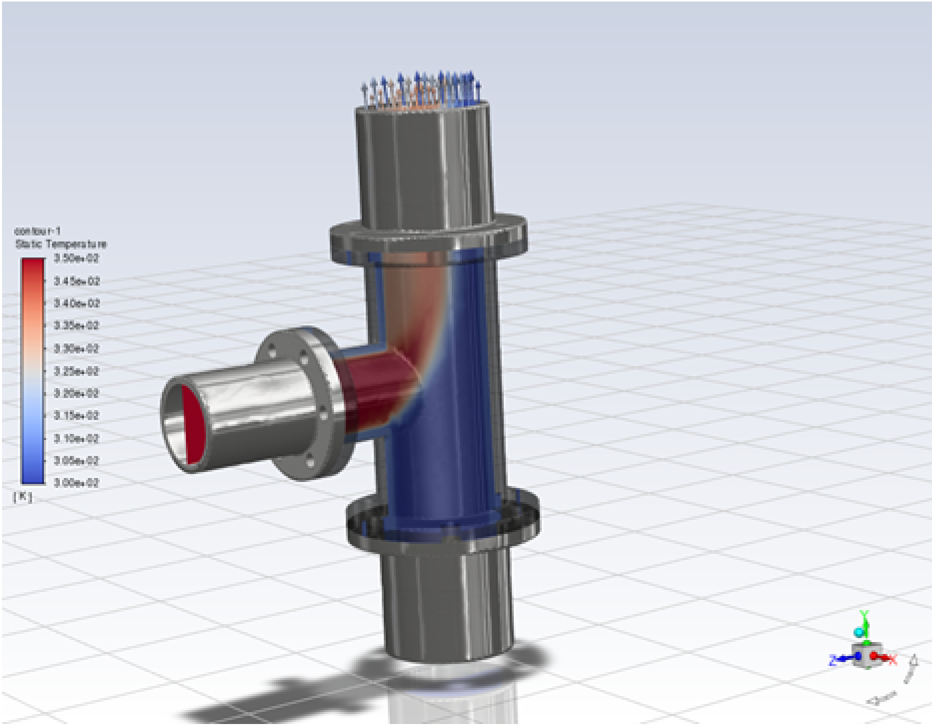
- Hovering over postprocessing graphics objects (such as contours and pathlines) with the left-mouse-button set to probe ( ) now displays the value of the postprocessing variable at that location. See Figure 2.2: Results Displayed on Hover (p. 10) for an example and refer to Pointer Tools in the *Fluent User's Guide* for additional information.

Figure 2.2: Results Displayed on Hover



- Batch renaming of face zones in bulk is now much easier and faster due to a revamp of the renaming option available in the **Adjacency** dialog box. Further, there is a new option for adding a suffix or prefix to the selected face zone(s). Refer to Renaming Zones Using the Adjacency Dialog Box in the *Fluent User's Guide* for additional information.

- You can render your model withseveral new realistic material options such as chrome, rubber, and glass to make your model appear more life-like. Refer to Realistic Rendering of Materials in the *Fluent User's Guide* for additional information on using this feature.



Solver-Meshing

- When using overset meshes in a transient simulation with the pressure-based solver, the Non-Iterative Time Advancement (NITA) option is now supported, along with the hybrid NITA option (which is available for cases that involve the Volume of Fluid model) and the Fractional Step method (FSM). For details about overset meshes, NITA, the hybrid NITA option, and FSM, see Overset Meshes, Setting Solution Controls for the Non-Iterative Solver, Hybrid NITA for the VOF Model, and Fractional Step Method, respectively.

- When using overset meshes, "partial cut faces" are now used by default; that is, an enhanced hole cutting method is used where cut faces partially overlap. This can help avoid the situation where a region or cell zone is erroneously identified as dead because of leakage during flood filling, even though it has overlapping boundaries that match well. You can disable this enhanced method using the following text command: `define/overset-interfaces/options/partial-cut-faces? no`. (Leakage Between Overlapping Boundaries)

- The creation of mesh interfaces with the one-to-one interface method is improved:

  - The time needed to create one-to-one interfaces is reduced, as this process is now faster; this improvement is most noticeable for cases with a large number of mesh interfaces.

  - The management of one-to-one interfaces from the outline view tree is now easier, as by default the intersected zones (which represent the overlapping region of the mesh interface) and the non-overlapping walls are now listed under the associated mesh interfaces (under **Setup** / **Mesh Interfaces**), and these zones can now be edited and displayed using the right-click context menu.

  For more details, see Using a Non-Conformal Mesh in Ansys Fluent.

- Sliding mesh performance is enhanced with optimization of the mesh update. The actual improvement is case-dependent butmay be up to 25%. For information about sliding meshes, see Modeling Flows Using Sliding and Dynamic Meshes.

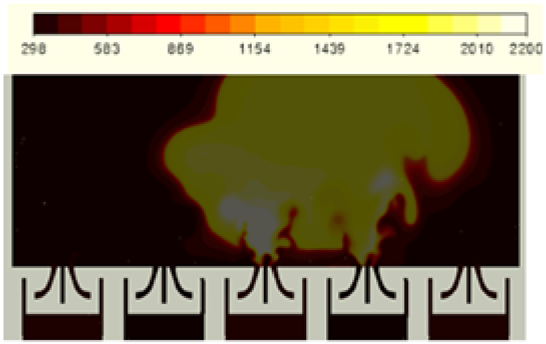
- The **Moving Mesh Courant Number** field variable can now be used in postprocessing of any transient simulations that involve moving or deforming meshes. In previous releases, this variable was available only for postprocessing of VOF analysis.

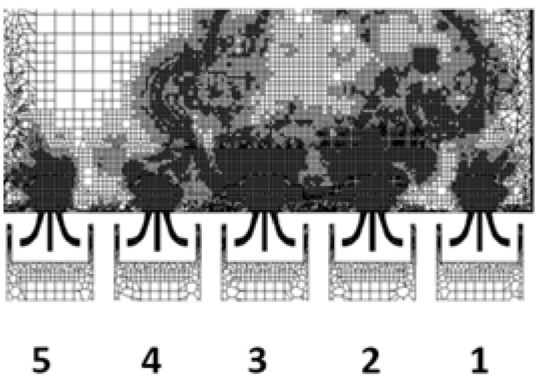
- When using dynamic meshes, overset meshes, and/or sliding meshes to simulate flow through narrow gaps that open or close over time (such as in a valve), a gap model is now available that can simulate blockage of the flow when the gaps are closed. This model marks interior cells in regions where two or more selected face zones are within a defined proximity threshold of each other, and then applies a zero-mass-flux boundary condition at the interface between marked and unmarked interior cells. This model allows you to create multiple gap regions with a unique proximity threshold value for each gap. (Controlling Flow in Narrow Gaps and Valves)

- The unified remeshing method is improved for dynamic mesh simulations, as the setup in the **Mesh Method Settings** dialog box is simplified, and by default an option is enabled that better retains the initial mesh size distribution even as the mesh moves. To revert to the algorithm used in the previous release, disable the `define/dynamic-mesh/controls/remeshing-parameters/retain-size-distribution?` text command and then specify the minimum and maximum length scales using the `define/dynamic-mesh/controls/remeshing-parameters/length-min` and `define/dynamic-mesh/controls/remeshing-parameters/length-max` text commands, respectively. (Unified Remeshing)

- Mesh adaption is improved with several advancements to improve both usability and flexibility:

  – New predefined criteria options are now available for simulations that use a combustion model (so that the mesh is refined along a progressing flame front, with or without an initial spark region) or for aerodynamic simulations with shocks that use the density-based solver or the pressure-based solver with a fluid that uses a real-gas or ideal-gas model for the density. These automatically configure suitable cell registers, expressions, and settings with minimal input. By optimizing the mesh during simulation, computation time can be reduced dramatically, in some cases by a factor of 5 or more. See Combustion Adaption and Aerodynamics Adaption, respectively.





- For automatic adaption (formerly referred to as dynamic adaption), you can now create named adaption criteria that have unique frequency settings and refinement and coarsening criteria. Such adaption criteria provide you greater control over the mesh adaption. They can be set up and managed from the ribbon or the outline view tree, and can be disabled and enabled as needed. (Refining and Coarsening)

- For the PUMA adaption method, you can now specify an approximate maximum cell count so that Fluent will stop marking cells for refinement. (Refining and Coarsening)

- PUMA-based anisotropic adaption can now be set up through the **General Adaption Controls** dialog box, rather than just through the `mesh/adapt/set/anisotropic-adaption?` text command. (Refining and Coarsening)

Cell Zones and Boundary Conditions

- You can now specify a partial-slip wall boundary condition which more accurately models flows of rarefied gases where the amount of friction is insufficient to use the typical no-slip wall. (Partial Slip for Rarefied Gases)

- Fluent is now significantly faster when deleting one or more cell zones. The process of using the **Delete Cell Zones** dialog box is also simplified, as you are now prompted to confirm all of your selections at once rather than repeatedly for each individual cell zone. For information about deleting cell zones, see Deleting Zones in the *Fluent User's Guide*.

- Fluent is now significantly faster when activating one or more cell zones. The speed will increase more when the cell zones have many boundary faces (particularly coupled walls), and may be

3-5 times faster. For information about activating cell zones, see Activating Zones in the *Fluent User's Guide*.

- When setting up the impedance boundary condition, the following new features are available for converting experimental impedance data into the settings needed for the **Impedance Parameters** group box of a boundary condition dialog box:

  - You can now use the graphical user interface within a Fluent session to calculate the impedance parameters; previously this was only possible through the text user interface and the `impedance` utility.

  - It is no longer necessary to manually enter the calculated values in the **Impedance Parameters** group box of a boundary condition dialog box, as it is now possible to apply the values after the calculation or import them from a pole / residue file.

  - The experimental data no longer has to be specific impedance data, as you can now instead use reflection coefficient data or absorption coefficient data.

  - It is now possible to define a residue tolerance for the calculation, which specifies the magnitude of residues that are kept in the fitting. This residue check helps to eliminate parasitic poles.

  - You can now request a higher verbosity, so that messages are printed in the console as the fitting calculation progresses.

  For details about these features, see Using the Impedance Boundary Condition and Calculating Impedance Parameters.

- For perforated walls boundary conditions that involve discrete injections with the dynamic injection method, you can now define your own formulation of the discharge coefficient using a UDF. (`DEFINE_PERFORATED_CD`)

- For perforated walls boundary conditions, you can now use the static injection method to directly specify the injection mass flow rate at the injection hole exit.

- The density-based solver now features a tangency-correction for the pressure far-field boundary condition. This correction is designed to regain robustness on boundary faces tangent to the imposed flow direction and is active by default. (Tangency Correction)

Materials

- The Granta MDS database for solid material properties, that is accessible in Fluent from the **Create/Edit Materials** dialog box, has been updated to the latest version.

Heat Transfer/Radiation

- Periodic boundary conditions are now supported by the Discrete Ordinates (DO) GPU-accelerated solver.

- The two-temperature model for hypersonic flows now uses a NASA-9-piecewise-polynomial function by default to define the specific heat of the flow. (Inputs for NASA-9-Piecewise-Polynomial Functions)

- The **Loosely Coupled Conjugate Heat Transfer** option is improved:

- It is no longer as limited with regard to non-conformal mesh interfaces. Previously, it was not available when the case included a many-to-many mesh interface, unless the mesh interface used the **Matching** option; now it is available for all many-to-many mesh interface option types except the **Mapped** option.
- The solution results are now more accurate with regard to time.
- It is no longer necessary to begin by running a few time steps with just the **Specify Solid Time Step Size** option enabled, and only then continuing with the **Loosely Coupled Conjugate Heat Transfer** option enabled. Now you can enable the **Loosely Coupled Conjugate Heat Transfer** option from the start and still ensure accurate results for the temperature in the solid zones. When doing this, note that it is a good practice is to use a high coupling frequency at the beginning and then to gradually decrease the coupling frequency when the temperature of the solid zones starts to reach its equilibrium state; this will help in the rapid and smooth convergence of the solid temperature without compromising the performance gain of using this option.

For details about using the **Loosely Coupled Conjugate Heat Transfer** option, see Loosely Coupled Conjugate Heat Transfer.

Acoustics

- You can now launch VRXPERIENCE Sound - Analysis and Specification from within the **VRX Sound Analysis** dialog box.

Turbulence

- The Algebraic Transition Model is now available in combination with various turbulence models. It is a simplification of the Intermittency Transition Model that avoids using the transport equation and formulates the intermittency as an algebraic equation. (Algebraic Transition Model)
- For the combination of Realizable k-ε turbulence model with Stress-Blended Eddy Simulation (SBES), the enhanced wall treatment (EWT) is now a released option for near wall treatment.

Turbomachinery

- You can now define mode shape components for periodic displacement (part of aerodamping analysis) using UDFs and expressions. (Defining the Periodic Displacement of the Blades)
- The wet steam model is now compatible with General Turbo Interfaces (GTI) that are used to simulate flows in rotating and stationary components of turbomachines, in particular steam turbines. Details of different GTI types are described in Blade Row Interaction Modeling.
- Postprocessing of blade flutter simulations has been expanded to include visualizing Fourier coefficients of pressure, velocity components, and temperature as well as exporting real and imaginary harmonic pressure loads on surfaces.Visualizing and Exporting Blade Flutter Harmonics

Reacting Flows

- For Flamelet Generated Manifold (FGM) model, the **Strained Laminar Flame Speed** option is now available. This option provides an improved response of the strain in the flow on flame characteristics. (Strained Laminar Flame Speed)

- For the **Laminar Flame Speed** material property, a new method, called **laminar-flame-speed-computed**, is now available. This is a generalized flame speed method that can be used for accurate modeling of the flame speed for hydrogen blended fuels. (Laminar Flame Speed)
- For partially-premixed combustion cases with FGM, you can now edit the progress variable definition. (Flamelet Generated Manifold)
- Surface reactions are now supported with the density-based solver.
- You can now use the ablation condition at walls to model aerodynamics, mass transfer, and heat transfer during the ablative process employed in Thermal Protection Systems (TPS) for high-speed applications, such as re-entry vehicles. (The Ablation Condition at Wall Boundaries)
- In the Species Model dialog box, the **None - Explicit Source** chemistry solver option has been renamed to **None - Direct Source** for clarity.

Pollutant Formation

- The following models have been deprecated:
  - SOx pollutant model

    Instead you are encouraged to use species transport with finite rate chemistry to model SOx.
  - Extended Coherent Flame Model (ECFM)

Discrete Phase Model

- For surface injections, the **Randomized Injection Starting Points** option is now fully supported. With this option, an arbitrary number of particles can be injected from positions distributed randomly over boundary surfaces. This was available as a beta feature in previous releases. (Defining Injection Properties)
- The Lagrangian wall film model can now be used with overset meshes if the film zone is entirely contained within a single overset mesh. (Limitations on Using the Lagrangian Wall Film Model)
- You can now define injection point properties in a local reference frame. This makes it easier to configure multiple similar injectors arranged in different positions/orientations. (Defining Injection Properties)
- The Huilin-Gidaspow, Gibilaro, EMMS, and Filtered drag correlations, that were previously available only with the Eulerian multiphase model, are now made available also for the Dense Discrete Phase Model (DDPM).
- The calibratable temperature option (which was previously available as a beta feature) can now be used to calculate critical transition temperature when modeling drop/wall impingement. Using this option may improve the predictions for Selective Catalytic Reduction (SCR) systems. (The **wall-film** Boundary Condition)
- For non- or partially-premixed combustion cases, the DPM source terms for mixture fractions and inert species (if enabled) can now be linearized with respect to the cell variable. This was available as a beta feature in the previous release. (Linearized Source Terms)

- The **not-vaporizing** evaporating species option (available in the **Set Injection Properties** dialog box, **Components** tab) has been renamed to **immiscible-not-vaporizing** for clarity.
- For 3D cases with unsteady particle tracking, the volume injection type is now available. This was a beta feature in previous releases. In addition to volume zones, you can now define a volume in which particle injections occur using a bounding geometry such as sphere, cylinder, cone, and hexahedron. Note that for DEM cases, the volume injection type still remains beta. (Point Properties for Volume Injections)

Multiphase Models

- For the mixture and Eulerian multiphase models, you can now use the wall drag enhancement for improved predictions of near-wall velocity. (Near-Wall Drag Enhancement)
- For VOF cases, a text interface option has been added to control anti-diffusion strength. This option can be used to minimize artificial waves due to excessive sharpening when using the anti-diffusion treatment. (Interface Modeling Type)
- For VOF cases that involve Hybrid NITA, a new text option for choosing either Global or Interfacial advective CFL type has been introduced for detecting an unstable event when using the instability detector. (Hybrid NITA for the VOF Model)
- For multiphase species transport cases, the following field variables are now available for postprocessing (in the **Species...** category):
  - **Mass Concentration of *`species-n`* (Phase Level)**
  - **Mass Concentration of *`species-n`* (Mixture Level)**

  See Field Function Definitions in the *Fluent User's Guide* for the definitions.

  The volume integral of **Mass Concentration of *`species-n`* (Mixture Level)** can be used for reporting mass of a phase component.
- For Eulerian multiphase species transport cases, the stiff chemistry solver can now be used for solving intraphase (homogenous) reactions. In-Situ Adaptive Tabulation (ISAT) can be used to accelerate detailed stiff chemistry integration in such simulations. (The Stiff Chemistry Solver)
- When using the Wet Steam model, the equation of state for steam now uses the formulation developed by Vukalovich by default. This formulation has shown to be more accurate in the supercooled vapor region compared to the previous default (Young's formulation). For details, see (Built-in Thermodynamic Wet Steam Properties).
- For Eulerian multiphase simulations, you can now model polydisperse boiling. (Setting Up Polydisperse Boiling)
- Fluent is now significantly faster when reading a case file that both uses a multiphase model and has many cell zones and/or boundary zones. The speed improvement increases as the number of zones and/or phases increases, and can be up to three times faster than the previous release. Note that this applies to case files in the Common Fluids Format (`.cas.h5`) as well as the legacy format (`.cas`).
- For the DPM-to-VOF multiphase model transition mechanism, a separate VOF volume fraction threshold is now available for Lagrangian wall film particles. This helps to avoid the coexistence

of the VOF liquid (at volume fractions less than unity) and the Lagrangian wall film, which is often undesirable. (Setting up the DPM-to-VOF Model Transition)

Eulerian Wall Film Model

- The Eulerian Wall Film model is now compatible with the cell/face separation and cell/face merge operations. (Overview of Using the Eulerian Wall Film Model)
- The High Order Explicit time discretization scheme is now available. This option improves the accuracy of simulations that involve the Eulerian wall film. (Setting Eulerian Wall Film Solution Controls)

Battery Model

- For the dual-potential MSMD-based battery model, to simplify customization of battery model parameters, the following user-defined functions (UDF) have been added:

  - `DEFINE_BATTERY_CLUSTER`: To specify the target variable of cell clustering
  - `DEFINE_BATTERY_PROPERTY`: To specify material properties used in the Newman's P2D model

  and the following UDFs have been removed:

  - `DEFINE_BATTERY_NEWMAN_OCP`
  - `DEFINE_BATTERY_NEWMAN_PROP_ELECTRODE`
  - `DEFINE_BATTERY_NEWMAN_PROP_ELECTROLYTE`

  For more information about the new UDFs, see `DEFINE_BATTERY_CLUSTER` and `DEFINE_BATTERY_PROPERTY`.

- For the Four-Equation Kinetics thermal abuse model, you can now include an additional abuse reaction due to the internal short circuit. (Thermal Abuse Model)
- For the Equivalent Circuit model, Newman P2D model and User-defined E-model, you can use the Dynamic method for clustering cells for the voltage, temperature, or user-defined target variable. (Specifying Battery Model Options)
- For the Newman’s P2D model, the following input parameters have been added:

  - **OCP**: The open circuit potential of the electrode material
  - **Ionic Cond**: The ionic conductivity

  See Inputs for the Newman’s P2D Model in the *Fluent User's Guide* for their definitions.

  For simplicity and clarity, the labels for the following Newman P2D model parameters have been changed:

Old Name	New Name
Max. Solid Li+ Conc.	Cs,max
Init. Solid Li+Conc.	Init. Cs

Init. Electrolyte Li+Conc.	Init. Ce
Volume Fraction of Electrolyte	VOF of Electrolyte
Ref. Diffusivity	Ds
Bruggeman Tortuosity exponent	Brugg
Electrolyte Diffusivity	De

- For the Newman's P2D model, the Ansys Fluent Echem material database is now available. The database contains commonly-used anode, cathode, and electrolyte materials. The database can be used when specifying model parameters related to material properties. (Inputs for the Newman's P2D Model)
- For the Four-Equation Kinetics thermal abuse model, the Ansys Fluent database of the kinetics data of the abuse reactions is now available. The database can be used to assist in the reaction parameter specification. (Specifying Advanced Options)
- For the Newman's P2D model, the following postprocessing variables have been added (under the **Battery Variables** category):
  - **Cell Cluster ID**
  - **Newman P2D: Average ce in Cathode**
  - **Newman P2D: Average ce in Separator**
  - **Newman P2D: Average ce in Anode**
  - **Newman P2D: Average cs in Cathode**
  - **Newman P2D: Average cs in Anode**
  - **Newman P2D: Average phis in Cathode**
  - **Newman P2D: Average phis in Anode**
  - **Newman P2D: Average phis in electrolyte**
  - **Newman P2D: Average phi_s-phi_e at anode-separator interface** (Newman's P2D model)
- For the four-equation thermal abuse model, the following postprocessing variables have been added (under the **Battery Variables** category):
  - **Thermal-Abuse Heat Source: SEI Decomposition**
  - **Thermal-Abuse Heat Source: Negative-Solvent**
  - **Thermal-Abuse Heat Source: Positive-Solvent**
  - **Thermal-Abuse Heat Source: Electrolyte Decomposition**
  - **Thermal-Abuse Heat Source: Internal Short** (with the **Enable Internal Short Option** only)

- For the NTGK and ECM battery models, you can now plot model parameters varying with DOD or SOC to confirm their behavior. (Inputs for the NTGK Empirical Model, Inputs for the Equivalent Circuit Model)
- For the NTGK and ECM battery models, you can now plot the resulting fitted curves of the battery discharge or HPPC data after running the parameter estimation tool to verify the fitting process. (Inputs for the NTGK Empirical Model, Inputs for the Equivalent Circuit Model)
- A library of HPPC test data is added in the ECM parameter estimation tool. (The HPPC Library)
- The Single-Potential Empirical battery model has been deprecated.

Polymer Electrolyte Membrane Fuel Cell (PEMFC) Model

- To improve user experience, you can now specify total voltage and total current of a cell or stack directly in the **PEM Fuel Cell Model** dialog box (**Model** tab). **Ansys Fluent** will automatically set up the corresponding boundary conditions internally at the anode and cathode tabs that are now specified in the **Electrical Tabs** tab. (Setting Up the PEMFC Module)
- You can now write data relevant to the fuel cell (cell voltage, mean and minimum values of H_2 mole fractions in anode fluid zones, mean and minimum values of O_2 mole fractions in cathode fluid zones, and total current generated in anode and cathode catalyst zones) to output files. The electrolyte project area is now automatically calculated by **Ansys Fluent**, and you no longer need to specify it. (Reporting on the Solution (**Reports** Tab))
- The inlet condition calculator is now available for calculating inputs for boundary conditions at anode and cathode. (Reporting on the Solution (**Reports** Tab))

Structural Model for Intrinsic Fluid-Structure Interaction (FSI)

- The **Linear Elasticity** structural model now supports the 2D **Axisymmetric** or **Axisymmetric Swirl** option.
- Rayleigh damping for unsteady simulations is now available using the `define/models/structure/controls/unsteady-damping-rayleigh?` text user interface command. See Modeling Fluid-Structure Interaction (FSI) Within Fluent for more information.

Solver

- A new **Local Residual Scaling** option is available in **Preferences** (under the **Simulation** branch) that allows you to use local residual scaling by default for each computation. Refer to Controlling Normalization and Scaling in the *Fluent User's Guide* for additional information on local residual scaling.
- When reporting residuals with local scaling for the pressure-based solver, an enhanced formulation for the continuity residuals is now supported as a full feature and is enabled by default. This formulation provides scaling that uses the absolute mass flow rate at each cell, and can improve the continuity residual results by producing behavior that is more consistent with the other residuals. To disable this enhanced formulation for new case files or to enable it for case files created using a previous release, use the following text command: `solve/monitors/residual/enhanced-continuity-residual?`. For further details, see Definition of Residuals for the Pressure-Based Solver.

- For transient cases with the **Bounded Second Order Implicit** formulation, a text interface option has been added that allows you to use the first order time implicit formulation for species transport, population balance, and energy equations while retaining the second order time formulation for other equations. This option provides a remedy for certain unbounded variables.

- You can now specify **BCD Scheme Boundedness** as an expression when using the Bounded Central Differencing spatial discretization scheme. For more details, see Bounded Central Differencing Scheme in the *Fluent Theory Guide*.

- When using the pressure-based solver for a simulation that does not use a multiphase model (except for the wet steam model), you can now choose how the mass flux is calculated: the high-order velocity interpolation with a Rhie-Chow correction for the pressure gradient difference can now be weighted by either distance (which is the default, and was the only option starting in Release 2019 R3) or momentum (which was the only option prior to Release 2019 R3). You may want to select the momentum-weighted method for simulations that use an incompressible fluid, as it can be more robust; you can enable an option so that this flux type is automatically selected for such cases. (Mass Flux Types)

Density-based Solver

- The linearization of the finite-rate chemistry models featuring backward reactions has been improved, when using the density-based solver. This allows enhanced robustness and faster convergence when using **None - Direct Source** terms in the species equations.

- The two-temperature model for hypersonic flows can now be coupled with the finite-rate chemistry model to more accurately predict thermal-chemical non-equilibrium flows. (Finite-Rate Chemistry with the Two-Temperature Model)

Adjoint Solver Module

- In previous releases, the AMG does not do coarsening during the residual minimization scheme which increases the speed of each sub-iteration and reduces the outer convergence. In this release, during the residual minimization scheme, the AMG does coarsening making the speed of each sub iteration similar to normal iterations, as well as improving the outer convergence.

- The shape sensitivity calculation has been decoupled from the adjoint solver advancement, which requires AMG allocation. This reduces the memory cost of postprocessing and the design tool calculation. This improvement allows for postprocessing and the design tool calculation to be performed on a smaller machine.

- Adjoint solution methods now supports the partial coupling feature which decouples some of the adjoint equations from the AMG and solves them in a segregated manner. Partial coupling significantly improves memory costs and the speed per iteration, although the outer convergence may be slightly reduced.

- Adjoint solution methods now supports discretization options for the energy, k, and omega equations.

Graphics, Reporting, and Postprocessing

- You can now optionally preserve the rendering details of graphics objects (such as camera angle, zoom level, ruler status, and so on) so that they appear the same when you redisplay them in future sessions. These properties are collectively referred to as "display states" and they can also be shared between graphics objects, allowing you to create multiple persistent graphics objects with the same orientation and visual specifics. Refer to Controlling the Display State and Modifying the View in the *Fluent User's Guide* for additional information.

- The default colormap is upgraded to **field-velocity**. You can control the default colormap, as well as other colormap settings under **Colormap Settings** in the **Graphics** branch of **Preferences** (**File**>**Preferences...**).
- The initial display time for postprocessing visualizations is improved for large, complex cases.
- Embedded window layouts may now be saved with the case file, allowing them to be automatically recreated in for future sessions. Refer to Embedded Graphics Window Dashboards in the *Fluent User's Guide* for additional information.
- New colormaps are available for displaying simulation results. Some of the new colormaps include symmetric coloring, value coloring, and highlighting. Refer to Changing the Colormap in the *Fluent User's Guide* for additional information and a full listing of the predefined colormaps in Ansys Fluent.
- Cumulative force/moment plots now provide an option to use averaged force statistics as an alternative to the default instantaneous values. Cumulative plots are also now defined as persistent objects, so you can include multiple plots for different forces/moments. Refer to Cumulative Force, Moment, and Coefficients Plots in the Cumulative Force, Moment, and Coefficients Plots in the *Fluent User's Guide* for additional information on creating cumulative plots.
- Multiple lighting default changes combine to improve the appearance of postprocessing graphics objects such as contours and vectors. All of these settings are controlled in **Preferences**, and a summary of the changes is provided below, along with the path to the settings in **Preferences**:
  - Ambient lighting is increased from 0.3 to 0.7. **Preferences**>**Graphics**>**Lighting**>**Ambient light intensity**.
  - Surface specularity (other than for contours) is decreased from 1 to 0.3. **Preferences**>**Appearance**>**Surface specularity for other surfaces**.
  - Contour specularity is decreased from 1 to 0. **Preferences**>**Appearance**>**Surface specularity for Contours**.
- A new preference is available for controlling the intensity of the ambient lighting in the graphics window. **Ambient light intensity** is listed under **Lighting** in the **Graphics** branch of **Preferences.** *Ambient lighting is automatically increased when you switch to* ***Pastel colors*** *for the* ***Model Color Scheme*** *(****Appearance*** *branch of* ***Preferences****).*
- Pictures are now saved in PNG format by default. You can control the default format under **Save Picture Settings** in the **Graphics** branch of **Preferences** (**File**>**Preferences...**).
- A new **Local Residual Scaling** option is available in **Preferences** (under the **Simulation** branch) that allows you to use local residual scaling by default for each computation. Refer to Controlling Normalization and Scaling in the *Fluent User's Guide* for additional information on local residual scaling.

- The **Average Over** value for report definitions can now be changed after the conclusion of a simulation, providing you the flexibility to update an associated plot with the best averaging interval for your case. Refer to Moving Average Monitors in the *Fluent User's Guide* for additional information.
- Report plots now provide the option of plotting the instantaneous values of a report definition along with the averaged values.
- Cumulative plots are now created as reusable plot objects that are saved with the case. Refer to Cumulative Force, Moment, and Coefficients Plots in the *Fluent User's Guide* for additional information.
- Flux report definitions computed/printed to the **Console** now show the contributions of sources in the computation (when applicable).

Parallel Processing

- For improved computational efficiency, automatic repartitioning occurs for any imbalanced Fluent (tetrahedral and prism) mesh when it is read into the Fluent solver. This happens regardless of any differences in the parallel node distribution of the mesh. In other words, upon being read into the Fluent parallel solver, an imbalanced parallel tetrahedral or prism mesh created in the Fluent mesher will be automatically repartitioned.

Simulation Reports

- If you use any named expressions as part of your boundary condition definitions, the named expressions are displayed in the Boundary Condition table as hyperlinks that take you to the Named Expressions section of the simulation report, and vice versa. For more information about simulation reports, see Using Simulation Reports in the *Fluent User's Guide*.
- You are now able to save a copy of your plots (residuals, report definition, etc.) as an PNG image file. For more information, see Using Simulation Reports in the *Fluent User's Guide*.
- You are able to arrange multiple result plots in a tabular style, rather than as multiple tabs in your simulation report. Such arrangements can be made to individual categories of result plots, or as a global setting for all plots in your reports. For more information, see Organizing Your Simulation Report in the *Fluent User's Guide*.
- You are able to customize and re-order your simulation report contents by dragging and dropping items in the report tree. For more information, see Organizing Your Simulation Report in the *Fluent User's Guide*.

Expressions

- You can now define expressions for fluxes, including: mass flow rate, total heat transfer rate, radiation heat transfer rate, film mass flow rate, film heat transfer rate, and sensible heat transfer rate. Refer to Fluent Expressions Language in the *Fluent User's Guide* for additional information on expressions.
- Statistics for named expressions that compute to a field can now be computed by selecting them in the **Zone-Specific Sampling Options** dialog box. Refer to User Inputs for Time-Dependent Problems and Data Sampling for Steady Statistics in the *Fluent User's Guide* for more information on computing statistics.

- The list of supported variables is expanded to include those that are available on nodes, which primarily add support for variables under the **Mesh...** and **Structure...** categories. Refer to Appendix: Supported Field Variables in the *Fluent User's Guide* for a full listing of supported variables.

Load Managers

- When running Fluent on Linux with a load manager / job scheduler, new options are now available from the command line that allow you to:
  - specify the scheduler job submission machine name
  - specify the name / directory of the scheduler standard error file
  - specify the name / directory of the scheduler standard output file
  - ensure that your environment variables related to the job scheduler are used when using custom scheduler scripts instead of relying on the standard Fluent option (either the `-scheduler=<scheduler>` option from the command line or the **Use LSF**, **Use SGE**, or **Use PBSPro** option in Fluent Launcher)

  For further details, see Scheduler Options in the *Fluent User's Guide* and Running Fluent Using a Load Manager.

- When using Linux, Fluent can now be run under Slurm, an open-source workload management tool for local and distributed environments. For further details, see Scheduler Options in the *Fluent User's Guide* and Running Fluent Using a Load Manager.

Solid Oxide Fuel Cell (SOFC) With Unresolved Electrolyte Model

- To simplify the SOFC model setup, Ansys Fluent now automatically applies a number of recommended settings after you initially enable the SOFC model. These settings can be adjusted as needed. See Solid Oxide Fuel Cell With Unresolved Electrolyte Module Set Up Procedure in the *Fluent User's Guide* for more information.
- The SOFC Module can now be used to model SOFC Electrolysis. (Modeling Electrolysis)

Fluent in Workbench

- Fluent with Fluent Meshing is now available as a component system within Ansys Workbench. Refer to Fluid Flow (Fluent with Fluent Meshing) in the *Workbench User's Guide* for additional information.

Beta Features

- There are also some exciting new enhancements available as beta features that you may be interested in trying out. Detailed documentation is in the *Fluent 2021 R2 Beta Features Manual*.

2.2.3. Fluent Applications

New features available in the client applications of Ansys Fluent 2021 R2 are listed below.

Fluent Remote Visualization Client

- place holder

Fluent Icing

Fluent Icing allows users to easily conduct in-flight icing simulations within a dedicated Fluent Application Client environment. The following additional functionality has been added in this release.

- **Improvements to the user interface & Project Management Structure**
  - New application layouts have been introduced to Fluent Icing. All layouts are available in the layout menu located at the top right corner of the user interface. The selected layout is automatically saved and used the next time the user opens the user interface. A **Custom** layout has been added in order to enhance user experience.
  - The ribbon menu has been reorganized to simplify the set-up of projects, simulations, and workspaces.
  - Contextual help is available for most settings by clicking the question mark symbol (?) located at the right side of a parameter in the **Properties** panel.
  - Runs can be grouped into subfolders within its parent **Simulation** folder. Runs can be renamed and deleted from the **Project View** panel.
  - Result or solution files located in the **Project View** panel are now displayed using a naming system that is more similar to its simulation type (**Airflow**, **Particle**, **Ice**, **Case**). These files are now grouped into numbered sub-folders when performing multishot icing or mesh adaptation simulations that require several steps or cycles. Additionally, you can launch, from these files, a post-processor to analyze its content or access the convergence history and its run properties. Result files are now saved with the same filename root than the current case file.
  - A scriptable python API for project management and runs is now available.
- **Improvements to Post-processing Results**
  - The integrated post-processing tool now supports cutting plane surfaces and scenes.
  - EnSight can be also used as a post-processor for airflow, particles and icing solution files.
  - The quick-view commands in the **Results** ribbon will directly load the solution from the **Project View** into the post-processor (Viewmerical, CFD-Post, EnSight).
  - The Ansys license is shared between the Fluent solver and the first post-processing tool launched from Fluent Icing, consuming a single license seat.
- **Multishot Simulations**
  - In-flight icing conditions and convergence settings can be modified at each shot. (Beta)
  - Restart capabilities have been added to resume calculations at the beginning of a simulation type (**Airflow**, **Particles**, **Ice**, **Case**) of a given shot. (Beta)

- Remeshing parameters, that are used to rebuild the new 3D domain at each shot, have been added to the **Properties** panel of **Ice** under **Solution**. This will ease the set-up of different type of icing simulations instead of editing a separate file. (Beta)

- **Mesh Adaptation Cycles**
  - OptiGrid mesh adaptation capabilities have been more intimately integrated within Fluent as libraries. This enabled the simulation of efficient multicore simulations on clusters. (Beta)

Fluent Aero (Beta)

Fluent Aero allows you to easily explore the aerodynamic performance of aircraft under a wide range of flight regimes, from subsonic to hypersonic conditions, all within a dedicated Fluent Application Client environment. A streamlined workflow guides you through the creation of a matrix of flight conditions or design points where single and multiple flight parameters, such as AoA, Mach number, altitude, etc., can vary. Most common models, solvers and convergence settings of Fluent are tuned using the latest best practices for external aerodynamic problems and are available in Fluent Aero's user interface. In this manner, simulations can be conducted in a quicker and more user-friendly environment. The full capabilities of the Fluent Solution Workspace remain accessible when its session is displayed through the Fluent Aero ribbon **Workspace** → **Solution** button. Beta tutorials are available to provide examples on how to conduct exploratory simulations using single and multiple flight parameters.

A broad range of features have been added for the 2021R2 beta release.

- The workflow steps have been substantially reorganized and redesigned for 2021R2 to provide a more flexible and user friendly setup environment.
- A new **Component Groups** step has been added to allow you to group aircraft components into engine, wing or other aircraft component groups. This grouping facilitates the setup of component specific boundary conditions and enhances post-processing and analyses.
- Support for hypersonic simulations has been improved, including the support for two temperature model and new air properties.
- Two domain types are now supported: **Freestream** and **WindTunnel**.
- Additional inputs parameters have been added: **True Airspeed**, **Sideslip Angle**, **Total Pressure** and **Temperature**, and **Reynolds number**.
- Additional output parameters have been added: **Pitching**, **Yawing** and **Rolling Moments**.
- Experimental data can now be loaded into **Graphs** and **Plots** to ease validation of CFD results.
- Fluent Journal files can now be used to complement Fluent Aero's simulation setup at various stages of your calculation.
- Additional **Initialization** control is available. This allows you to initialize a single or a group of design points without calculating the flow solution at each design point. This can be used to improve not only the quality of the initial solution but also the convergence of the flow solution at each design point.

2.3. Updates Affecting Ansys Fluent 2021 R2 Code Behavior

The following sections list the code changes in Ansys Fluent 2021 R2.

General

2.3.1. Meshing Mode

This section contains a list of code changes implemented in the meshing mode of Ansys Fluent 2021 R2 that may cause behavior and/or output that is different from the previous release.

Mesh Generation

- For workflows that contain fluid region capping, the default global maximum will be reduced in value from previous versions, leading to higher cell counts.

- When using the meshing workflows, after generating the volume mesh, the default quality measure has been changed from Inverse Orthogonal Quality to a more improved Orthogonal Quality measure.

Watertight Geometry Workflow

- Automatic assignment of fluids and solids has been improved in this release, resulting in fewer "dead" regions which previously needed corrections in the **Update Boundary** task. Journals or templates that include these corrections may result in errors if these corrections are not removed and/or modified.

Fault-tolerant Meshing Workflow

- In the **Generate the Volume Mesh** task, the **Volume Mesh Target Skewness** setting has been replaced with **Quality Improve Limit** which uses the new orthogonal mesh quality measure, rather than the skewness quality measure.

- Improvements have been made to the **Add Local Sizing** task, where the **boi** and **soft** size function types are no longer scaled based on the **Wrap/Target Size Control Ratio** (only the **proximity** and **curvature** types are scaled), making them more consistent with the default options in the **Choose Mesh Control Options** task.

2.3.2. Solution Mode

This section contains a list of code changes implemented in the solution mode of Ansys Fluent 2021 R2 that may cause behavior and/or results that are different from the previous release.

Files

- When reading mesh files using a ribbon tab item, mesh files in the Common Fluids Format (CFF)—that is, files with the `.msh.h5` extension—are now prioritized by default over the legacy `.msh` files:

  – When using the **File/Read/Mesh...** ribbon tab item, **CFF Mesh Files (*.msh.h5)** is now selected by default from the **Files of type** drop-down list in the **Select File** dialog box, and so `.msh` files will not be displayed for selection.

  – When using the **File/Read/Mesh...** or **File/Read/Case...** ribbon tab items, if you enter a name without an extension in the **Mesh File** or **Case File** field of the **Select File** dialog box, a file will now be read (in previous releases you had to include an extension to read a mesh files) and CFF files are prioritized. For example, if both `myfile.msh` and `myfile.msh.h5` exist in the same directory and you enter `myfile` in the **Mesh File** or **Case File** field, you will read `myfile.msh.h5`.

  To change the behavior:

  – You can select **All Files (*)** from the **Files of type** drop-down list to view legacy files in the current **Select File** dialog box.

  – To have all mesh / case files selected by default from the **Files of type** drop-down list in the **Select File** dialog box and/or have `.msh` files take priority when reading a mesh file without an extension, enter `file/cff-file? no` in the console, or (to continue to use this setting in future sessions) select **Legacy** from the **Default Format for I/O** drop-down list in the **Preferences** dialog box (under **General**); note that either of these actions will also result in you reading and writing legacy case and data files.

- Values at boundary surfaces exported to ASCII format with **Cell Values** enabled are now taken from face centers rather than from cell centers, which brings the export in line with the intended and documented behavior.

Solver-Meshing

- When adapting the mesh using one of the predefined criteria for the Volume of Fluid (VOF) multiphase model, note the following:

  – The refinement criteria for all VOF predefined criteria is now improved to account for cell volume and/or cell refinement level.

  – The settings that you must define as part of the **VOF-to-DPM [Advanced]** criterion have changed in order to increase the ease of setup.

Cell Zones and Boundary Conditions

- When reading a case created using a previous release, all solid zones in which the only defined motion is frame motion will be converted to instead use solid motion, with the same motion variable values for origin, axis, and velocities. Solid motion is generally preferred, as it avoids unsupported applications (for example, frame motion is not supported for a solid zone that is moving relative to an adjacent solid zone). Any solid zone that combines frame motion with solid motion and/or mesh motion will remain unchanged. To revert this change, use the `define/boundary-condition/set/solid` text command to revise the settings for one or more solid zones (that is, set `solid-motion?` to `no` and set `mrf-motion?` to `yes`). For further details about motion options for solid zones, see Defining Zone Motion.

Discrete Phase Model

- The high resolution tracking is now enabled by default, which may result in more robust performance.
- For the Langrangian wall film model, an improved film boiling model is now used for the multicomponent film particles. This may result in changes to all Lagrangian wall film variables; specifically, the predictions for **Wall Film Temperature** and **Film Mass Fraction** of liquid components are now expected to be more accurate. The previous behavior can be restored by issuing the below scheme command sequence in the Ansys Fluent console:

```
(rpsetvar ‘dpm/mc-film-boiling-formulation 2)
(dpm-parameters-changed)
```

Multiphase Models

- For VOF cases that involve Hybrid NITA, the default for **Maximum CFL** has been changed from 10 to 25 to enhance the performance of simulations with the **Instability Detector** option. Existing case file settings will be respected.

Solver

- A gradient computation enhancement for interior mesh interfaces is now enabled by default; this helps resolve convergence issues and may improve accuracy in cases that have interior type mesh interfaces (such as fluid-fluid interfaces) with poor mesh quality. If you suspect that an individual case is being negatively impacted by this enhancement, you can undo it by using the following text command: `solve/set/previous-defaults/undo-2021r2-default-changes?`.
- For all case files that were created using a previous release and use the pressure-based solver, the correction form of the discretized momentum equation is now enabled when the case file is read, which may result in faster calculations when using the double-precision solver. For details on other possible benefits, see Using the Correction Form of Momentum Discretization in the *Fluent User's Guide*. You can disable it by using the following text command: `solve/set/advanced/correction-form`.
- Transient simulations with adaptive time stepping can now recalculate the time step size during the solution if the mesh changes as a result of automatic adaption (formerly referred to as dynamic adaption). When the **CFL-Based** or **Multiphase-Specific** method is used, this may result in better time accuracy. When the **Error-Based** method is used or if any adaptive method is used in a case with a dynamic mesh and/or sliding mesh, the solutions will not change but the recalculated time step size will be noted in the console as a warning; if there is a large discrepancy with the time step size that is used, you should consider revising the setup to ensure time accuracy. Note that the time step size is not recalculated if the **Extrapolate Variables** option is enabled in the **Run Calculation** task page and/or if the case combines an overset mesh with the Volume of Fluid (VOF) multiphase model.

Turbulence

- The Intermittency Transition model (γ model) has been revised to use a harmonic averaged value of the turbulent kinetic energy for the calculation of the turbulence intensity, whereas a local value was used in previous Fluent versions. This helps to overcome hysteresis effects (dependence of transition location on initial condition) observed mainly in applications with low turbulence intensity and natural transition. It should have a small effect on flows with separation induced transition or bypass transition. For the majority of transition cases, the impact on the transition location should be small. For consistency reasons, the harmonic averaging is also used in the Algebraic Transition model (Alg-γ), which is newly released in Ansys 2021 R2. This model used an upstream value in previous Fluent versions (as a beta feature), which already helped to avoid hysteresis effects.

  The modification can be reverted using the TUI command `/solve/set/previous-defaults/undo-2021r2-default-changes?`. For the Algebraic Transition model (Alg-γ), this also reverts the value of the model coefficient **CTU_LOWTU** from 2.25 to 2.6.

Graphics, Reporting, and Postprocessing

- Computation of transient field statistics has been slightly changed to now compute the statistics at the end of each time-step rather than at the beginning of each timestep. This corrects some defective behavior that could previously occur with:

  - Using the Extrapolate Variables option together with statistics calculation
  - Statistics computation for certain time-derivative quantities (such as dp/dt)

  Changes to reported statistics values in other scenarios should generally be small.

- Several default changes in lighting, as well as the new default colormap, **field-velocity**, may lead to the appearance of changes in results when none are present. You may encounter this issue when comparing images of new graphics objects (contours, vectors, and so on) with results from previous releases. All of these settings are controlled in **Preferences**, and a summary of the changes is provided below, along with the path to the settings in **Preferences**:

  - Ambient lighting is increased from 0.3 to 0.7. **Preferences**>**Graphics**>**Lighting**>**Ambient light intensity**.
  - Surface specularity (other than for contours) is decreased from 1 to 0.3. **Preferences**>**Appearance**>**Surface specularity for other surfaces**.
  - Contour specularity is decreased from 1 to 0. **Preferences**>**Appearance**>**Surface specularity for Contours**.
  - The default colormap changed from **bgr** to **field-velocity**. **Preferences**>**Graphics**>**Colormap Settings**>**Colormap**.

- Transparency specified on a mesh object that is included within a scene will not be applied on any mesh edges.

- Flux report definitions computed/printed to the **Console**, including those being used as output parameters, now show the contributions of sources in the computation (when applicable).

Solid Oxide Fuel Cell (SOFC) With Unresolved Electrolyte Model

- The default settings for the SOFC model have been adjusted to improve robustness and performance and reduce the need for manual tuning. The new defaults should work well for most cases.

Load Managers

- When running Fluent under LSF software, tight integration between LSF and MPI is now enabled by default for Intel MPI (the default) without requiring the inclusion of the `-scheduler_tight_coupling` command line option or the enabling of the **Tight Coupling** option in Fluent Launcher. Note that tight integration is not supported with the `-gui_machine=<hostname>` command line option or if the **First Allocated Node** option is not enabled in Fluent Launcher.

2.3.3. Fluent Applications

This section contains a list of code changes implemented in the client applications of Ansys Fluent 2021 R2 that may cause behavior and/or output that is different from the previous release.

Chapter 3: Resolved Issues and Limitations in Ansys Fluent 2021 R2

For lists of issues that have been resolved in the meshing mode and solution mode of Ansys Fluent 2021 R2, see the Resolved Issues and Limitations document.

Chapter 4: New Limitations in Ansys Fluent 2021 R2

For lists of new or recently discovered limitations known to exist in the meshing mode and solution mode of Ansys Fluent 2021 R2, see the Known Issues and Limitations document.

Chapter 5: Text Command Changes in Ansys Fluent 2021 R2

The following sections list the text command changes in Ansys Fluent 2021 R2.

5.1. Meshing Mode

Changes to each of the text command menus are listed. Note that a modified text command may have new, revised, and/or removed prompts, while a new text command is one that did not exist in previous releases of the meshing mode of Ansys Fluent.

mesh/

prepare-for-solve (Changed)

The `prepare-for-solve` command has been updated, and no longer cleans up face/cell zone names. That functionality is now available using the `zone-names-clean-up` command.

mesh/rapid-octree/

distribute-geometry? (New)

Enables/disables the distribution of the input geometry across partitions / compute nodes, so that it is not copied to each process. This reduces the memory requirements of the mesh generation significantly, especially for geometries with a high number of triangles. Note that this geometric distribution is enabled by default and is automatically deactivated if the geometry is not fully enclosed by the defined bounding box.

mesh/

zone-names-clean-up (New)

Cleans up face and cell zone names by removing excessive characters so that the zone names are easier to read. During the course of creating a mesh, various mesh manipulations can cause the face and cell zone names to become cluttered (with colons, dashes, and numbers for example). Using this command, zone names are reduced to include only the original zone prefix, a single colon, and the zone's ID appended to the end.

5.2. Solution Mode

Changes to each of the text command menus are listed. Note that a modified text command may have new, revised, and/or removed prompts, while a new text command is one that did not exist in previous releases of Fluent.

`define/boundary-conditions/`

`wall` (Changed)

New prompts have been added for setting the calibratable temperature for the wall-film boundary condition. This was available as a beta feature in the previous release. In addition, prompts related to the Solar Load Model no longer appear when the Solar Load Model is disabled. Journal files created prior to Ansys Release 2021 R2 can still be used by setting TUI version to a prior version using the command `file/set-tui-version`. In addition, new prompts for setting the ablation conditions on the wall have been added.

`define/dynamic-mesh/controls/remeshing-parameters/`

`retain-size-distribution?` (New)

Enables/disables the use of local size criteria when marking cells for unified remeshing (in an attempt to maintain the initial mesh size distribution even as the mesh moves), rather than marking cells based on the minimum and maximum length scale values of the cell zone in the initial mesh. Either marking can be overridden if more restrictive values are specified using the `define/dynamic-mesh/controls/remeshing-parameters/length-min` and `define/dynamic-mesh/controls/remeshing-parameters/length-max` text commands.

`define/`

`gap-model/` (New)

Enters the gap model menu, where you can define one or more gap regions where the flow is controlled when face zones move within a specified proximity threshold of each other.

`define/gap-model/`

`advanced-options/` (New)

Enters the advanced options menu for the gap model.

`define/gap-model/`

`create` (New)

Creates a single gap region, so that when selected face zones move within a specified proximity threshold of each other, flow control is applied to the cells that lie within the threshold.

`define/gap-model/`

`delete` (New)

Deletes an existing gap region.

`define/gap-model/`

`delete-all` (New)

Deletes all of the existing gap regions.

`define/gap-model/`

`edit` (New)

Edits an existing gap region.

`define/gap-model/`

`enable?` (New)

Enables/disables the gap model.

`define/gap-model/`

`list-gap-cell-zones` (New)

Lists the names of the cell zones that can be excluded for individual gap regions (so that such cells are not marked for flow blockage).

`define/gap-model/`

`list-gap-face-zones` (New)

Lists the names of the face zones that can be used for creating gap regions.

`define/gap-model/`

`list-gap-regions` (New)

Lists the properties of the gap regions.

`define/materials/`

`change-create` (Changed)

For the laminar flame speed, a new method, `laminar-flame-speed-computed`, has been added. This option provides you accurate laminar flame speed for hydrogen and hydrogen blended fuels.

`define/mesh-interfaces/`

`display` (New)

Displays the specified mesh interface zone.

`define/mesh-interfaces/`

`draw` (Deleted)

This text command is removed and is now replaced with `define/mesh-interfaces/display`.

`define/mesh-interfaces/`

`enhance-gradients-at-interior-interfaces?` (Deleted)

This text command is no longer available, as the enhancement is now enabled by default and can be enabled / disabled by using the following text command: `solve/set/previous-defaults/undo-2021r2-default-changes?`.

`define/models/`

`ablation?` (New)

Enables/disables the ablation model.

`define/models/dpm/injections/injection-properties/set/location/`

`reference-frame` (New)

Sets the reference frame for injection point properties. You can choose either `global` (default) or one of the local reference frames that you created. This option is available only if at least one injection has been selected using `define/injections/properties/set/pick-injections-to-set`.

`define/models/dpm/numerics/high-resolution-tracking/`

`always-use-face-centroid-with-periodics?` (New)

When enabled, Ansys Fluent uses quad face centroids when creating subtets in cases with periodic boundaries.

`define/models/dpm/numerics/high-resolution-tracking/`

`enhanced-cell-relocation-method?` (Deleted)

This command has been removed.

`define/models/dpm/numerics/high-resolution-tracking/`

`sliding-interface-crossover-fraction` (New)

Specifies the fraction of the distance to the subtet center to move the particle.

At non-conformal interfaces, the nodes used for the barycentric interpolation are different on either side of the interface. This may result in incomplete particles due to discontinuities in the variable interpolation. The number of incomplete particles may be reduced by moving the particles slightly off of the sliding interface. Recommended values range between 0 and 0.5.

`define/models/eulerian-wallfilm/`

`solution-options` (Changed)

New prompts have been added for using the high order explicit time discretization scheme and changing the discretization order. The default order is 2. The prompts appear only if the response to the `First Order Explicit Time Discretization?` prompt is **no**.

`define/models/multiphase/population-balance/phenomena/`

`growth` (Changed)

A new method, called `pb-boiling`, has been added for modeling polydisperse boiling.

`define/models/multiphase/population-balance/phenomena/`

`nucleation` (Changed)

A new method, called `pb-boiling`, has been added for modeling polydisperse boiling.

`define/models/multiphase/wet-steam/set/`

`virial-equation` (New)

Sets the equation of state for steam to either Vukalovich formulation (default) or Young formulation.

`define/phases/`

`interaction-domain` (Deleted)

This text command has been removed; its functionality is available through the `define/phases/set-domain-properties/` text menu. Existing script files can still run and do not require modification, though if you are using the dpm-to-vof model transition, there is a new prompt for defining the VOF phase volume fraction threshold for Lagrangian wall film particles.

`define/phases/`

`phase-domain` (Deleted)

This text command has been removed; its functionality is available through the `define/phases/set-domain-properties/` text menu. Existing script files will still run and do not require modification.

`define/phases/set-domain-properties/interaction-domain/forces/`

`drag` (Changed)

New drag factor method, `wall-drag-enhancement`, has been added.

`define/phases/set-domain-properties/interaction-domain/heat-mass-reactions/`

`mass-transfer` (Changed)

For the Zwart-Gerber-Belamri cavitation model, the `Bubble Diameter` prompt has been changed to `Bubble Radius`. The existing script and journal files are respected.

`display/`

`display-states/` (New)

Enter the display states menu.

`display/display-states/`

`apply` (New)

Apply a display state to the active graphics window.

`display/display-states/`

`copy` (New)

Copy the settings of an existing display state to another existing display state.

`display/display-states/`

`create` (New)

Create a new display state.

`display/display-states/`

`delete` (New)

Delete a display state.

`display/display-states/`

`edit` (New)

Edit a display state.

`display/display-states/`

`list` (New)

Print the names of the existing display states to the console.

`display/display-states/`

`read` (New)

Read in display states from a file.

`display/display-states/`

`use-active` (New)

Save the display state settings of the active graphics window to an existing display state. *This command is not available when the active window is displaying a 2D plot.*

`display/display-states/`

`write` (New)

Write one or more of the saved display states to a file.

`display/objects/`

`create` (Changed)

When manually coloring a mesh object, there is an additional option for specifying `material-color`.

`file/`

`table-manager` (New)

Enters the table file manager menu.

`define/models/pemfc/`

`electrical-tabs-setup` (New)

Sets anode and cathode external contacts where electrical potential (UDS0) boundary conditions are applied.

`define/models/pemfc/`

`model-options` (Changed)

New prompts for selecting the electrical boundary type have been added.

`define/models/pemfc/`

`reports` (Changed)

Instead of setting the electrolyte project area and external contacts, this command now allows you to enable writing data relevant to the fuel cell (cell voltage, mean and minimum values of H_2 mole fractions in anode fluid zones, mean and minimum values of O_2 mole fractions in cathode fluid zones, and total current generated in anode and cathode catalyst zones) to output files and set the monitoring frequency.

`define/models/sofc-model/`

`model-parameters` (Changed)

New prompt has been added to set F-cycle (in AMG) for all equations.

`define/models/sofc-model/electric-field-model/`

`conductive-regions` (Changed)

New prompts have been introduced to set the conductive regions and conductivity values.

`define/models/sofc-model/electric-field-model/`

`contact-resistance-regions` (Changed)

New prompts have been introduced to set the contact surfaces and the resistance values.

`mesh/adapt/`

`manage-criteria/` (New)

Enters the manage criteria menu, which provides text commands for managing automatic adaption criteria.

`mesh/adapt/manage-criteria/`

`add` (New)

Adds a new automatic adaption criterion.

`mesh/adapt/manage-criteria/`

`delete` (New)

Deletes an existing automatic adaption criterion.

`mesh/adapt/manage-criteria/`

`edit` (New)

Edits an existing automatic adaption criterion.

`mesh/adapt/manage-criteria/`

`list` (New)

Lists all the existing automatic adaption criteria.

`mesh/adapt/manage-criteria/`

`list-properties` (New)

Lists the properties of an existing automatic adaption criterion.

`mesh/adapt/predefined-criteria/`

`aerodynamics/` (New)

Enters the aerodynamics menu, which provides text commands that create cell registers and define adaption criteria that can be useful for aerodynamic simulations.

mesh/adapt/predefined-criteria/aerodynamics/

shock-indicator/ (New)

Enters the shock indicator menu, which provides text commands that create cell registers and define adaption criteria that can be useful for simulations with shocks.

mesh/adapt/predefined-criteria/aerodynamics/shock-indicator/

density-based (New)

Creates cell registers and defines an adaption criterion that is suitable for simulations with shocks that use the density-based solver or the pressure-based solver with a fluid that uses a real-gas or ideal-gas model for the density.

mesh/adapt/predefined-criteria/

combustion/ (New)

Enters the combustion menu, which provides text commands that create named expressions and cell registers and define adaption criteria that can be useful for combustion simulations.

mesh/adapt/predefined-criteria/combustion/

flame-indicator (New)

Creates named expressions and cell registers and defines adaption criteria that are suitable for combustion simulations, so that the mesh is refined along a progressing flame front using various criteria like temperature, vorticity, species, and DPM concentration (depending on which models are used). There is also an option for refining a spherical spark region prior to a transient run, which after a specified time is then coarsened back to the original mesh.

mesh/adapt/

profile/ (New)

Enters the profile menu.

mesh/adapt/profile/

clear (New)

Clears the adaption profiling counters.

mesh/adapt/profile/

disable (New)

Disables adaption profiling.

mesh/adapt/profile/

enable (New)

Enables adaption profiling.

`mesh/adapt/profile/`

`print` (New)

Prints adaption profiling results.

`mesh/adapt/set/`

`maximum-cell-count` (New)

Sets an approximate limit to the total cell count of the mesh during adaption. Fluent uses this value to determine when to stop marking cells for refinement. A value of zero places no limits on the number of cells.

`mesh/adapt/set/`

`minimum-edge-length` (New)

sets an approximate limit to the edge length for cells that are considered for refinement. Even if a cell is marked for refinement, it will not be refined if (for 3D) its volume is less than the cube of this field or (for 2D) its area is less than the square of this field. The default value of zero places no limits on the size of cells that are refined.

`plot/`

`cumulative-plot/` (Changed)

Defining a cumulative plot now creates a plot object that is saved with the case file.

`plot/cumulative-plot/`

`force-or-moment-setup` (Deleted)

Cumulative plots are now created as objects saved with the case. Use `plot/cumulative-plot/add` to create a new cumulative plot.

`plot/cumulative-plot/`

`add` (New)

Create a new cumulative plot.

`plot/cumulative-plot/`

`delete` (New)

Delete an existing cumulative plot object.

`plot/cumulative-plot/`

`edit` (New)

Edit an existing cumulative plot object.

plot/cumulative-plot/

list (New)

Print the names of the existing cumulative plot objects to the console.

plot/cumulative-plot/

list-properties (New)

Print the properties of the specified cumulative plot object to the console.

preferences/appearance/

surface-specularity (New)

Specify the specularity of contour surfaces. Sepecularity is the reflectiveness of a surface; higher values (closer to 1) equate to a more reflective surface.

preferences/general/

flow-model (Deleted)

This command is relocated to `preferences/simulation/flow-model`.

preferences/graphics/performance/

fast-display-mode/ (New)

Enter the menu for controlling fast graphics performance preferences.

preferences/graphics/performance/fast-display-mode/

faces-shown (New)

Specify whether or not faces are shown when interacting with the model.

preferences/graphics/performance/fast-display-mode/

nodes-shown (New)

Specify whether or not nodes are shown when interacting with the model.

preferences/graphics/performance/fast-display-mode/

perimeter-edges-shown (New)

Specify whether or not the perimeter edges are shown when interacting with the model.

preferences/graphics/performance/fast-display-mode/

silhouette-shown (New)

Specify whether or not the silhouette is shown when interacting with the model.

`preferences/graphics/performance/fast-display-mode/`

`status` (New)

Specify whether the fast interactive display settings are on, off, or automatic.

`preferences/graphics/performance/fast-display-mode/`

`transparency` (New)

Specify whether or not transparencies remain when interacting with the model.

`preferences/`

`simulation/` (New)

Enter the menu for simulation preferences.

`preferences/simulation/`

`flow-model` (New)

Specify the default turbulence model (viscous).

`preferences/simulation/`

`local-residual-scaling` (New)

Use local residual scaling for solution residuals.

`preferences/simulation/`

`report-definitions/` (New)

Enter the menu for report definition preferences.

`preferences/simulation/report-definitions/`

`automatic-plot-file` (New)

New report definitions will automatically create associated report files and plots.

`report/`

`modified-setting/` (New)

Enter the modified settings menu.

`report/modified-setting/`

`modified-setting` (New)

Specify which areas of setup will be checked for non-default settings for generating the **Modified Settings Summary** table. The table is displayed tabbed with the graphics window.

`report/modified-setting/`

`write-user-setting` (New)

Write the contents of the **Modified Settings Summary** table to a file.

`solve/monitors/residual/`

`enhanced-continuity-residual?` (New)

Enables/disables an enhanced formulation for the local scaling of the continuity residuals with the pressure-based solver, so that the absolute mass flow rate at each cell is used. This text command is only available when the computing of the local scale is enabled through the `solve/monitors/residual/scale-by-coefficient?` text command.

`solve/report-definitions/`

`add` (Changed)

There is a new option `retain-instantaneous-values?` (available when the **Average Over** value is greater than `1`) that, when enabled, allows for the containing report file to store both the averaged values as well as the instantaneous values. This provides the flexibility to vary the **Average Over** value in a future session.

`solve/report-files/`

`add` (Changed)

There is a new option `write-instantaneous-values?` which, when enabled, saves instantaneous values, along with the specified averaged values. This preserves the ability to change a report definition's **Average Over** value in a future session.

`solve/report-plots/`

`add` (Changed)

There is a new option `plot-instantaneous-values?` which allows you to plot the instantaneous values of the plotted report definition along with the averaged values of that report definition.

`solve/set/advanced/`

`bcd-scheme-type` (Deleted)

This command has been removed.

`solve/set/advanced/`

`bcd-weights-freeze` (New)

Enables/disables freezing of weighting coefficients of the central differencing and the upwind components of the BCD scheme. This dialog command requires the iteration number, after which the BCD scheme weights are to be frozen at each timestep. Freezing the BCD weighting coefficients may help to improve convergence of the timestep iterations as described in Bounded Central Differencing Scheme in the *Fluent Theory Guide*.

`solve/set/`

`data-sampling` (Changed)

This command now has an additional prompt for force statistics.

`solve/set/`

`flux-type` (Changed)

This text command is now available when using the pressure-based solver for cases that do not use a multiphase model (except for the wet steam model). It allows you to select a high-order velocity interpolation with a Rhie-Chow correction for the pressure gradient difference that is weighted by either the distance (the default) or the momentum coefficient; it also provides an `Auto Select` option that automatically selects the flux interpolation type based on the setup of the case.

`solve/set/multiphase-numerics/advanced-stability-controls/`

`anti-diffusion/` (New)

Enters the anti-diffusion menu. This item is available for VOF cases with the **Interfacial Anti-Diffusion** option enabled.

`solve/set/multiphase-numerics/advanced-stability-controls/anti-diffusion/`

`enable-dynamic-strength?` (New)

Enables dynamic strength to reduce compression in the direction tangential to the interface.

`solve/set/multiphase-numerics/advanced-stability-controls/anti-diffusion/`

`set-dynamic-strength-exponent` (New)

Sets the cosine exponent in the dynamic strength treatment in Equation 22.1 in the *Fluent User's Guide*).

`solve/set/multiphase-numerics/advanced-stability-controls/anti-diffusion/`

`set-maximum-dynamic-strength` (New)

Sets the maximum value of dynamic anti-diffusion strength in Equation 22.1 in the *Fluent User's Guide*).

`solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/instability-detector/`

`set-cfl-type` (New)

Selects the CFL number type for detection of an unstable event. This command becomes available once the `enable-instability-detector?` text option has been enabled.

`solve/set/previous-defaults/`

`undo-2021r2-default-changes?` (New)

Allows you to undo the following enhancements introduced in version 2021 R2 of Ansys Fluent:

- Using a gradient computation enhancement for interior mesh interfaces, which resolves convergence issues and may improve accuracy in cases that have interior type mesh interfaces (such as fluid-fluid interfaces) with poor mesh quality.
- Using harmonic averaging of the turbulent kinetic energy for the calculation of the turbulence intensity used by the Intermittency Transition model and Algebraic Transition model, which can help to overcome hysteresis effects (dependence of transition location on initial condition) observed mainly in applications with low turbulence intensity and natural transition.
- Computation of transient field statistics has been slightly changed to now compute the statistics at the end of each time-step rather than at the beginning of each timestep. This corrects some defective behavior that could previously occur with: Using the Extrapolate Variables option together with statistics calculation and statistics computation for certain time-derivative quantities (such as dp/dt). Reverting this change may result in hidden errors.

`solve/set/`

`second-order-time-options` (Changed)

For cases with the **Bounded Second Order Implicit** formulation, an option to use the first order upwind descritization scheme for species transport, population balance, and energy equations has been added.

`solve/set/`

`stiff-chemistry` (Changed)

A new prompt has been added for an ID of the multi-species phase where the stiff chemistry solver is used.