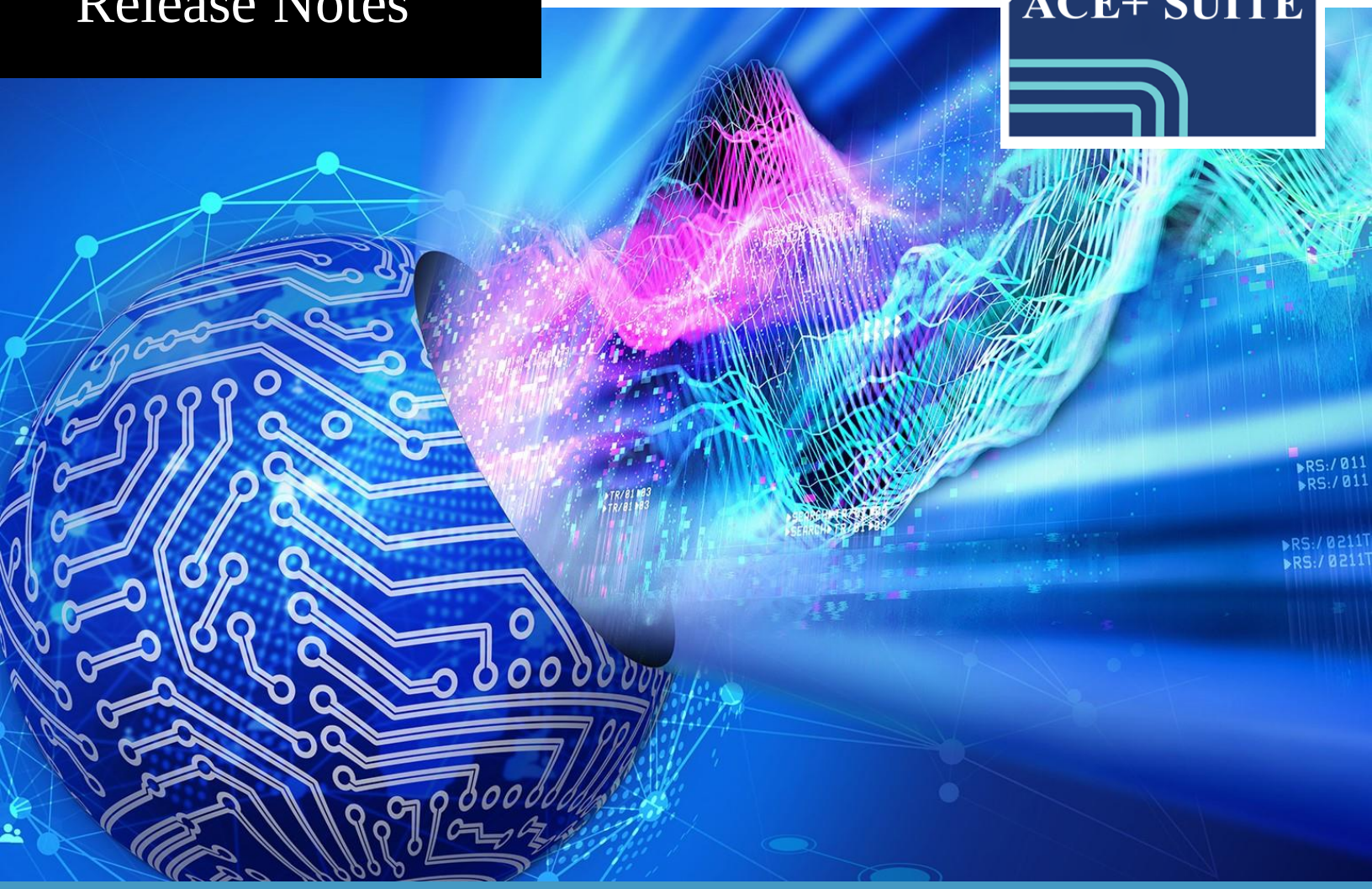
Release Notes

ACE+ SUITE



2026.0

Applied Materials Confidential

Table of Contents

Confidentiality Notice

The materials and information contained herein are provided by Applied Materials to its Customer solely for Customer's internal business purpose related use. Applied Materials retains all right, title, interest in and copyrights to the materials and information herein. The materials and information contained herein constitute confidential information of Applied Materials, and Customer shall not disclose or transfer any of these materials or information to any third party.

Introduction

This document introduces the V2026.0 release of the ACE+ Suite of applications. It summarizes the new features, enhancements and the list of issues that are corrected since the V2025.0 release.

ACE+ SUITE of Applications

The ACE+ SUITE package includes applications that provide the necessary tools for advanced multiphysics analysis in a virtual prototype environment. The complete list of applications is shown below:

- GEOM - Interactive CAD-based Geometry Creation and Surface Meshing Tool
- VisCART - Automated Cartesian Mesh Generator for Complex Geometries
- ACE+ - Multiphysics Solver for Fluid, Thermal, Chemical and Plasma Dynamics
- FASTRAN - Compressible flow and rigid body motion solver
- TOPO - Physics-based simulator for multi-material deposition and etch workflows
- SimManager - Parametric analysis and optimization support tool
- VIEW - Advanced Post-Processing and Visualization Tool
- Toolkit – Utilities for DTF import / export, mapping and other diagnostics

Supported Platforms

ACE+ SUITE V2026.0 is available on the following **64-bit** platforms:

- **Windows:**
  - Windows 10 and later (Intel or AMD)
- **Linux:**
  - Red Hat Enterprise Linux 8+ (glibc 2.28)
  - Red Hat Enterprise Linux 9+ (glibc 2.34)
  - (Intel or AMD)

User Subroutine - Intel FORTRAN Compiler Versions

The table below outlines the versions of Intel® Fortran Compiler and Microsoft® Visual Studio supported for user subroutine development in ACE+ SUITE. Each entry indicates the ACE+ version from which a specific compiler or IDE is supported, until it is superseded by a newer version.

ACE+ Version	Intel Compiler Version (Linux)	Intel Compiler Version (Windows)	Visual Studio Version (Windows)
2026.0	oneAPI 2024	oneAPI 2024	VS2022

The table below lists older ACE+ SUITE versions along with the corresponding supported Intel® Fortran Compiler and Microsoft® Visual Studio versions.

ACE+ Version	Intel Compiler Version (Linux)	Intel Compiler Version (Windows)	Visual Studio Version (Windows)
2013.0	2011.1	2011.1	VS2008
2013.4	2011.1	2011.1	VS2010
2015.0	2015.3	2015.3	VS2010
2017.0	2015.3	2015.3	VS2013
2017.5	2016.4	2016.4	VS2013
2018.5	2016.4	2016.4	VS2015
2021.0	2020.0	2020.0	VS2019

Running Older Versions

Multiple versions of ACE+ SUITE can generally be installed and run in parallel. However, please note that legacy ESI ACE+ SUITE and Applied Materials ACE+ SUITE require separate licenses.

Environment Requirement

If you install V2026.0 software alongside previously installed older version(s), modify your `PATH` environment variable so that the newest `UTILS\bin` directory appears first in your `PATH`.

Then, it is possible to run older versions of the software using the `-runver` **<version>** option from the command prompt or from the MS Windows **Start** > **Run** app.

Example

```
CFD-GEOM -runver 2024.0
```

This command will launch CFD-GEOM software V2024.0.

To use -runver option with both the legacy ESI ACE+ versions and Applied Materials ACE+ Suite, it is recommended that users install both software in the same directory. This enables the environment variables **ESI_HOME** (used by the legacy version) and **ACE_SUITE_HOME** (used by the new versions) to be the same.

License Requirement

You will need a 2026 or later license to run V2026.0 applications.

Flexnet V11.19.1 (provided in this package or downloadable from the Applied Materials Portal) should be used for the V2023.0 and later applications to work.

- Install the FNP Licensing Service (install_FNP) provided in the toolkit.
- Run the getLmHostIds program (from a terminal) to get the host information needed for license generation.
- Use the lmadmin program to install and administer the license. Refer to the README.txt file and the fnp_LicAdmin.pdf manual within the Flexnet toolkit for detailed instructions.

Installation

This section summarizes the main steps to install ACE+ SUITE.

Installing ACE+ SUITE on Windows

1. Extract the **ACE+Suite_V2026.0_Windows-x64.zip** archive to a temporary directory.
2. Click **setup.exe** and follow on-screen instructions.

The installer installs the required packages and configures environment variables, including ACE_SUITE_HOME and updates to the system PATH.

Installing ACE+ SUITE on Linux

1. Extract the **ACE+Suite_V2026.0_Linux-x64-glibc2.28.tar.gz** (for RHEL 8) or **ACE+Suite_V2026.0_Linux-x64-glibc2.34.tar.gz** (for RHEL 9) archive to a temporary directory.
2. From within the extracted directory, run **install.com** and follow on-screen instructions to complete installation.
3. Login as the end user and run **configure_user_account.sh** to setup the necessary environment variables (ACE_SUITE_HOME and PATH).

Manually Setting Up Environment Variables

To manually configure the ACE+ SUITE environment, follow these steps:

1. Set ACE_SUITE_HOME

   This variable defines the installation directory of the ACE+ SUITE software.

   Windows: *ACE_SUITE_HOME = C:\Program Files\Applied_Materials*

   Linux – BASH: *export ACE_SUITE_HOME=/usr/local/Applied_Materials*

   Linux – CSH: *setenv ACE_SUITE_HOME /usr/local/Applied_Materials*

2. Update PATH

   They should reflect the new path to UTILS/bin.

   Windows:

   Variable Name: **PATH**

   Value: **%ACE_SUITE_HOME%\ACE+Suite\2026.0\UTILS\bin**

Linux – BASH:

```
export PATH=$ACE_SUITE_HOME/ACE+Suite/2026.0/UTILS/bin:$PATH
```

Linux – CSH:

```
setenv PATH $ACE_SUITE_HOME/ACE+Suite/2026.0/UTILS/bin:$PATH
```

New Features and Enhancements

This chapter describes the new features and improvements of ACE+ SUITE V2026.0 release.

GEOM

CFD-GEOM V2026.0 includes the following new features and enhancements. Please refer to the *ACE+ SUITE 2026.0 CFD-GEOM User's Guide* for further details.

Periodic Mesh Setup for Unstructured Domains

In semiconductor reactor modeling - such as in plasma etch, deposition, or cleaning systems - periodic and cyclic boundary conditions are commonly used to simulate repeating unit structures within the reactor chamber. These conditions allow engineers to model a representative segment of the geometry while capturing the behavior of the full system, reducing computational cost and complexity.

Typical examples include:

- Repetitive showerhead or gas inlet arrays in CVD/ALD reactors
- Periodic baffle or exhaust structures in chambers
- Segmented wafer holders or susceptor designs

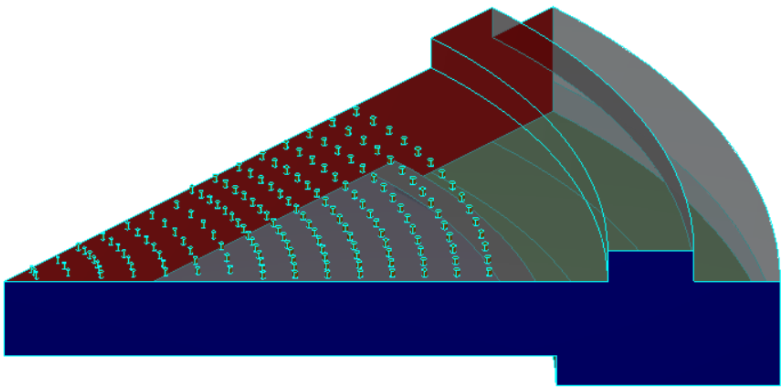


Figure 1. Rotational periodicity example (red and blue surfaces)

For solvers to correctly apply these boundary conditions, the surface meshes on periodic boundaries must be identical under a translation or rotation operation. Previously, users had

to rely on structured face meshing or semi-structured meshing to manually enforce this requirement, which limited flexibility and increased setup time.

CFD-GEOM now introduces support for translational and rotational periodicity in meshing on unstructured domains, eliminating the need for structured faces. This enhancement allows users to:

- Define periodic relationships between surface pairs
- Preview and adjust transformations interactively
- Synchronize surface meshes prior to volume meshing.

Setting Periodicity

The periodic mesh setting tools are in the model tree. Navigate to **Mesh Tools** in the model tree. Choose either **Translational** or **Rotational** periodicity. Select surface pairs using the middle mouse button. Adjust transformation parameters until the preview aligns correctly. Click Apply or use the middle mouse button to confirm. Indicators (e.g., T1, S1) will appear for each surface pair. The **Update** button becomes active once settings are applied. At any time, the user can update the periodic meshes by using the **Update** button. On a successful update surface which border the periodic surfaces will require meshing therefore, the unstructured surface mesh tool will be automatically activated and any surfaces which do not have a mesh will be selected. The user can then set the desired mesh type for those surface and mesh. To ensure that the periodic meshes are not removed, it is recommended that the **Remesh Curves** option is unchecked.

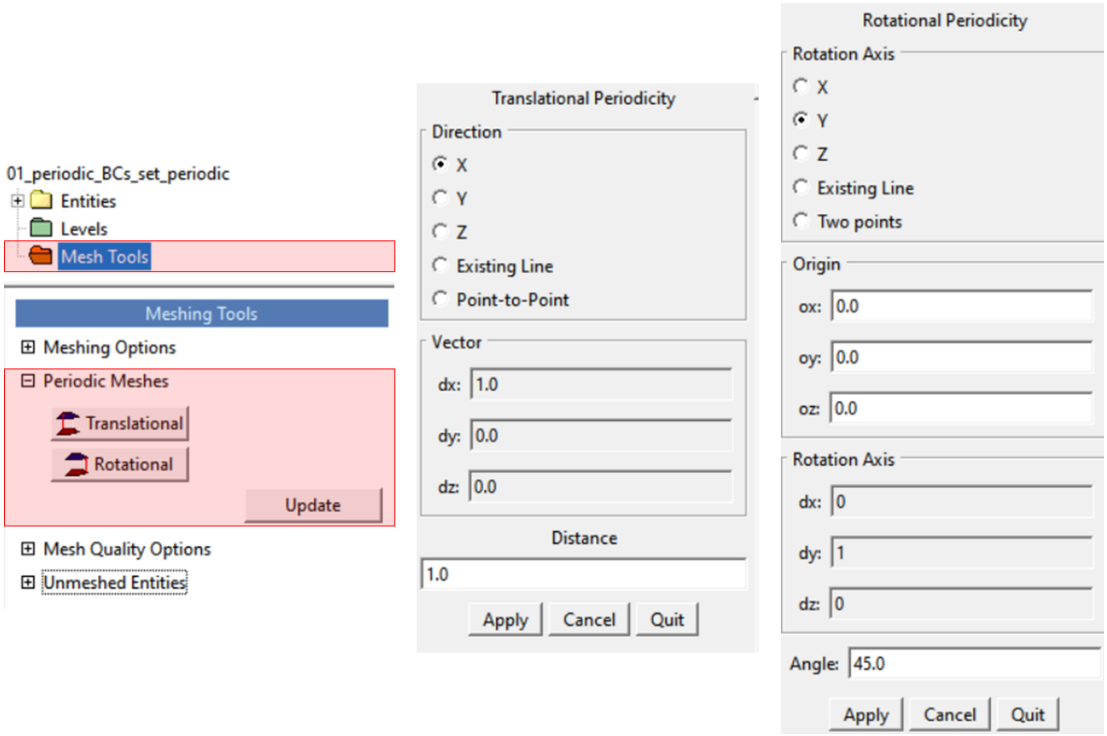


Figure 2. Periodicity settings in CFD-GEOM

Any periodic meshes should be updated prior to the volume meshing. During DTF output, if the user has a periodic mesh setting and CFD-GEOM determines that the surface meshes are not periodic a dialog will be shown allowing the user to abort the DTF output or continue.

Limitations

- Only one translation or periodic settings is allowed at any given time; however, the user can specify multiple periodic mesh settings, one at a time, and update as needed.
- Use of structured edges is prohibited; the direct curve meshing tool can be used in lieu of the use of structured edges.
- Boundary layers are allowed however, projected boundary layers which affect the periodic surfaces will cause a loss of periodicity.
- Generation of honeycomb meshes will cause loss of periodicity.

Quadrilateral-Dominant Surface Meshing

This release introduces a new quad-dominant surface meshing algorithm that generates meshes composed primarily of quadrilateral elements with less than 1% triangles. Compared to the quad morphing algorithm, this new method delivers lower cell counts and enhanced mesh quality; the meshes are very similar to the results from quad paving. Additionally, it offers greater robustness than the traditional paving algorithm for complex models, while achieving comparable results in terms of cell count and mesh quality. In cases where the quad dominant mesh fails the user can slightly modify the Expansion Factor and attempt to regenerate the mesh. To utilize the quad-dominant meshing feature, navigate to the **Surface Meshing** settings and select the **Quad Dominant** option.

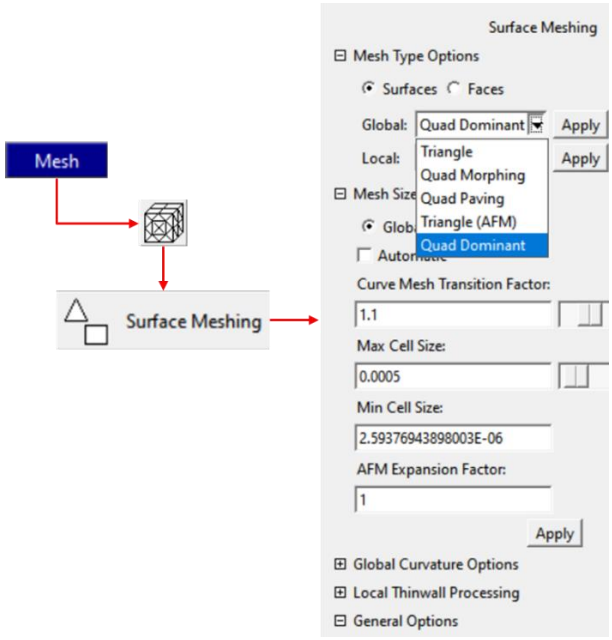


Figure 3. New Quad Dominant surface meshing tool

CAD Connection with NX for Geometry Regeneration

In CFD workflows, preparing CAD geometry for simulation often involves manual steps such as defeaturing, scaling, and ensuring watertightness. These tasks can be time-consuming and error-prone, especially when design changes require repeated geometry updates. Traditional workflows rely on exporting geometry from CAD tools and importing it into CFD pre-processors, which introduces inefficiencies and risks of data loss.

To streamline this process, CFD-GEOM V2026.0 introduces CAD Connection with NX for Geometry Regeneration, enabling users to drive geometry updates, based on user-defined parameters or suppression of features, directly from within the CFD-GEOM environment, while leveraging the modeling capabilities of Unigraphics NX. The CAD connection is only available currently on Windows. Key aspects of this connection include:

- **Native CAD Access:** Geometry and associated parameters are imported directly from NX, preserving fidelity and eliminating the need for intermediate translation formats.
- **Parameter-Driven Regeneration:** Users can modify geometry parameters within CFD-GEOM. These changes are communicated to NX, which regenerates the geometry accordingly.
- **Model Comparison Tools:** The regenerated geometry can be displayed using the "Allow Selective Import" option, allowing the user to verify the model changes and select the desired parts prior to full import.
- **Automatic Metadata Transfer:** CAD attributes such as units, orientation, and entity grouping are automatically transferred to CFD-GEOM, ensuring consistency and reducing setup errors.

Usage Instructions

To use the NX connection:

1. **Enable CAD Connectivity**

   Go to **Edit** → **Preferences** and check **Connect to CAD Systems**

2. **Modify Parameters**

   Access geometry parameters via **Model Manager** → **CAD Data**. Adjust values as needed.

3. **Regenerate Geometry**

   Click **Generate Model** to trigger NX-based regeneration. The updated model will appear in a new viewer, which can be tiled for comparison with the original.

Limitations

- **Supported CAD Platform:** Only Unigraphics NX is supported in this release. This functionality has only been tested with NX 2022 but other versions may be successful.

- **Editing Scope:** Only parameter values can be modified. Direct editing of geometry topology or expressions is not supported. Feature suppression is also possible.
- **Automation:** Python scripting for CAD connection and parameter control is not yet available. Future releases may introduce scripting support for automation and batch processing.

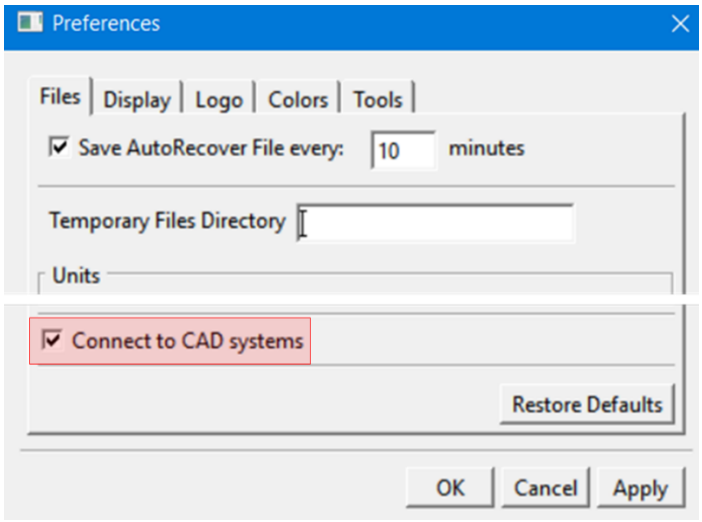


Figure 4. Enabling connection to CAD systems

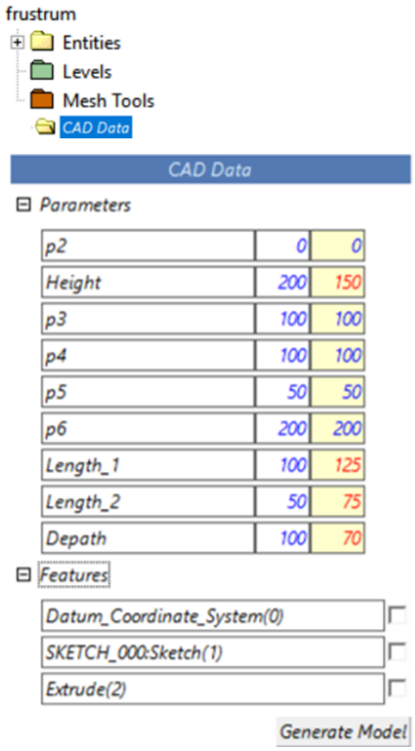


Figure 5. Modifying parameters and regenerating a model

Transformation Tool for Geometry Alignment

Accurate alignment of geometry is a fundamental step in simulation workflows, particularly when preparing models for meshing, symmetry exploitation, or comparative analysis. Manual alignment of arbitrarily oriented geometries—whether 2D sections or full 3D bodies—can be tedious and error-prone. To address this, a new transformation tool has been introduced that enables precise vector-based alignment to any desired orientation. This tool may be particularly useful for aligning 2D models to the XY plane.

The **Align Entities to Direction** tool provides a robust mechanism to reorient any 2D and 3D geometries using vector-based transformations. By specifying initial (source) and final (target) vectors, users can align geometry sections or entire bodies to a desired direction.

This tool is especially useful for:

- Aligning 2D cuts for axisymmetric models and automating geometry setup
- Aligning 3D geometry orientation for consistent analysis

Usage Instructions

This tool can be invoked from **Geometry** → **Transform Entities** → **Align Directions**

- **Select Geometry:** Choose any 2D or 3D geometry to be transformed.
- **Define Source Vector:** Specify the current orientation of the geometry.
- **Define Target Vector:** Define the desired orientation.
- **Apply Transformation:** The tool computes and applies the necessary rotation and translation to align the geometry.

Additional post-transformation operations may be needed to bring model to its desired final position.

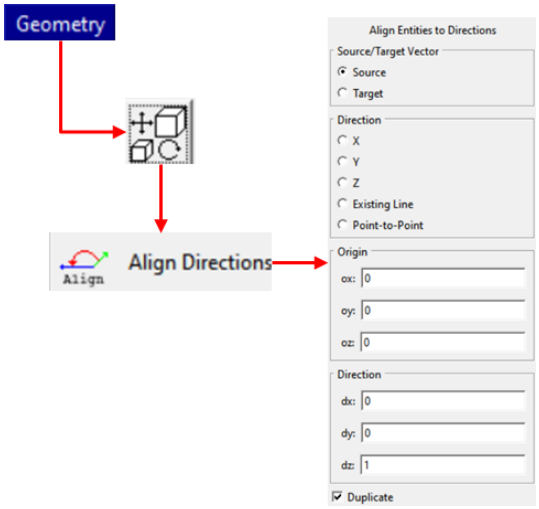


Figure 6. Transformation tool for geometry alignment

Improvements to Replace Parts Feature

The Replace Parts workflow in GEOM has been improved to address certain scripting limitations involving duplicate part names. In previous versions, Python scripts failed to replace all instances when multiple objects shared the same name—only one would be replaced. This issue was limited to scripting mode and did not affect GUI operations.

Enhancements

1. A Duplicate Names checkbox has been added to the Replace Parts file dialog.

   Unchecked: Only explicitly selected parts are imported and replaced.

   Checked: All parts with matching names are imported and replaced.

2. A new argument has been added to the journaled Python function GGeometry.ReplaceParts:

   duplicate_names = 0 (default): Replaces only selected parts.

   duplicate_names = 1: Replaces all parts with matching names.

These updates ensure consistent behavior between GUI and scripting modes.

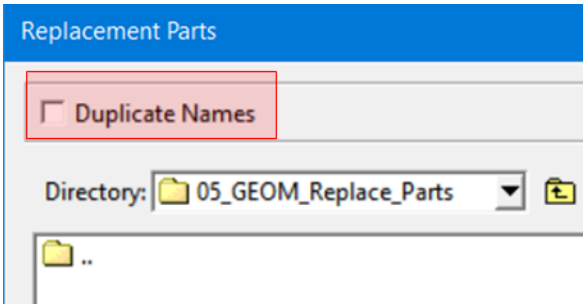


Figure 7. Replace parts dialog

Improvements to Selective Parts Import

The selective parts import has been improved to allow import of multiple parts with the same name by selection of one such instance of the part. Previously, all parts in the same model were given unique names thus, allowing the user to import exactly the parts specified.

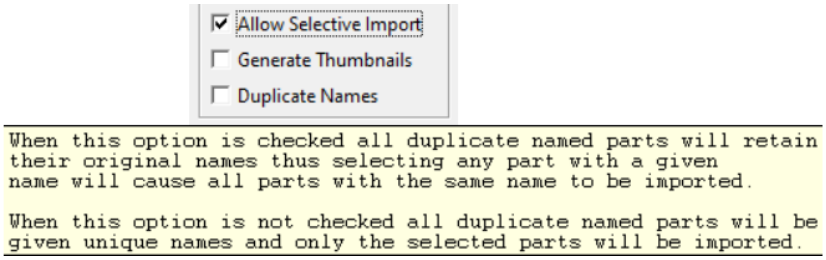


Figure 8. Selective Import with option to import multiple parts with same name

When the “Allow Selective Import” option is activated the option to allow “Duplicate Names” will be activated. When the “Duplicate Names” option is activated then duplicate named parts will retain their original names thus selecting any part with a given name will cause all parts with the same name to be imported. When this option is not checked all duplicate named parts will be given unique names and only the selected parts will be imported.

In all instances, parts without names will be assigned a unique name.

Expanded Scripting Capabilities

This release introduces enhancements to CFD-GEOM’s scripting interface, enabling greater automation, improved visualization, and flexible execution modes.

New Capabilities

1. Color Assignment to Entities

   Users can now assign custom colors to geometry entities directly through scripts.

   SetColor(entities, red=255, green=255, blue=255, alpha=255)

2. PNG Export from Scripts

   Scripts can now generate PNG images of the geometry view, enabling automated documentation and visual output. This feature requires OpenGL version 3.0 or higher. PNG export is silently ignored in headless or XCFD-GEOM environments.

   SavePNG(filename, reset_view = 1, display_meshes = 0, width = 640, height = 480)

3. Headless Execution Mode

   The default **-script** option now launches a background GUI (not visible to the user) to support features like PNG export. A new **-nogui** option allows scripts to run without invoking the GUI and replicates the previous behavior.

   CFD-GEOM -nogui -script <script.py> (replicates old behavior)

VisCART

CFD-VisCART V2026.0 includes the following new features and enhancements. Please refer to the *ACE+ SUITE 2026.0 CFD-VisCART User's Guide* for further details.

Improvements to Geometry Operations

Repeatable Translate / Rotate Operations

CFD-VisCART now allows users to specify the number of repetitions for rotate/translate operations, making it easy to duplicate geometry in a consistent manner. This feature is particularly useful when creating patterned arrangements or replicating components around a central axis without manual repositioning.

In the rotate/translate dialog, set the "Number of Times" field to the desired count. CFD-VisCART will apply the specified transformation repeatedly, generating evenly spaced copies based on your input parameters.

Geometry Operations on Sources

Geometry operations such as scale, translate, rotate, and mirror can now be applied directly to sources, just like geometries. This means you can adjust refinement regions and other source-based controls without recreating them, keeping meshing intent aligned with the design. The only exception is linked surface sources, which act as pointers inside the source. If you modify the surface geometry, the associated source updates automatically. If linked surface sources are selected during an operation, a popup will remind you to deselect them before proceeding.

Select the desired sources and apply rotate, translate, or scale operations using the standard geometry operation dialogs. For repeated transformations, specify the "Number of Times" field in the dialog to duplicate sources consistently.

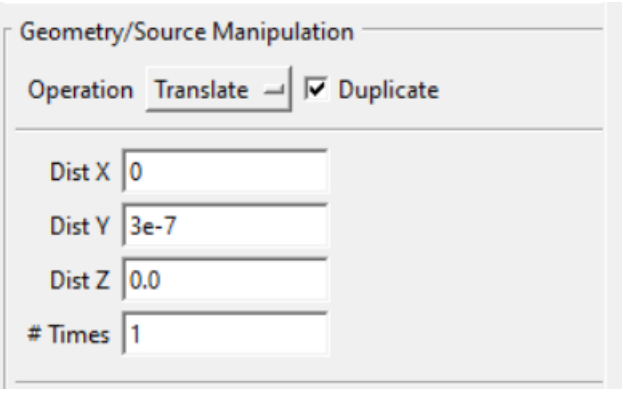


Figure 9. Geometry / Source Manipulation Tool

Domain Management and Grouping

Domain Grouping and Unified Zone Export

A new domain grouping mechanism has been introduced to simplify managing complex projects. You can now group selected domains and export them into a single zone in the DTF file. This capability supports writing multiple domains inside the same ACE+ Volume Condition, enabling consistency in model setup and improving solver performance.

The domain panel now includes a Domain Group Tree on the left side of the domain list. You can create groups by selecting domains and clicking the Group button. A domain cannot belong to multiple groups - if you add a domain to a new group, it is automatically removed from its previous group. In the Domain Panel, select the domains you want to group and click the Group button.

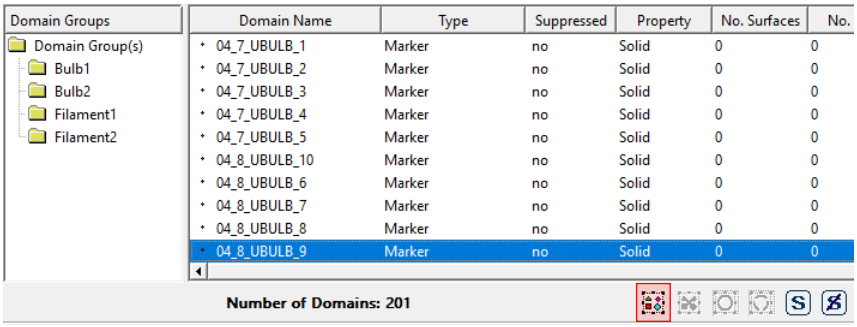


Figure 10. Domain Grouping

When saving the DTF file, enable the “Group Domains into One Zone” checkbox in the export dialog. By default, this option is checked, so grouped domains will be written into the same Volume Condition automatically.

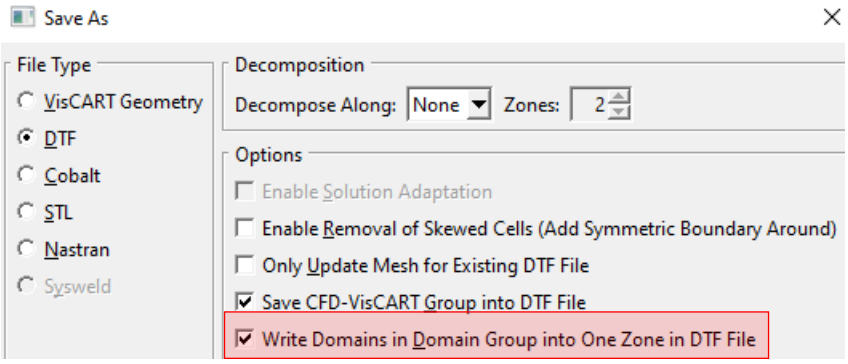


Figure 11. Write Domain Groups into One Zone in DTF

Auto-Merge Connected Domains

Connected auto-generated domains can now be merged automatically to simplify domain structure and reduce manual cleanup. This enhancement ensures that domains sharing compatible boundaries are combined into a single coherent unit, improving clarity and

reducing clutter in the domain tree. Additional checks have been added to prevent incorrect merges in edge cases, such as disconnected domains linked through a third domain or domains with identical left/right sets.

Enable the “Merge Auto Domains After Meshing” option in the Multi Domain settings panel.

Optionally, check “Delete Remaining Auto Domains” to remove any unmerged auto-generated domains after the merge process.

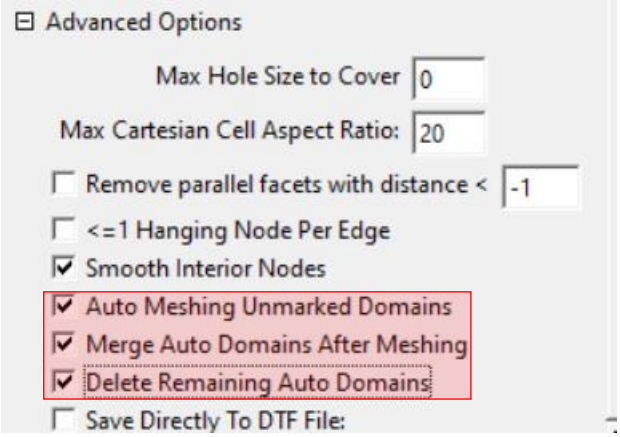


Figure 12. Merge Auto-generated Domains

Mesh Quality and Cell Splitting

Improved Cell Splitting Near Feature Lines and Curved Regions

Feature lines are now used to separate triangles within the same group during cell splitting. This prevents incorrect normal averaging that previously caused artificial bumps or dents in the mesh near sharp intersections or curved corners. By isolating triangles across feature lines, the mesher calculates normal more accurately, resulting in smoother transitions and better mesh fidelity around complex geometry.

Similarly, cell splitting in curved regions now uses multiple reference points within the same group to define splitting planes. This approach improves mesh fidelity by better capturing local curvature without excessively increasing cell count. The result is smoother transitions and more accurate representation of curved surfaces.

Older versions often produced distorted cells in these regions, leading to uneven surfaces and associated near wall mesh qaulity. The updated approach ensures that even challenging intersections—such as cylinders sharing a surface—are handled correctly without manual intervention.

Boundary Facet Skewness Reduction

This improvement addresses skewness issues in high aspect ratio cells created during splitting. Previously, when a long cell was divided into two, the interface between them often had an extremely small skewness angle, making the mesh less useful. The new approach

merges adjacent cells on both sides of the split interface and repositions their centers toward the middle area. This significantly improves skewness and reduces the number of distorted facets caused by elongated cells.

The process runs automatically at the end of meshing and does not require user intervention. Improved skewness should result in better solver stability and accuracy.

Aspect Ratio Control for Cartesian Cells

A new global setting limits the maximum aspect ratio for Cartesian cells to prevent severe skewness in shared facets. To maintain a skewness angle of at least 5°, a new input field "Max Cartesian Cell Aspect Ratio" has been added under **Global Settings → Advanced Tab**, with a default value of 20.0. This ensures better mesh quality and solver stability by avoiding extremely elongated cells. This setting applies to single-domain and multi-domain meshing workflows.

In batch scripts:

set_max_aspect_ratio [aspect_ratio_value]

In Python scripts:

pyviscart.set_max_aspect_ratio(aspect_ratio_value)

Performance and Logging Enhancements

Accelerated DTF Saving for Large Domain Counts

DTF export has been optimized for models with hundreds of domains by introducing domain-specific face index arrays. Instead of traversing the entire mesh for each domain, the system now uses precomputed face indices, dramatically reducing save time. For example, a case with 370+ domains that previously required over 4 hours to save now completes in about 18 minutes. This improvement ensures that even very large projects can be saved efficiently.

Enhanced DTF Logging with Domain Identifiers

When writing the DTF file, the log now prints the Domain Name of each specific volume along with its ID. This enhancement allows users to easily correlate the domain name specified in the VGD file with the corresponding zone in the DTF export. Clear domain-to-zone mapping improves traceability and simplifies troubleshooting for complex models.

Usability Improvements

DTF Grid Quality Check

CFD-VisCART allows users to evaluate mesh quality directly from DTF files, even if the DTF was not generated by VisCART. This feature provides immediate access to mesh quality

metrics—such as volume and face angle statistics—without waiting for solver pre-checks, which can be time-consuming for large models.

From Command Line:

CFD-VisCART -check_dtf_q dtf_file

Sim number is assumed to be 1. DTF -fdel_sim can be used to delete simulations in a DTF file.

In Batch Mode:

check_dtf_quality dtf_file sim#

Within a Python Script:

pyviscart.check_dtf_quality(dtf_file, sim#)

Batch Renaming of Entities

Users can now rename multiple groups, solids, and domain markers in one operation with automatic indexing. This feature ensures consistent naming across large projects, reduces manual effort, and improves clarity for downstream processes. Select multiple items and use the Rename dialog. Suffixes such as _1, _2, etc., are added automatically to maintain uniqueness.

ACE+

CFD-ACE+ V2026.0 includes the following new features and enhancements. Please refer to the *ACE+ SUITE 2026.0 CFD-ACE User's Guide* for further details.

Hybrid Steady–Transient Solver

Industrial simulations often involve long quasi-steady phases interrupted by short, physics-rich events. In many cases, certain physics remain effectively steady over the simulation timescale—such as bulk gas flow or chamber wall temperature—while other phenomena demand accurate transient modeling to capture rapid changes and localized dynamics. For example, in ALD, precursor transport and chamber pressure may be steady, but surface reactions during pulsing steps are highly transient. In RTP, wafer heating may appear steady overall, yet radiative flux and temperature gradients during ramp-up or cooldown are strongly time-dependent. Running the entire simulation as fully transient wastes computational resources, while running it entirely as steady-state misses critical unsteady dynamics.

The Hybrid Steady–Transient capability introduces time-segmented solver mode switching, allowing models to run transient only where necessary and hold steady elsewhere for efficiency and accuracy. An advanced scheduler in GUI enables users to configure per-module time dependence so each physics module (Flow, Heat Transfer, Chemistry, etc.) can operate in Steady, Transient, or Hold mode at specific solver times. The advanced scheduler has two intuitive modes:

- Graph Mode: Visual, color-coded interface for scheduling time-dependent behavior.
- List Mode: Manual input for precise control over start times, durations, and end times.

It also allows to perform flexible simulation control:

- Insert, split, merge, or delete module/time segments.
- Crop schedules before or after specific time points.
- Import/export schedules via editable text files.

In addition, users can also specify restart files per module or segment for controlled initialization.

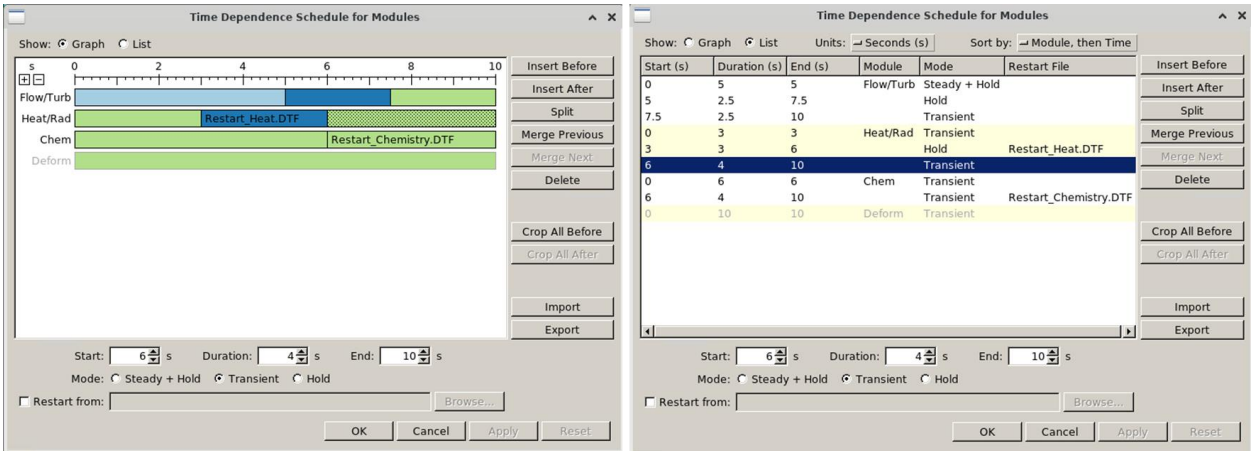


Figure 13. Scheduler in Graph Mode (left) and List Mode (right)

Usage

1. Enable Hybrid Mode:

   Navigate to **MO → Transient Conditions → Advanced Scheduling**.

   Select **Per Module Time Dependence** and click **Define** to open the Advanced Scheduler.

2. Set Up Segments:

   For each module, define Start Time, Duration, or End Time.

3. Choose the mode for each segment:

   Steady: Solver computes a steady-state solution at the start of the segment.

   Hold: Maintains the last computed solution without further updates.

   Transient: Performs full time-dependent calculations for the segment.

4. Restart Options:

   Assign restart files for any module or segment to initialize from a previous solution.

5. Visualization:

   Use Graph Mode for quick visual scheduling.

   Use List Mode for detailed numeric control.

   During runtime, the residual plotter distinguishes steady vs transient segments for clarity.

Limitations & Notes

- Supported modules: Flow, Turbulence, Heat Transfer, Radiation, Chemistry, and Grid Deformation.
- Dependent modules (e.g., Turbulence with Flow, Radiation with Heat) must share the same schedule.

- Restart files must match the simulation number of the current run.
- All outputs are generated in transient format for consistency.

Monte Carlo Radiation - Memory Optimization

Monte Carlo radiation simulations are inherently memory-intensive because they track millions of rays across sub-patches and store complex data structures – most notably, the view factor matrix. This matrix represents the ratio of rays absorbed to rays emitted by each sub-patch and is central to computing net radiation energy flow.

For large-scale models, storing this matrix can consume hundreds of gigabytes of memory, limiting scalability and requiring high-end hardware. Users need a transparent way to balance memory footprint against computational efficiency without compromising accuracy.

The new capability introduces two selectable methods for handling radiation energy computations:

1. Store Modified View Factor Matrix

   This traditional approach stores the matrix for reuse across iterations, reducing computation time but consuming significant memory. Memory requirements grow proportionally to the square of the number of sub-patches and scale further with the number of parallel processors.

2. Directly Compute

   This new method calculates radiation energy flow on the fly at each skipping interval instead of storing the matrix. This dramatically reduces memory consumption in large-scale models—while maintaining accuracy.

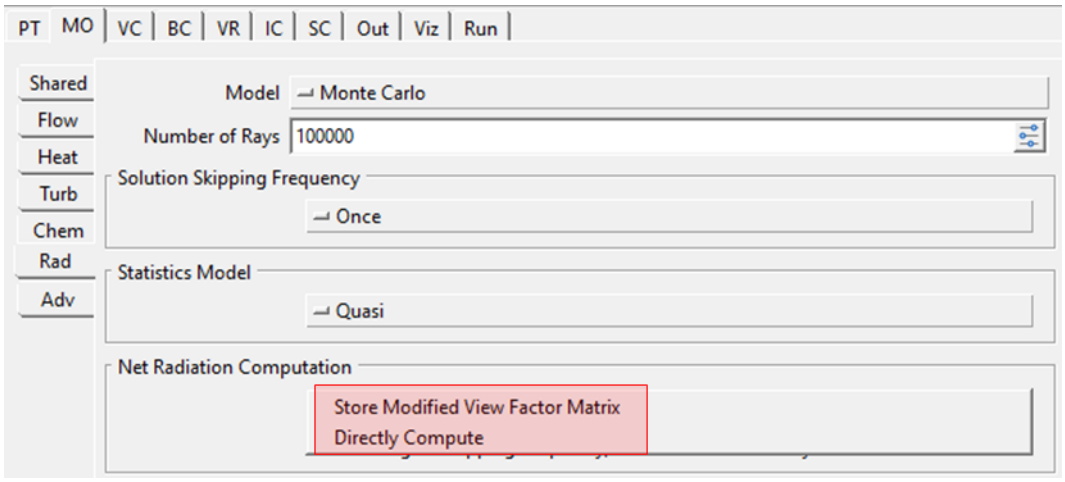


Figure 14. Net Radiation Computation Options

For example, in one representative case, storing the matrix consumed hundreds of gigabytes of memory, while the new Directly Compute approach required only a fraction of that – reducing memory usage by more than an order of magnitude. This trade-off allows engineers to run significantly larger models on systems with limited memory and to leverage parallel scalability without hitting memory bottlenecks. However, Directly Compute achieves this by recalculating radiation energy flow at every interval instead of reusing stored matrices, which can increase computation time. Users should choose based on their priorities – speed vs memory footprint.

When to Use Each Option

1. Store Modified View Factor Matrix:

   Best for smaller models or systems with abundant memory.

   Allows higher skipping frequencies for faster turnaround.

2. Directly Compute:

   Ideal for large-scale models or limited-memory hardware.

   Requires lower skipping frequency (recommended: 1) to maintain accuracy.

   May introduce a time penalty but benefits from parallel scalability.

Monte Carlo Radiation - Wall Radiation Flux Outputs

Previously, users who needed detailed radiation flux information were required to perform manual calculations to obtain quantities such as irradiation, radiosity, and absorbed flux on different boundary surfaces. This approach often introduced unnecessary complexity and clutter into the workflow. Based on user feedback, we have streamlined the process to deliver comprehensive outputs with minimal setup effort.

The new feature provides a single checkbox in the graphical interface, leveraging the existing wall radiation flux outputs from the solver to automatically include all relevant derived quantities. These include Irradiation, Radiosity, Emitted, Absorbed, Reflected, Transmitted, and Net flux. This enhancement ensures that users receive a complete set of radiation flux data without additional configuration or manual post-processing.

To use this feature, navigate to **Out** → **Graphic** → **Radiation** and enable Wall Radiation Fluxes. Once selected, the solver outputs are imported, and VIEW organizes and presents the derived quantities for visualization and reporting.

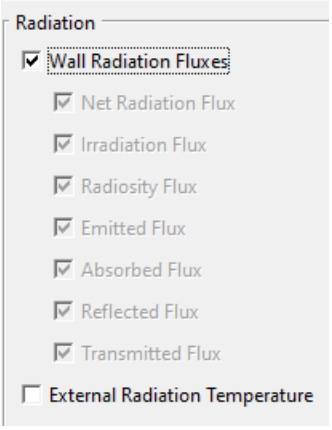


Figure 15. Additional Radiation Flux Outputs

Currently, this capability is available only for the Monte Carlo Radiation module.

Monte Carlo Radiation - Farfield Boundary Condition

In this release, the GUI options for Farfield boundary conditions has been simplified to align with current solver behavior. The previous option "**Include Radiation from Boundary Temperature**" has been removed because the solver now always accounts for radiation from the boundary temperature as part of its standard calculation. This change eliminates ambiguity and ensures the interface reflects the correct physics without requiring user input.

The updated UI consistently displays External Radiation Source Temperature and External Radiation Source Emissivity for all Farfield boundary conditions, with safe defaults provided. Other related fields, such as Surface Property, Number of Sub-patches, and Transmissivity, remain available but are organized for clarity.

Legacy scripts referencing the removed control are ignored gracefully, so existing workflows remain compatible.

Improved Heat of Reaction Source for Surface Reactions

The calculation of chemical heat release at reacting surfaces has been improved to ensure accurate energy conservation across all Heat Boundary Condition (BC) subtypes. This update enhances the fidelity of simulations involving conjugate heat transfer, isothermal walls, and reacting surfaces, particularly in processes such as CVD and catalytic reactions.

Previously, the heat source term was calculated using partial species sets and inconsistent logic for different boundary conditions. Surface species were not included, and conjugate boundaries used separate formulas that did not fully account for all reaction contributions. In the new implementation, the heat source now consistently includes gas-phase, bulk, and surface species for all applicable BC types. For conjugate boundaries, the chemical heat is split between the gas and solid sides using thermal weighting factors, ensuring proper energy partitioning in coupled heat transfer scenarios.

For isothermal boundaries, the heat source is explicitly set to zero. This change reflects the physical constraint that the wall temperature remains fixed, and any chemical heat is internally balanced by the solver rather than applied as an external flux. For Newton and external radiation boundaries, the full chemical heat source is applied, enabling accurate prediction of wall heat flux under convective or radiative exchange.

Including all species contributions and applying consistent logic across boundary conditions improves energy conservation and accuracy in thermal predictions. This is particularly important for CVD and catalytic systems where surface reactions dominate heat release. The updated approach also ensures robust handling of conjugate heat transfer by aligning heat partitioning with physical thermal resistances.

No user action is required. Existing models will automatically benefit from improved wall temperature and heat flux predictions. Users may observe small to moderate changes in thermal results for cases with significant surface reaction heat.

Structural Modal Analysis

CFD-ACE+ V2025.5 reintroduces the Modal Analysis capability. This feature builds on the transition to PETSc/SLEPc libraries for the solid mechanics module in earlier versions. Users can now compute mode shapes and natural frequencies with improved flexibility and accuracy.

Usage

When Modal Analysis is selected under **MO** → **Stress** → **Analysis** → **Linear**, the interface provides additional options under **SC** → **Adv** for configuring the eigenvalue solver. Users can specify the number of modes (nev) to compute and control key parameters such as the number of column vectors, solver type, iteration count, and convergence tolerance. Column vectors (ncv) can be either code-calculated or user-specified, but must satisfy the condition ncv ≥ nev. For reliable convergence, typical values should be two to four times the number of requested eigenvalues. The GUI enforces minimum column vector counts for 3D cases to ensure stability.

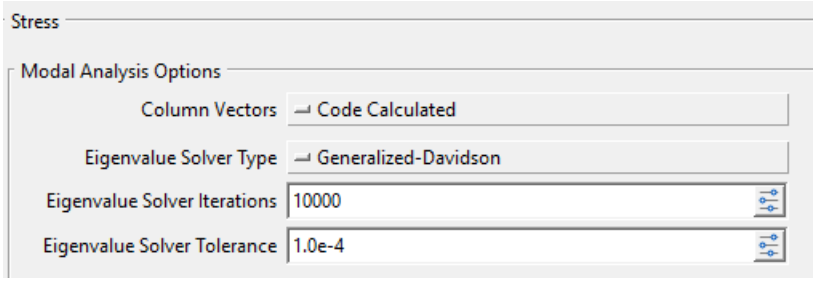


Figure 16. Modal Analysis Options

Four eigenvalue solvers are available: **Generalized-Davidson (default), Jacobi-Davidson, Krylov-Schur, and Rayleigh-Quotient**. The default solver is suitable for most problems, and users are encouraged to start with Generalized-Davidson or Jacobi-Davidson before trying

the other options. Additional controls allow users to set the maximum number of iterations and the convergence tolerance. The solver stops if the specified tolerance is not met within the iteration limit. Output files include mode frequencies and solver convergence details for transparency and validation.

Limitations and Notes

To avoid divergence, users should consider increasing the column vector count or the number of iterations.

Krylov-Schur does not support second-order elements with a consistent mass matrix, though it works with lumped mass matrices.

Deprecated user-sub mode shapes are no longer supported.

Legacy scripts will either map to valid paths or return guided error messages.

Profile in Time for Voltage

In plasma processing for semiconductor manufacturing, controlling ion energy and angular distribution at the wafer surface is critical for achieving desired etch and deposition results. While sinusoidal RF signals are commonly used to sustain plasma, they do not provide fine control over ion bombardment energy. Pulsed DC biasing addresses this need by dynamically adjusting the bias level over time, reducing substrate heating, minimizing charging damage, and improving process selectivity. By superimposing a time-dependent voltage profile on a base signal, this feature enables accurate simulation of advanced techniques such as pulsed plasma etching and tailored waveform control.

CFD-ACE+ V2026.0 introduces the ability to apply an electric potential boundary condition as a time-dependent profile. This feature allows users to define voltage as a function of time using a tabular input, enabling control of electrode bias in electromagnetic and plasma simulations.

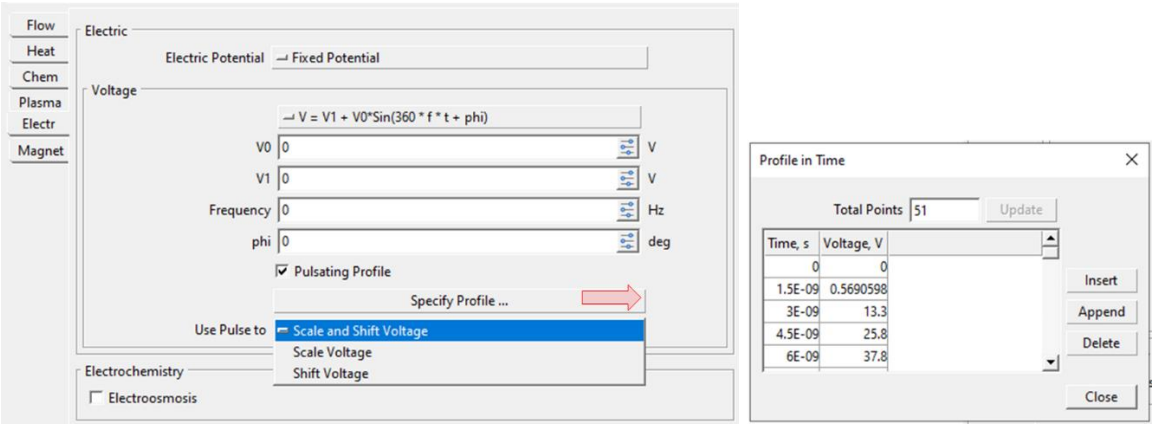


Figure 17. Pulsating Profile Specification

The GUI allows to combine a base voltage signal with a user-defined time profile. The base signal can be constant or sinusoidal, and the profile is entered through **Pulsating Profile** → **Specify Profile** option. Users can choose how the profile interacts with the base signal.

Usage

1. Select the Boundary Condition

   Navigate to **BC** → **Electric** → **Fixed Potential** for the chosen boundary.

2. Choose the Base Voltage Mode

   Set the voltage type to **Constant, Sinusoidal, or Multiple Sinusoidal Frequencies** as required for your simulation.

3. Enable Pulsating Profile

   Check the **Pulsating Profile** option under Fixed Potential to activate time-dependent voltage input.

4. Specify the Profile

   Click **Specify Profile** to open the time-voltage table.

5. Enter or paste data starting from 0 seconds.

   To import from a CSV or text file, first set the Total Points, click Update, then paste the data.

6. Apply Settings
7. After completing the profile, close the table and apply the configuration.
8. Combine with Base Voltage

   Use the Use Pulse to menu to select how the profile interacts with the base signal:

   Shift Voltage: Adds the profile as an offset.

   Scale Voltage: Modulates the amplitude of the base waveform.

   Scale and Shift Voltage: Applies both operations for complex waveform shaping.

Limitations

Restarting from another DTF requires enabling Reset time to 0 Second in the IC tab.

Improvements to Monte Carlo Plasma

This release brings significant improvements to the Monte Carlo Plasma solver, focusing on accuracy, numerical robustness, and diagnostic flexibility. The enhancements improve particle motion fidelity, surface interaction modeling, initialization consistency, and trajectory logging options.

Physically Accurate Particle Motion

The solver now accounts for local electric field acceleration when calculating the time for a particle to reach a cell face. This replaces older constant-velocity approximations and ensures that particle trajectories reflect realistic physics, particularly in high-field regions such as plasma sheaths. This removes non-physical energy spikes and face penetration issues. To complement this, two additional controls improve motion stability:

- Fractional Move Limit: Each sub-step is capped at a fraction of the local cell size (default 5%). This prevents overshooting in steep gradients and improves spatial resolution.
- Move Factor: A conservative scaling factor (default 0.5) is applied to the chosen time-step limiter, reducing numerical overshoot and improving robustness.

These changes produce cleaner energy distributions and reliable wall flux predictions. While runtime may increase slightly due to more sub-steps, the trade-off is improved physical fidelity.

Accurate Handling of Face Crossing and Wall Hits

When particles cross multiple faces in a single time-step, the solver must "re-bin" them to the correct cell. The previous approach relocated particles to the neighbor cell center, which could distort impact angles and positions at surfaces. The new default method relocates particles near the actual intersection point along their trajectory, preserving true geometry.

Similarly, when particles hit walls, a new relocation method places them just inside the cell at the intersection point rather than using arbitrary offsets. This ensures accurate surface interaction modeling for sputtering, collection, and secondary emission.

These methods ensure correct incidence angles and positions, improving surface physics accuracy.

Improved Particle Initialization

For axisymmetric cases, particles are now initialized with uniformly distributed angles (0 to 2π) and positions scattered within cells. Velocities follow a Maxwellian distribution. This ensures physically consistent starting conditions.

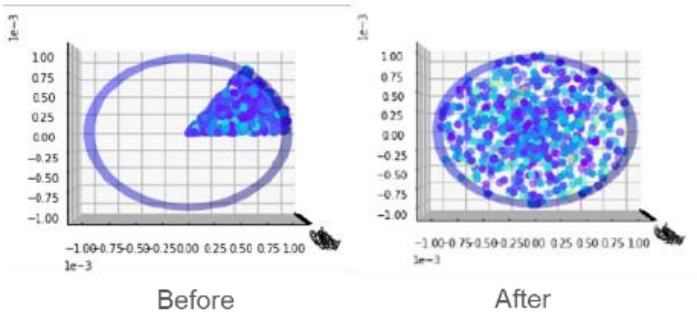


Figure 18. Particle Initialization

Angle Preservation for Polar Motion

For axisymmetric geometries, a new motion algorithm integrates particle trajectories in radial, angular, and axial components while preserving the angular coordinate (θ). Optional logging of angle information provides deeper insight into particle dynamics.

Correct Angle Handling for Charge-Exchange Products

Previously, particles created through charge-exchange reactions defaulted to an angle of zero, skewing angular statistics. The solver now copies the parent particle's angle or computes it from the new position using the ACE+ cylindrical convention.

Transparent Particle Accounting

A new counter tracks particles that could not be added due to particle tracking limits. Simulation statistics now report: generated = added + not_added. This transparency helps users diagnose discrepancies and adjust particle tracking limit or particle creation goals for large-scale runs.

time spent tracking mcparticles so far = 1.0000E-04
number of mcparticles still to be tracked = 223

specie	created	collected	below_Emin	clean_Allpc	lost	removed	% done	not_added
AR	457	0	0	234	0	234	51	168
AR+	400	399	0	1	0	400	100	0
AR*	80	80	0	0	0	80	100	0
totals	937	479	0	235	0	714	83	168

Figure 19. Particle Statistics

Particle Trajectory

Users can now write out particle trajectories for visualization and other diagnostics purposes. This can be enabled by using the special DTF Update **'mcplas_write_particle_info int 1 1'**. Solver writes a file with the extension mc.pc.info, which contains detailed particle location and velocity information. Note that these files can become very large. This option is provided primarily for diagnosis using smaller number of particles.

```
#########################################################################################################################
#
#  MC Particle Information
#
#  2D polar geometry (angle included) with Cartesian format:
#  time [sec], x [m], y [m], z [m], vx [m/s], vy [m/s], vz [m/s], KE [eV]
#
#
#########################################################################################################################
MODEL:2D_polar
NUMVAR:8
ITERATION:1
COUNTSTEPS:1
move_for_this_TimePeriod:1.000000000000000E-004
START_MOVING:AR+
ID:1
    0.00000E+00    0.40619E-04    0.66871E-03    0.15370E-03   -0.34962E+02    0.17301E+03   -0.54560E+03    0.68073E-01
    0.35350E-07    0.35610E-04    0.67482E-03    0.13441E-03   -0.24842E+03    0.17301E+03   -0.54560E+03    0.80595E-01
    0.52094E-07    0.30604E-04    0.67772E-03    0.12528E-03   -0.34952E+03    0.17301E+03   -0.54560E+03    0.93110E-01
    0.64980E-07    0.25599E-04    0.67995E-03    0.11825E-03   -0.42734E+03    0.17301E+03   -0.54560E+03    0.10562E+00
```

Figure 20. Particle Trajectory

Electroplating Parallelization

ACE+ 2026.0 introduces parallel execution for the Electroplating module, enabling multi-core scaling for large and complex plating simulations. The implementation was tested on representative cases and demonstrated consistent results compared to serial runs, with significant reductions in wall-clock time.

Other Bug Fixes and Improvements

- Users can now apply Profile in Time with ramp-up or ramp-down, including step changes, using the Standard time-stepping scheme. Previously, this capability was limited to Auto time step, making it useful for time-dependent mixtures and voltage boundary conditions.
- Resolved an issue where the Z-component of wall shear stress was incorrectly reported, often duplicating the X-component value. The calculation now correctly reflects the Z-direction shear stress.
- Fixed a solver crash when using 'Profile X' for Temperature together with 'Profile from file' for mixture inlet boundaries.
- Fixed an issue where parametric updates for volume reaction Arrhenius parameters (Ap, Ea/R) were not applied during parametric studies, causing all cases to use original database values. The solver now correctly updates and applies these parameters across cases. Added limited backward compatibility for mixed step types (non-Arrhenius before Arrhenius) to prevent corrupted data and ensure correct parameter handling after save and reopen.
- Fixed a regression where surface reaction parameters failed to load; parameters now load correctly.
- Fixed a regression in transient thinwall cases where multilayer/default models produced inaccurate temperatures versus resolved coatings. Corrections to the interface coupling between multilayer thinwalls and the meshed domain produces results that align with earlier versions and resolved-layer solutions.
- Fixed a spray parcel tracking issue where droplets exiting the domain were not detected at certain boundary faces, causing parcels to continue updating outside the mesh. The tracking algorithms have been improved to terminate parcels correctly. Note that the fix may increase search time in some cases with bad quality mesh near boundaries.
- Resolved a crash occurring in parallel runs for spray simulations. The crash was caused by an error during MPI exchange, where the neighbor process failed to assign a valid cell index to the spray drop. This typically happens when the only available cell at the cell location (on a parallel boundary) is skewed.

- Addressed a crash in parallel runs for VOF simulations involving arbitrary interfaces. This issue has been present in all previous versions and is now addressed. While the crash has been fixed, the limitation of switching to SLIC from PLIC still remains and thus surface tension will not be accounted for in the polyhedral cells formed across the arbitrary interface.
- Fixed a Scalar module issue where non-mass-specific scalars ('solve for φ') produced large flux imbalance between inlet and outlet surfaces. The delta-form discretization and flux summary calculations have been corrected. Results now match the behavior of mass-specific scalars.
- Fixed a divergence issue in Full Stefan–Maxwell multi-component diffusion cases. The solver now handles diffusion coefficients correctly, eliminating negative values and restoring stable convergence.
- Resolved an issue in CCP simulations where blank lines in SPICE circuit output caused solver crashes on RHEL8 and RHEL9. The READ logic has been updated to handle empty lines correctly, ensuring consistent behavior across Linux and Windows platforms.
- Fixed an issue where Auto Time Step could generate negative dt values in shared time step size with certain CCP models. The solver now respects minDT settings to ensure stability and proper profile capture. Users should select minDT carefully based on minimum intervals in profile specifications.
- CFD-SOLVER now defaults to using a single thread for runs submitted through supported schedulers such as Slurm. This change ensures consistent performance on cluster environments where multi-threading can introduce overhead. The -maxThreads option has been added to the solver help menu and can be used to set the number of threads for a run. This option overrides the OMP_NUM_THREADS environment variable, allowing users to control threading behavior directly from the command line.
- The maximum number of rows in profile tables (such as Profile X, Y, Z, 2D, and Profile in Time) is no longer fixed at 1000 when data is pasted from external sources like Excel. The table now updates the total points to match the pasted data, even if it exceeds 1000. This ensures that all pasted values are saved correctly in the DTF file. Additionally, the maximum limit for direct entry has been increased to allow larger datasets.
- When Radiation is enabled with SnDOM or CAFVM and Non-Gray modeling is selected, the wavelength band table previously initialized all entries to zero due to a regression introduced in an earlier revision. This caused validation checks to prevent users from entering values in a natural order. The issue has been fixed, and the table now restores the original behavior.
- Changes to boundary-condition objects (VCs, BCs, and ICs) were already journaled only on Accept, ensuring that pressing Reset did not record intermediate edits.

Entities such as Injectors, Fans, VRs, Macro Particles, MRFs, and Visualization Objects now follow the same behavior. Their edits are buffered and written to the journal only when Accept is pressed, so pressing Reset no longer produces incorrect or partial journal entries. This update makes journaling consistent across all object types and prevents unintended commands in saved journals.

- The BC Explorer now displays boundary conditions with their attached volume types—for example, “Fluid Wall,” “Fluid–Fluid Interface,” “Fluid Inlet,” and “Block Outlet”—instead of only the generic BC type. This makes it clear whether an Inlet, Outlet, or Wall is applied to a Fluid or Block volume. GUI scripting has been updated to honor these qualified types, so commands like GetGeomObjByType("Fluid Wall") return the expected objects. For backward compatibility, unqualified simple types such as "Wall", "Inlet", and "Outlet" continue to work as before.
- The BC Explorer now updates the BC Type column for Interfaces whenever the attached volume type changes. Previously, the detailed type information (such as “Fluid–Solid Interface”) was only refreshed after saving and reloading the model or toggling the interface type.

TOPO

CFD-TOPO V2026.0 includes the following new features and enhancements. Please refer to the *ACE+ SUITE 2026.0 CFD-TOPO User's Guide* for further details.

Flux Profile Cycles Option

This release adds a new capability to simplify flux input for cyclic processes. When using the Profile in Time option, users can now specify the number of cycles for which the defined flux profile should repeat by selecting “Perform <n> Cycles” in the GUI.

This enhancement eliminates the need to manually enter large sets of flux vs. time data for repeated patterns, reducing user effort and minimizing input errors. It is particularly useful for processes involving multiple identical cycles, such as atomic layer deposition (ALD).

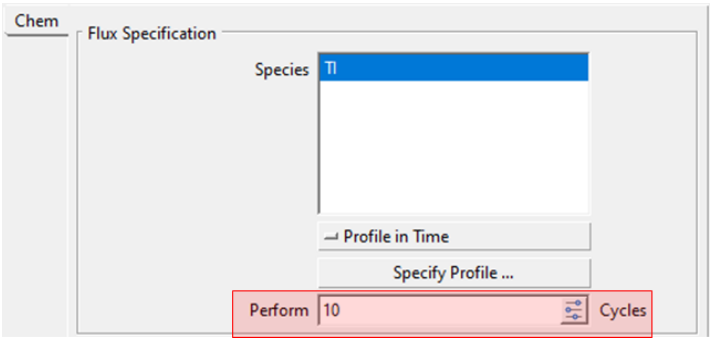


Figure 21. Flux Profile Cycles

Reactant Order Restriction Removed

In previous versions, reactants had to be entered in a strict sequence—gas-phase species first, followed by surface species, and then bulk species. If this order was not followed, the solver could produce incorrect results. This requirement has been removed now. Users can specify reactants in any order. The solver automatically arranges them internally to ensure accurate calculations and provides a message in the output file confirming the reordering.

```
Note: The reactants for step 1 of reaction SiN_etch were reordered.
The ordering used for reactants and products is: gas, bulk, surface

    The reaction as specified in the GUI:
    1 SIN(B) + 1 F+ => SINF

    The new reordered reaction:
    1 F+ + 1 SIN(B) => SINF
```

Figure 22. Reordering of Reactants by Solver

Chemistry CFL Time Scale Correction

This release corrects the computation of the timestep used when Apply "CFL Limit for chemistry" is enabled. The chemistry time scale is now calculated as the site density divided by the net production rate, providing a physically consistent limit on the time step for feature-scale ALD scenarios. In previous versions, the numerator incorrectly used site density multiplied by site fraction, and the minimum across species was taken. In previous versions, if a species had zero site fraction, the resulting time scale approached zero, forcing extremely small timesteps and, in some cases, repeated creation of level-set fields at each step. With this correction, ALD cases that are not chemistry-limited, can proceed using the configured maximum timestep, and the solver no longer exhibits unintended level-set additions.

Enhance Level Set Function Away from Interface

In CFD-TOPO, the level set method is used to track material interfaces. After solving the level set equations, the computed level set function can sometimes become less accurate in regions far from the zero level set (the actual interface). This feature introduces an option called “Reset Level Set Distances After Solution” in the SC → Adv panel. When enabled, it recalculates and improves the level set function values away from the interface, leading to better accuracy in those regions without affecting the interface itself.

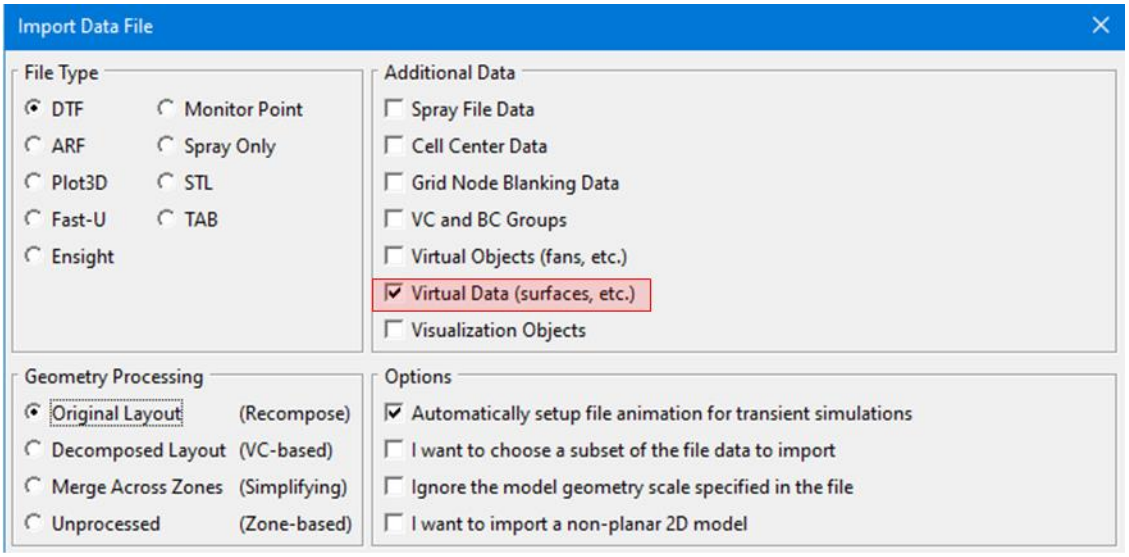
Robust Handling of Large MC Bundle Cases

In previous versions, CFD-TOPO simulations involving a very large number of Monte Carlo (MC) bundles for flux emission - such as 40,000 bundles - could run indefinitely without normal termination. This issue occurred when a solid surface met a flux inlet boundary, causing the solver to enter an infinite loop. To address this, the solver now detects when a solid surface intersects a flux inlet boundary and terminates the simulation gracefully. When this condition is encountered, detailed information is written to the output file, including the solid name and the location(s) of contact.

Material Identity Export for CFD-VIEW

This release adds the ability to export material identities to the DTF file for visualization in CFD-VIEW. Material identity at each location is determined using an explicit minimum-distance calculation to the material surfaces, providing robust assignment throughout the domain. This method improves accuracy compared to relying solely on level set values, which are more approximate away from interfaces. With material identities written to DTF, users can load CFD-VIEW and display materials as contour plots (using variable Material ID), including viewing multiple materials simultaneously.

In addition, the different material interfaces are written to the DTF as surfaces. This eliminates the need for users to create isosurfaces of different materials. Use the Virtual Data check box, to include these surfaces while importing the DTF.



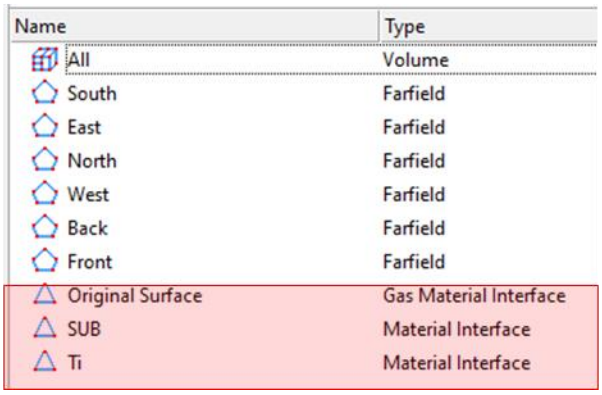


Figure 23. Import of Interfaces from TOPO DTFs

Consistent Cell-Centered Data in TOPO

This update resolves inconsistencies in cell-centered data caused by internal renumbering within TOPO. Cell-centered fields, including level set values and velocities, are now written to DTF using a mapping that aligns with CFD-VIEW's expected cell order. Corresponding changes to the DTF read functionality ensure that restart cases correctly restore data to TOPO's internal ordering.

With these changes, visualizations in CFD-VIEW display the correct cell-centered values, and restart simulations accurately reconstruct the previous solution state.

Consistent Naming for Transient DTF Files

Previously, when TOPO wrote transient DTF files, the file naming convention depended on the number of time steps. For example, a case with 100 steps produced files named Model.001.DTF through Model.100.DTF, while a case with only 5 steps produced Model.1.DTF through Model.5.DTF. This inconsistency could complicate workflows, especially when switching between cases with different time-step counts. This prevented users from using certain CFD-VIEW features while loading transient DTF files. To ensure uniformity and compatibility with ACE+, TOPO now uses a fixed-width naming format for transient DTF files. Each time-step index is written with seven digits, starting from Model.0000001.DTF and incrementing sequentially. This change provides consistent file naming across all cases and aligns TOPO with ACE+ conventions, simplifying automation and post-processing. Other changes now also enable users to automatically animate transient TOPO DTF files in CFD-VIEW.

VIEW

CFD-VIEW V2026.0 includes the following new features and enhancements. Please refer to the *ACE+ SUITE 2026.0 CFD-VIEW User's Guide* for further details.

Circumferential Average Operator

A new operator has been introduced in CFD-VIEW to calculate circumferential averages on surfaces. This operator is intended for cases where wafers or similar components rotate during processing, and an averaged representation of variables over angular positions is required.

The operator works on a single surface. If the model contains multiple surfaces that need to be averaged together, the user should first merge them into one surface using the existing Surface Merge functionality. No additional user input is required for specifying the center or axis of rotation. The operator determines these automatically by using the bounding box of the selected surface.

When executed, the operator creates a new surface. On this new surface, the circumferential average is computed for all variables present on the original surface. The averaged values are applied to the nodes of the new surface, and the result can be visualized using standard contour plotting options in VIEW. The original surface remains unchanged. The operator also allows to create radial slices on surfaces.

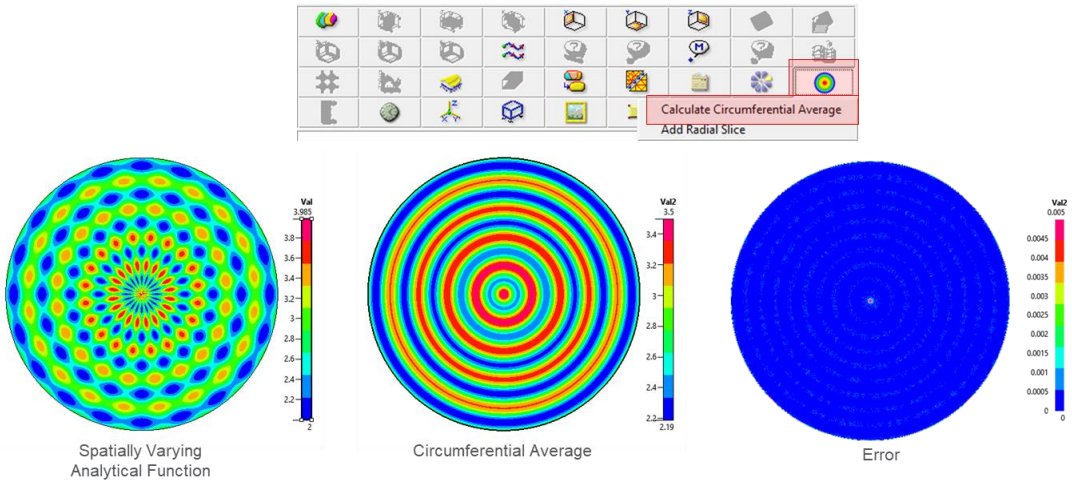


Figure 24. Circumferential Average Operator

This operator is recommended for use with mostly planar surfaces. While it will work with any surface, including complex 3D geometries, the results may not match user expectations in those cases because the center and axis of rotation are automatically determined and cannot be controlled by the user.

Faster File Dialogs

File dialogs in CFD-VIEW have been optimized to open immediately, even when navigating directories that contain thousands of files. This improvement eliminates delays that previously occurred when working with large or remote file systems.

Support for CFD-VIEW Script Arguments

CFD-VIEW now supports passing arguments to Python scripts directly from the command line. This capability enables scripts invoked by CFD-VIEW to receive user-specified parameters at startup, rather than relying on hard-coded values or interactive prompts. The implementation follows the same convention used in GUI, where each argument is preceded by the -sarg flag. Arguments are collected in the order they appear, and the script can access them via the standard sys.argv list (index 0 returns the script filename itself).

To use this feature, launch CFD-VIEW script with your preferred options and include -script <scriptname.py> followed by one -sarg for each argument you want to pass to the script. For example, a command such as:

CFD-VIEW -nogui -script view_script_arg.py -sarg inlet -sarg 2 -sarg FALSE

will result in the script receiving the three arguments ["inlet", "2", "FALSE"] in sys.argv[1:]. This functionality is available in both interactive and headless runs.

Export Surfaces to DTF

CFD-VIEW now supports exporting surfaces directly to DTF, in addition to the existing STL and VOF options. Any single surface or a collection of surfaces can be written to a new DTF file. The export records all nodal and cell attributes present on the surface, along with the information needed to reconstruct the surface in VIEW, such as the surface name and mesh metadata. The workflow remains the same: File > Save Object > and choose DTF format.

Unit System Dialog

Canceling or closing the custom units dialog no longer changes the unit system. The previous selection is retained as expected.

Improved Probe and Trace Handling

Probes, traces, and strip charts are now restricted to remain within the bounds of their input domain. This ensures that snapping and rotation operations behave consistently and that positional data remains valid.

Viewer Interaction with Strip Charts

Strip charts can now be manipulated directly within the viewer. This interaction works in the same way as probes and traces, allowing users to adjust positions without relying solely on numeric input or sliders.

Color Rendering and Colormap Enhancements

Several improvements have been made to color handling in CFD-VIEW. Color interpolation now uses the CIELab color space, resulting in smoother and more accurate gradients across colormaps. Clipping behavior for colormaps has been corrected so that values outside the specified range are handled consistently in both smooth and flooded color modes, even when the range spans very large or very small numbers. In addition, contour lines and flooded color regions now render correctly when a colormap is set to logarithmic mode, ensuring that visualizations accurately reflect the intended scale.

Point History Plotting Restored

The ability to plot multiple points in a single graph has been restored. This applies to both steady-state and transient cases, regardless of whether time data is used from the file.

Crash Fixes

Several stability issues have been resolved. These include crashes related to clipboard handling, crashes when selecting large numbers of files for animation, and crashes that occurred when saving models with plots or creating traces without initial vectors.

3050 Bowers Avenue,

Santa Clara, California 95054 United States

Tel. +1-408-727-5555

www.appliedmaterials.com