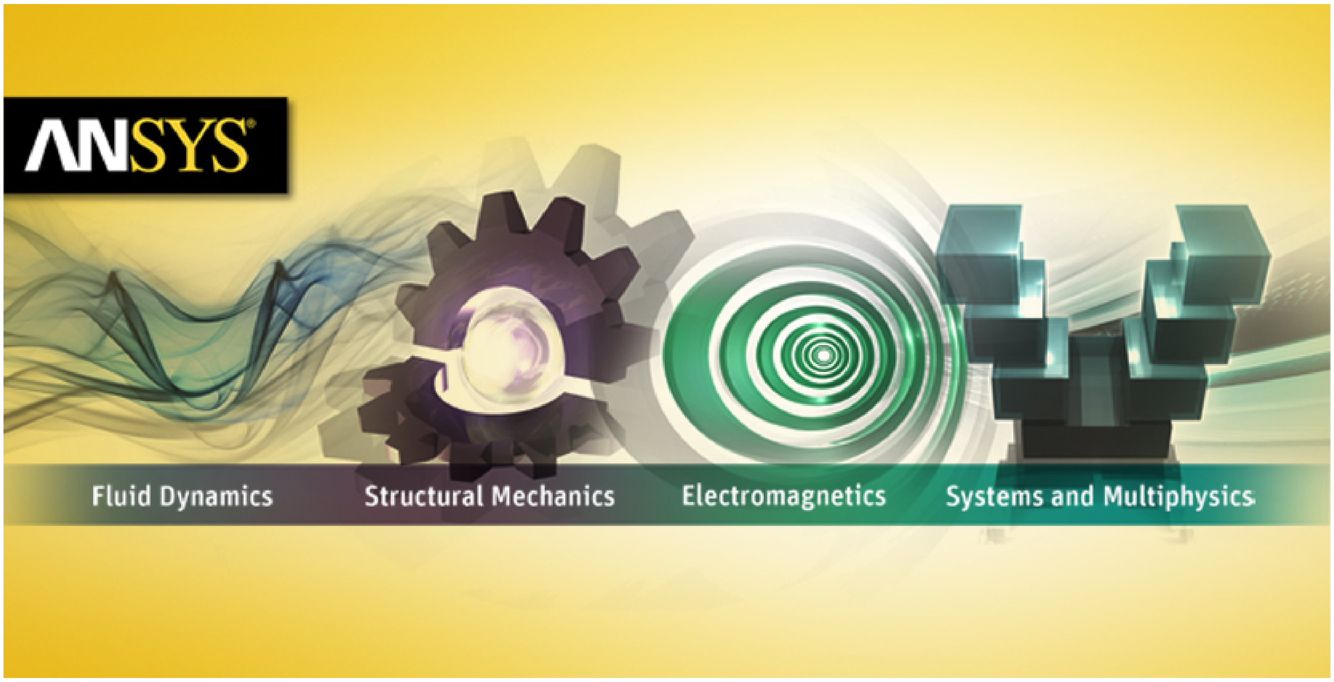

Migration Manual

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

Release 14.0
November 2011

ANSYS, Inc. is certified to ISO 9001:2008.

Published in the U.S.A.

Table of Contents

Preface

This preface is divided into the following sections:

1. The Contents of This Manual

The ANSYS FLUENT Migration Manual highlights the changes between ANSYS FLUENT 13.0 and ANSYS FLUENT 14.0. This document will include new features in ANSYS FLUENT 14.0, expected solution changes after migrating from ANSYS FLUENT 13.0, and changes in text command settings.

2. The Contents of the FLUENT Manuals

The manuals listed below form the FLUENT product documentation set. They include descriptions of the procedures, commands, and theoretical details needed to use FLUENT products.

FLUENT Getting Started Guide contains general information about getting started with using FLUENT.
FLUENT User's Guide contains detailed information about using FLUENT, including information about the user interface, reading and writing files, defining boundary conditions, setting up physical models, calculating a solution, and analyzing your results.
FLUENT in Workbench User's Guide contains information about getting started with and using FLUENT within the Workbench environment.
FLUENT Theory Guide contains reference information for how the physical models are implemented in FLUENT.
FLUENT UDF Manual contains information about writing and using user-defined functions (UDFs).
FLUENT Tutorial Guide contains a number of example problems with detailed instructions, commentary, and postprocessing of results.
FLUENT Text Command List contains a brief description of each of the commands in FLUENT's text interface.
FLUENT Adjoint Solver Module Manual contains information about the background and usage of FLUENT's Adjoint Solver Module that allows you to obtain detailed sensitivity data for the performance of a fluid system.
FLUENT Battery Module Manual contains information about the background and usage of FLUENT's Battery Module that allows you to analyze the behavior of electric batteries.
FLUENT Continuous Fiber Module Manual contains information about the background and usage of FLUENT's Continuous Fiber Module that allows you to analyze the behavior of fiber flow, fiber properties, and coupling between fibers and the surrounding fluid due to the strong interaction that exists between the fibers and the surrounding gas.
FLUENT Fuel Cell Modules Manual contains information about the background and the usage of two separate add-on fuel cell models for FLUENT that allow you to model polymer electrolyte membrane fuel cells (PEMFC), solid oxide fuel cells (SOFC), and electrolysis with FLUENT.
FLUENT Magnetohydrodynamics (MHD) Module Manual contains information about the background and usage of FLUENT's Magnetohydrodynamics (MHD) Module that allows you to analyze the behavior of electrically conducting fluid flow under the influence of constant (DC) or oscillating (AC) electromagnetic fields.

FLUENT Migration Manual contains information about transitioning from the previous release of FLUENT, including details about new features, solution changes, and text command list changes.
FLUENT Population Balance Module Manual contains information about the background and usage of FLUENT's Population Balance Module that allows you to analyze multiphase flows involving size distributions where particle population (as well as momentum, mass, and energy) require a balance equation.
Running FLUENT Under LSF contains information about the using FLUENT with Platform Computing's LSF software, a distributed computing resource management tool.
Running FLUENT Under PBS Professional contains information about the using FLUENT with Altair PBS Professional, an open workload management tool for local and distributed environments.
Running FLUENT Under SGE contains information about the using FLUENT with Sun Grid Engine (SGE) software, a distributed computing resource management tool.

3. Technical Support

If you encounter difficulties while using ANSYS FLUENT, please first refer to the section(s) of the manual containing information on the commands you are trying to use or the type of problem you are trying to solve. The product documentation is available from the online help, or from the ANSYS Customer Portal (www.ansys.com/customerportal).

If you encounter an error, please write down the exact error message that appeared and note as much information as you can about what you were doing in ANSYS FLUENT.

Technical Support for ANSYS, Inc. products is provided either by ANSYS, Inc. directly or by one of our certified ANSYS Support Providers. Please check with the ANSYS Support Coordinator (ASC) at your company to determine who provides support for your company, or go to www.ansys.com and select **About ANSYS> Contacts and Locations**. The direct URL is: http://www1.ansys.com/customer/public/supportlist.asp. Follow the on-screen instructions to obtain your support provider contact information. You will need your customer number. If you don't know your customer number, contact the ASC at your company.

If your support is provided by ANSYS, Inc. directly, Technical Support can be accessed quickly and efficiently from the ANSYS Customer Portal, which is available from the ANSYS Website (www.ansys.com) under **Support> Technical Support** where the Customer Portal is located. The direct URL is: http://www.ansys.com/customerportal.

One of the many useful features of the Customer Portal is the Knowledge Resources Search, which can be found on the Home page of the Customer Portal.

Systems and installation Knowledge Resources are easily accessible via the Customer Portal by using the following keywords in the search box: `Systems/Installation`. These Knowledge Resources provide solutions and guidance on how to resolve installation and licensing issues quickly.

NORTH AMERICA

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Toll-Free Telephone: 1.800.711.7199

Fax: 1.724.514.5096

Support for University customers is provided only through the ANSYS Customer Portal.

GERMANY

ANSYS Mechanical Products

Telephone: +49 (0) 8092 7005-55

Email: support@cadfem.de

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

National Toll-Free Telephone:

German language: 0800 181 8499

English language: 0800 181 1565

International Telephone:

German language: +49 6151 3644 300

English language: +49 6151 3644 400

Email: support-germany@ansys.com

UNITED KINGDOM

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +44 (0) 870 142 0300

Fax: +44 (0) 870 142 0302

Email: support-uk@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

JAPAN

CFX , ICEM CFD and Mechanical Products

Telephone: +81-3-5324-8333

Fax: +81-3-5324-7308

Email: *CFX:* japan-cfx-support@ansys.com; *Mechanical:* japan-ansys-support@ansys.com

FLUENT Products

Telephone: +81-3-5324-7305

Email: *FLUENT:* japan-fluent-support@ansys.com; *POLYFLOW:* japan-polyflow-support@ansys.com; *FfC:* japan-ffc-support@ansys.com; *FloWizard:* japan-flowizard-support@ansys.com

Icepak

Telephone: +81-3-5324-7444

Email: japan-icepak-support@ansys.com

Licensing and Installation

Email: japan-license-support@ansys.com

INDIA

ANSYS Products (including FLUENT, CFX, ICEM-CFD)

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +91 1 800 233 3475 (toll free) or +91 1 800 209 3475 (toll free)

Fax: +91 80 2529 1271

Email: *FEA products:* feasup-india@ansys.com; *CFD products:* cfdsup-india@ansys.com; *Installation:* installation-india@ansys.com

FRANCE

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Toll-Free Telephone: +33 (0) 800 919 225

Email: support-france@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

BELGIUM

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +32 (0) 10 45 28 61

Email: support-belgium@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SWEDEN

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +44 (0) 870 142 0300

Email: support-sweden@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SPAIN and PORTUGAL

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +33 1 30 60 15 63

Email: support-spain@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

ITALY

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +39 02 89013378

Email: support-italy@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

Chapter 1: Migrating to ANSYS FLUENT 14.0

The purpose of the ANSYS FLUENT Migration Manual is to help you transition from ANSYS FLUENT 13.0 to ANSYS FLUENT 14.0. Please read through the entire document to understand the changes that have taken place. The information is enclosed in the following chapters:

Please visit the ANSYS Customer Portal (`www.ansys.com/customerportal`) to obtain a list of Known Issues and Limitations. Click the Documentation link to access this document.

For a list of platform/OS levels that are supported in the current release, please visit the ANSYS website.

Please 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 the Release 13.0 beta features and the Release 14.0 standard FLUENT features that concern you, please contact technical support for assistance. Your feedback is appreciated and will help us improve the overall quality of the product.

Chapter 2: New Features in ANSYS FLUENT 14.0

New features available in ANSYS FLUENT 14.0 are listed below. References to the appropriate section in the User's Guide is provided for each new feature (unless otherwise noted).

Solver-Numerics

- Second order advection scheme is the default setting for all models, except for the mixture and Eulerian multiphase flows, which will remain first order by default
- Hybrid initialization method as default with enhanced initialization option settings (Steps in Using Hybrid Initialization)
- Convergence acceleration available for meshes containing highly stretched cells for the implicit density based solver (Convergence Acceleration for Stretched Meshes (CASM))
- High order term relaxation available when applying higher order spatial discretization (High Order Term Relaxation (HOTR))
- Preconditioned conjugate gradient method (CG) available as a stabilization method for the AMG linear equation solver (Setting the AMG Method and the Stabilization Method)
- Modifications to the expert settings for the pseudo transient method. Note that the old case settings for the pseudo transient method in the **Expert** tab of the **Advanced Solution Controls** dialog box are now obsolete and no backward compatibility is provided. Please update case files using FLUENT 14.0. (Setting Solution Controls for the Pseudo Transient Method)

Solver-Meshing

- Remeshing
  - Option to preserve interior surfaces for postprocessing following polyhedral mesh conversion via a TUI command (Converting the Domain to a Polyhedra)
  - Ability to switch from hanging node mesh representation to polyhedral mesh representation via a TUI command (Converting Cells with Hanging Nodes / Edges to Polyhedra)
  - Ability to remesh 3D wedge/prism cells in a boundary layer mesh as part of cell zone and face region remeshing methods (Cell Zone Remeshing Method and Face Region Remeshing with Prism Layers)
  - Ability to print the poor element statistics in the console via the **Solution Methods** task page (Repairing Meshes and Robustness on Meshes of Poor Quality)
  - Ability to automatically convert the cells that have hanging nodes / edges as a result of the CutCell zone remeshing to polyhedral cells (Using the CutCell Zone Remeshing Method)
- Dynamic Meshes
  - Ability to include polyhedral cells in dynamic mesh problems (Limitations)
  - Ability to specify that the diffusion coefficient is a function of the cell volume, when diffusion-based smoothing is used to update a dynamic mesh (Diffusivity Based on Cell Volume)
  - Ability to specify a piston pin offset for in-cylinder dynamic mesh applications (In-Cylinder Settings).

- Moving Meshes
  - Automatic calculation of rotational axis origin for nested sliding mesh reference frames
  - Ability to associate zone specific boundary motion with data from system couplings

Models

- Turbulence
  - Compatibility of the Spalart-Allmaras turbulence model with enhanced wall treatment (Spalart-Allmaras One-Equation Model)
  - Curvature correction available, but not applicable for 2d axisymmetric geometries (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models)
  - Algebraic Wall-Modeled LES available for the subgrid-scale models (Algebraic Wall-Modeled LES Model (WMLES) in the Theory Guide)
  - The implementation of the Delayed DES (DDES) shielding function, f_d (Equation 4–228 in the Theory Guide), has been optimized in the SST and Realizable k-ε Detached Eddy Simulation (DES) models to provide effective shielding. The constant was changed from 8 to 20. With this change, DDES is now the recommended shielding function for the SST k-omega model with Delayed DES enabled and is used by default.
- Heat Transfer
  - Ability to model heat transfer in porous media without the assumption of thermal equilibrium between the media and the fluid flow, via a dual cell approach (Non-Equilibrium Thermal Model)
  - Ability to create a duplicate mesh for a single fluid zone directly in FLUENT, e.g. when setting up a dual cell heat exchanger (Copying Cell Zones)
- Finite-Rate Chemistry Model
  - Ability to set the surface reaction parameters for the Non-Equilibrium Thermal Model using the `define/models/species/surf-reaction-netm-param` text command.
  - Ability to model chemically activated bimolecular pressure dependent reaction types (Inputs for Reaction Definition)
- Partially Premixed Combustion Model
  - Ability for internal combustion engines to convert products at the end of one cycle to inert for the next cycle when using the partially premixed combustion model (Modeling In Cylinder Combustion)
  - Ability to include the effects of heat loss or gain in the unburnt mixture, as well as equivalence-ratio fluctuations, on the laminar flame speed (Laminar Flame Speed in the Theory Guide)
- Reacting Channel Model
  - Ability to efficiently solve reacting flow in shell and tube heat exchangers (including curvilinear configurations) with long and thin channels (Reacting Channel Model)
- Solidification and Melting Model
  - Thermal and solutal buoyancy options available as full features (beta features in Release 13) (Modeling Thermal and Solutal Buoyancy)
- Discrete Phase Model
  - Stochastic secondary droplet (SSD) model available as full feature (beta feature in Release 13) (Modeling Spray Breakup)

- Discrete Element Method (DEM) available as full feature (beta feature in Release 13) (Modeling Collision Using the DEM Model)
- Implementation of a boiling rate equation for multicomponent particles to be able to simulate multicomponent vaporization when the total vapor pressure at the droplet surface exceeds the cell pressure
- Improvements for handling particle interactions with moving walls for general meshes
- Extension visualization of particle data, including filtering of particle tracks, sizing of particle spheres with any particle variable, and displaying DEM specific data to understand the particle physics (Specifying Particles for Display and Particle Filtering)
- Ability to control the coupled heat-mass solution of droplets and multicomponent particles (Including Coupled Heat-Mass Solution Effects on the Particles) and to include vaporization options (Enabling Pressure Dependent Boiling and Including the Effect of Droplet Temperature on Latent Heat)

• VOF
  - Ability to model surface tension using continuum surface stress method (beta feature in Release 13) (Including Surface Tension and Adhesion Effects)
  - Coupled with volume fractions option for solving equations (Coupled Solution for VOF and Mixture Multiphase Flows, Selecting the Pressure-Velocity Coupling Method, and Controlling the Volume Fraction Coupled Solution)

• Eulerian Multiphase Model
  - Critical heat flux for wall boiling models available as full feature (beta feature in Release 13), including boiling model parameters (Including the Boiling Model)
  - Yao and Morel extension of the volumetric interfacial area transport model to include mass transfer and nucleation effects (beta feature in Release 13) (Defining the Interfacial Area Concentration)
  - Two new drag functions are available for granular flow: the Huilin and Gidaspow drag law and the Gibilaro drag law (Specifying the Drag Function)
  - The Immiscible Fluid Model from previous releases of ANSYS FLUENT has been renamed to Multi-Fluid VoF Model.
  - The Full Multiphase Coupled pressure-velocity coupling scheme from previous releases of ANSYS FLUENT has been renamed to Coupled with Volume Fractions and is now selected by choosing **Coupled** in the Solution Methods task page and enabling the **Coupled with Volume Fractions** option (Selecting the Pressure-Velocity Coupling Method).

• Eulerian Wall Film Model
  - Eulerian wall film model available as full feature (beta feature in Release 13) ("Modeling Eulerian Wall Films")
  - Heat transfer support for the Eulerian wall film model ("Modeling Eulerian Wall Films")

• Population Balance
  - Ability to include growth and nucleation phenomena for the Inhomogeneous Discrete population balance model
  - Availability of the DQMOM method in serial only (beta feature in Release 13) (Enabling the Population Balance Model)

• Acoustics

  - Ability to use the Ffowcs-Williams and Hawkings model to include convective effects (The Ffowcs-Williams and Hawkings Model in the Theory Guide) and specify the locations of moving receivers (Specifying Acoustic Receivers)

Material Properties

- Convection/diffusion controlled vaporization for droplets (Spalding mass transfer)
- Urea material extended to include droplet, particle mixture (urea-water) and mixture (urea-water-air) materials
- film-averaged temperature used for binary diffusivity of vaporizing droplets (Description of the Properties)

Mesh Morpher/Optimizer

- Ability to define the objective function that is minimized by the mesh morpher/optimizer as a custom function of output parameters, i.e., values from flux, force, surface integral, or volume integral reports (Setting Up the Mesh Morpher/Optimizer)
- Ability to define constraints on the boundary zones, in order to limit the freedom of particular zones that fall within the deformation region(s) during the morphing of the mesh (Setting Up the Mesh Morpher/Optimizer)
- Ability to specify commands that are executed before or after the calculation is run for each design stage generated by the mesh morpher/optimizer (Setting Up the Mesh Morpher/Optimizer)

Parallel Processing

- Improved distributed/shared memory hybrid AMG algorithm leading to significant improvements in solver scalability.
- Architecture-aware partitioning has been improved and is performed by default when the case file is read (Partitioning in the User's Guide).
- Ability to extend exterior cell creation based on interface face and node coverage (Extended Neighborhood in the UDF Manual).
- Ability to use Laplacian-coefficient-based AMG coarsening to partition cases with highly stretched cells (Partition Methods in the User's Guide)
- FLUENT now makes use of Platform MPI technology (formerly referred to as HP-MPI) from Platform Computing Corporation ("Parallel Processing")
- Support for PBS Professional in interactive mode (Starting ANSYS FLUENT Using FLUENT Launcher in the User's Guide).
- Changes to supported platforms. (refer to the updated tables in "Parallel Processing").
- Increased performance of view factor calculations utilizing the GPGPU hardware (beta feature).
- Enable FLUENT UDFs to execute on GPUS (beta feature).

User-Defined Functions (UDFs) and User-Defined Scalars (UDSs)

- UDF access to model boiling parameters and quenching correction (DEFINE_BOILING_PROPERTY in the UDF Manual)
- Linearized mass transfer UDF to model mass transfer in multiphase flows (beta feature in Release 13) (DEFINE_LINEARIZED_MASS_TRANSFER in the UDF Manual)

- UDF access to customize the variables in the PDF look-up table (DEFINE_PDF_TABLE in the UDF Manual)

Data Import and Export

- Ability to export solution data from select cell zone(s) to ANSYS CFD-Post, EnSight Case Gold, or FieldView formats (Exporting Solution Data after a Calculation)
- Ability to export a `.cdat` file for CFD-Post without also writing a case (`.cas`) file (ANSYS CFD-Post-Compatible Files and Exporting to ANSYS CFD-Post)
- Ability to export state (`.cst`) files , so that you can use CFD-Post to view most of the types of postprocessing surfaces created within FLUENT (e.g., isosurfaces) (ANSYS CFD-Post-Compatible Files and Exporting to ANSYS CFD-Post)

Graphics, Postprocessing, and Reporting

- Improved parallel simulation performance when using monitors
- Ability to calculate and postprocess time-averaged custom field functions
- Ability to display the time period over which data has been sampled for the postprocessing of the mean and RMS values
- Ability to set up multiple monitors in a single case for each one of the following: drag, lift, and moment (Setting Up Force and Moment Coefficient Monitors and Defining an Animation Sequence)
- Ability to plot and/or record how the objective function varies with each design stage when using the mesh morpher/optimizer (Setting Up the Mesh Morpher/Optimizer)
- Ability to postprocess the external temperature (shell), i.e., the temperature on the surface of a shell conduction wall that is away from the adjacent fluid/solid cell zone (Alphabetical Listing of Field Variables and Their Definitions)
- Ability to monitor and compute the uniformity index (weighted by area or mass) of a specified quantity over selected surfaces (Overview of Defining Surface Monitors and Generating a Surface Integral Report)
- The default settings for the **Save Picture** dialog box have been changed to save a color-scale copy of the picture in a JPEG format (Using the Save Picture Dialog Box)

User Interface

- Ability to set boundary conditions of same type using wildcards

Workbench

- Ability to perform one-way or two-way coupling with FLUENT and Ansoft products (Maxwell) (Performing FLUENT and Ansoft Coupling in Workbench)
- Output parameter support for drag, lift, and moments (Creating Output Parameters in the User's Guide).
- Automatic compilation of UDF libraries by FLUENT ("Compiling UDFs" in the UDF Manual).
- Source term parameters no longer need to only be specified using SI units (FLUENT in Workbench User's Guide).
- New text user interface commands (/solve/set/number-of-iterations; /solve/set/number-of-time-steps; and /solve/set/max-iterations-per-time-step) to set the number of iterations or time-steps (applicable to FLUENT in Workbench) (FLUENT Text Command List).

Add-Ons

- Ability to extend a CFD analysis with detailed sensitivity data using the FLUENT Adjoint Model add-on (FLUENT Adjoint Solver Module Manual).
- Ability to perform battery modeling using FLUENT Battery Model add-on (FLUENT Battery Module Manual).

Chapter 3: Solution Changes in ANSYS FLUENT

The sections in this chapter contain a comprehensive list of the code changes implemented in ANSYS FLUENT 14 which may affect the ANSYS FLUENT 13 solutions.

Please note that text that is in bold font represents key words that may facilitate your search for the changes in code behavior.

Solver-Numerics

- Change to **second order spatial discretization as the default method** for the pressure based solver.
  - The second order discretization scheme will provide improved solutions compared to the first order scheme used in previous releases. However, cases may take more iterations to converge and/or need changes to the solver settings for optimal convergence.
  - Previously setup cases are not affected and will retain the old default. New cases will use the updated default method.
- **Change in default method of boundary limiting**.
  - The new default boundary gradient limiting procedure improves solutions, particularly for cases with coarse meshes near boundaries. It also improves convergence by avoiding out of bound values during iterations. To revert to pre-FLUENT 14 code behavior, use the following rpvar command:

    ```
    (rpsetvar ‘recon/bc-minmax-id-new 1)
    ```

Solver-Meshing

- Several **dynamic mesh** algorithms related to remeshing and smoothing have been improved. These changes can result in slightly different meshes for dynamic mesh simulations that can effect the solution.
- The **polyhedra conversion** algorithm has been improved. Using the same mesh as a starting mesh, the polyhedra conversion might produce a slightly different polyhedra mesh.
- The **quality based mesh smoothing** (in the **Smooth/Swap** menu) has been improved and might return meshes of better quality.

Turbulence

- The new **default near-wall treatment for the Spalart-Allmaras turbulence model** is now the enhanced wall treatment with the Low-Re damping option enabled. The Low-Re damping option has been removed from the GUI. To revert to FLUENT 13 settings, first turn off the enhanced wall treatment for the Spalart-Allmaras model via the `/define/models/viscous> sa-enhanced-wall-treatment?` text command.

  A new text command is then available that allows you to turn the Low-Re Damping on or off: `/define/models/viscous> sa-damping?`
- Improvements have been made to scale-resolving turbulence simulations employing an underlying one- or two-equation RANS model (i.e. SAS or DES) and using a synthetic turbulence generator at an inlet or at a RANS/LES interface. Results may vary from previous releases.

- **Rough wall treatment has been improved for epsilon-equation based turbulence models** to avoid reduction in effective roughness when the near-wall mesh is refined. This is the new default treatment. Set the following rpvar command to `false` to return to pre-FLUENT 14 code behavior.

  ```
  (ke-rough-wall-treatment-r14? #f)
  ```

- The implementation of the **Delayed DES** (DDES) **shielding function**, f_d (Equation 4–228 in the Theory Guide), has been optimized in the SST and Realizable k-ε Detached Eddy Simulation (DES) models to provide effective shielding. The constant was changed from 8 to 20. With this change, DDES is now the recommended shielding function for the SST k-omega model with Delayed DES enabled and is used by default.
- The calculation of SAS-specific terms at periodic boundary conditions has been corrected and will yield improved model behavior.

Heat Transfer

- For the **shell conduction model at T-junctions** formed with 2 walls, the heat-conduction treatment has been corrected and will yield improved results.
- **Postprocessing Wall Function Heat Transfer Coefficient (WFHTC) has been corrected**. FLUENT no longer reports a value of zero for WFHTC on adiabatic walls. The previous behavior can be recovered with the following rpvar command.

  ```
  (rpsetvar 'wf/zero-wfhtc-on-adiabatic-walls? #t)
  ```

Reacting Flow

- The **diffusion for the spark model** is now limited to cells in close proximity to the spark region specified. This results in a more realistic prediction of spark propagation. Historically, the spark model would affect diffusion throughout the flow domain, and the new treatment only affects diffusion around the location of the spark.

Discrete Phase Model

- Movement and deformation of sliding, moving, and deforming meshes are now considered during the particle tracking. This improves the accuracy of particle tracks when particles are reflected from moving walls, especially in cases without wall boundary layers. Results may vary from previous releases. This effect can be disabled by using the following scheme commands:

  ```
  (rpsetvar 'dpm/consider-transient-mesh-movement? #f)

  (check-mesh-interpolate-in-time)
  ```

- A **boiling rate equation for multi-component particles** has been introduced, which has been derived consistently with the existing vaporization and boiling models in ANSYS FLUENT. This boiling rate replaces the rate equation used previously for the multicomponent particle boiling regime. The documentation has been updated in the Theory Guide. This change cannot be reversed through an rpvar.
- For multicomponent particles, the **true boiling temperature** is used to limit the Langrangian wall film model. Previously, the minimum of the component boiling points was used. The user cannot change this selection.
- In the DPM energy balance, the latent heat is computed consistently in the droplet and Lagrangian film models. Previously, the film model always used a constant latent heat value. The user cannot revert to the old method.

- Improvements to the droplet **Vaporization Law** numerics result in a more accurate vaporization history. As a result of the improved accuracy, computed trajectories may be longer compared with FLUENT 13.0. In addition, computational time may increase compared to FLUENT 13.0 if the computed vaporization time is longer. The change can be reverted by issuing the following commands in sequence:

```
(rpsetvar ‘dpm/limiting-time-algorithm? #f)

(rpsetvar ‘dpm/minimum-vapor-fraction-new 0.01)

(dpm-parameters-changed)
```

- The **Multicomponent Law** numerics have been revised to speed up the computation. When importing case files from previous versions, you will need to disable **Coupled Heat-Mass Solution** for **Multicomponent** droplets to take advantage of the increased computational speed. This setting is found on the **Numerics** tab of the of the **Discrete Phase Model** dialog box.
- Several changes have been made to the **Lagrangian wall film model** that lead to more consistent evaporation of the wall film for pure and multi-component wall films. In addition, **splashing of droplets** has been improved to consider only one sampling from the cumulative probability density function of the underlying size distribution. These changes cannot be reversed.

Eulerian Multiphase Models

- The expression for ‘b’ in the Luo breakage kernel model in Table 2.1: "Luo Model Parameters" of the Population Balance Manual has been changed by a scaling factor, β^{-1}, where β=`2.047`. A domainvar, `‘pb/luo-beta-factor`, has been introduced to make this factor user-modifiable using the following scheme command:

```
(domainsetvar <pb-domain-id> 'pb/luo-beta-factor <value>)
```

The FLUENT 13 behavior can be recovered by issuing the preceding command with `<value>=1`.

Acoustics

- Ffowcs Williams-Hawkings solver: **reception time calculation** is improved by interpolating the emitted timestep signal between the receiver timesteps covered by the received signal.

UDF Programming Interface

- **Node unions replaced with node SVARs**.
  - Two node union data members `n1` and `n2` in `node_struct` have been replaced by `SV_N_TMP_0` and `SV_N_TMP_1`. `SV_N_TMP_2` is also available if needed. Unlike previous versions, UDF developers will need to allocate/deallocate this storage in order to use the following node union macros:
    - → `NODE_MARK` (uses `SV_N_TMP_0`)
    - → `NODE_RVAL1` (uses `SV_N_TMP_0`)
    - → `NODE_VISIT` (uses `SV_N_TMP_1`)
    - → `NODE_RVAL2` (uses `SV_N_TMP_1`)

    For your convenience, two macros (`ALLOCATE_NODE_SVAR` and `DEALLOCATE_NODE_SVAR`) have been added to facilitate allocating this storage. For example, in order to use `NODE_MARK`, you would use the commands:

```
ALLOCATE_NODE_SVAR(SV_N_TMP_0)
```

```
DEALLOCATE_NODE_SVAR(SV_N_TMP_0)
```

  - Many node union macros such as `NODE_VISIT` and `NODE_MARK` have been used for flagging the nodes, so it is not really necessary to use a node union variable to do it. For your convenience, 3 new macros have been added. Please use `CLEAR_NODE_VISITED` to initialize a node flag, `SET_NODE_VISITED` to mark a node, and `NODE_IS_VISITED` to check the node status. You may also use function `Clear_Node_Flags (domain, NODE_VISITED_FLAG)` to initialize all nodes in the domain, and use `Exchange_Node_Flags (domain, NODE_VISITED_FLAG)` to exchange node flags in parallel.

- For multiphase simulations, the linearized mass transfer UDF is now used by default. To revert to the previous behavior, use the TUI command `solve/set/expert` and enter **no** at the `Linearized Mass Transfer UDF?` prompt. Alternatively, you can use the following scheme command:

```
(rpsetvar ‘mp/mt/udf/linearized? #f)
```

Chapter 4: Text Command List and Settings Changes

Changes to the text command settings are listed in the tables in *Modified Text Command Settings* (p. 13) and *New Text Command Settings* (p. 18). Each table will list the changes to each of the text command menus. Please note that the modified setting can either be a changed setting or a deleted setting, while a new setting is one that did not exist in previous versions of FLUENT.

4.1. Modified Text Command Settings

Changes to the text command settings are listed in the following tables:

Table 4.1 Modified Text Commands/Settings for the `define/` Menu

FLUENT 13.0 TUI Menu Command	Modified Command or Setting
`define/dynamic-mesh/actions/remesh-cell-zone-cutcell`	new setting: `Remesh with polyhedra (yes) or hanging nodes (no)?`
`define/dynamic-mesh/controls/smoothing-parameters/skew-smooth-skew-max`	command name changed to: `skew-smooth-cell-skew-max`

`define/dynamic-mesh/controls/in-cylinder-parameters/piston-data`	`piston stroke`
	
	setting renamed to:
	
	`crank radius`

`define/dynamic-mesh/zones/create`	new meshing setting added for `rigid-body` and `user-defined` motion of dynamic face zones: `deform adjacent boundary layer with zone?`
`define/dynamic-mesh/zones/create`	new option added for `motion type`: `system-coupling`

FLUENT 13.0 TUI Menu Command	Modified Command or Setting
define/materials/change-create	new options added for Vaporization Model: diffusion-controlled convection/diffusion-controlled
define/materials/change-create	Thermal Conductivity property is available for DPM materials only when the thermophoretic force option or the wall-film model is active
define/materials/change-create	new setting for droplet materials: Binary Diffusivity
define/mesh-interfaces/ smallest-polygon-size	**Deleted**
define/models/acoustics/ moving-receiver	command name changed to: moving-receiver?
define/models/acoustics/ convective-effects	command name changed to: convective-effects?
define/models/dpm/numerics/ coupled-heat-mass-update	new setting to use the coupled algorithm for droplets: Droplet
define/models/dpm/numerics/ coupled-heat-mass-update	new setting to use the coupled algorithm for combusting particles: Combusting
define/models/dpm/numerics/ coupled-heat-mass-update	new setting to use the coupled algorithm for multicomponent particles: Multicomponent
define/models/viscous/ rsm-ssg-pressure-strain?	if enabled while near-wall treatment is set to Enhanced Wall Treatment, the near-wall treatment will be switched to Scalable Wall Function (in previous releases, near-wall treatment was switched to Standard Wall Function)
define/models/viscous/ near-wall-treatment/ non-equilibrium-wall-fn?	If answered **no**, near wall treatment method will switch to the default Standard Wall Function, regardless of whether non-equilibrium-wall-fn? is currently selected for NWT. Previously,

FLUENT 13.0 TUI Menu Command	Modified Command or Setting
	this switch only occurred if `non-equilibrium-wall-fn?` was the current selection.
`define/models/viscous/near-wall-treatment/enhanced-wall-treatment?`	If answered **no**, near wall treatment method will switch to the default Standard Wall Function, regardless of whether `enhanced-wall-treatment?` is currently selected for NWT. Previously, this switch only occurred if `enhanced-wall-treatment?` was the current selection.
`define/models/viscous/near-wall-treatment/scalable-wall-functions?`	If answered **no**, near wall treatment method will switch to the default Standard Wall Function, regardless of whether `scalable-wall-functions?` is currently selected for NWT. Previously, this switch only occurred if `scalable-wall-functions?` was the current selection.
`define/models/viscous/near-wall-treatment/user-defined-wall-functions?`	If answered **no**, near wall treatment method will switch to the default Standard Wall Function, regardless of whether `user-defined-wall-functions?` is currently selected for NWT. Previously, this switch only occurred if `user-defined-wall-functions?` was the current selection.
`define/models/viscous/near-wall-treatment/werner-wengle-wall-fn?`	If answered **no**, near wall treatment method will switch to the default Standard Wall Function, regardless of whether `werner-wengle-wall-fn?` is currently selected for NWT. Previously, this switch only occurred if `werner-wengle-wall-fn?` was the current selection.

Table 4.2 Modified Text Command Settings for the `file/` Menu

FLUENT 13.0 TUI Menu Command	Modified Command or Setting
`file/export-to-cfd-post`	additional setting to specify whether or not to write a case file: `Write Case File?`
`file/export/cfd-post-compatible`	additional setting to specify for which cell zones data is to be exported: `cell zone id/name`
`file/export/cfd-post-compatible`	additional setting to specify whether or not to write a case file: `Write Case File?`

FLUENT 13.0 TUI Menu Command	Modified Command or Setting
`file/export/ensight-gold`	additional setting to specify for which cell zones data is to be exported: `cell zone id/name`
`file/export/fieldview-unstruct`	additional setting to specify for which cell zones data is to be exported: `cell zone id/name`
`file/export/fieldview-unstruct-mesh`	additional setting to specify for which cell zones data is to be exported: `cell zone id/name`
`file/export/fieldview-unstruct-data`	additional setting to specify for which cell zones data is to be exported: `cell zone id/name`
`file/transient-export/cfd-post-compatible`	additional setting to specify for which cell zones data is to be exported: `cell zone id/name`
`file/transient-export/cfd-post-compatible`	additional setting to specify whether or not to write a case file: `Write Case File?`
`file/transient-export/ensight-gold-transient`	additional setting to specify for which cell zones data is to be exported: `cell zone id/name`
`file/transient-export/ensight-gold-from-existing-files`	additional setting to specify for which cell zones data is to be exported: `cell zone id/name`
`file/transient-export/fieldview-unstruct`	additional setting to specify for which cell zones data is to be exported: `cell zone id/name`
`file/transient-export/fieldview-unstruct-mesh`	additional setting to specify for which cell zones data is to be exported: `cell zone id/name`

FLUENT 13.0 TUI Menu Command	Modified Command or Setting
`file/transient-export/fieldview-unstruct-data`	additional setting to specify for which cell zones data is to be exported: `cell zone id/name`
`file/em-mapping/hfss`	**Deleted**
`file/em-mapping/maxwell`	**Deleted**
`file/em-mapping/q3d`	**Deleted**
`file/em-mapping/maintain-loss`	command renamed `maintain-loss-on-initialization`
`file/import/patran/result`	**Deleted**

Table 4.3 Modified Text Command Settings for the `mesh/` Menu

FLUENT 13.0 TUI Menu Command	Modified Command or Setting
`mesh/modify-zones/replace-zone`	new setting to specify whether or not to interpolate existing data: `Interpolate data?`
`mesh/repair-improve/print-repair-improve-solver-statistics`	command renamed to: `report-poor-elements`

Table 4.4 Modified Text Command Settings for the `solve/` Menu

FLUENT 13.0 TUI Menu Command	Modified Command or Setting
`solve/max-iterations-per-time-step`	`solve/set/max-iterations-per-time-step`
`solve/monitors/force/clear-all-monitors-data`	replaced by: `solve/monitors/force/clear-monitors` with revised prompts
`solve/monitors/force/clear-drag-monitor-data`	replaced by: `solve/monitors/force/clear-monitors` which prompts whether or not to clear all monitor data

FLUENT 13.0 TUI Menu Command	Modified Command or Setting
`solve/monitors/force/clear-lift-monitor-data`	replaced by: `solve/monitors/force/clear-monitors` with revised prompts
`solve/monitors/force/clear-moment-monitor-data`	replaced by: `solve/monitors/force/clear-monitors` with revised prompts
`solve/monitors/force/drag-coefficient`	replaced by: `solve/monitors/force/set-drag-monitor`
`solve/monitors/force/lift-coefficient`	replaced by: `solve/monitors/force/set-lift-monitor`

`solve/monitors/surface/set-monitor`	new report types added:
	`Uniformity Index - Mass Weighted`
	`Uniformity Index - Area Weighted`

`solve/monitors/force/moment-coefficient`	replaced by: `solve/monitors/force/set-moment-monitor`
`solve/set/bad-mesh-numerics`	command renamed: `solve/set/poor-mesh-numerics`
`solve/set/data-sampling`	new setting: `Collect statistics for custom field functions?`
`solve/set/p-v-coupling`	new setting when using Volume of Fluid model: `Coupled with Volume Fractions?`
`solve/set/pseudo-eqn-time-step/`	**Deleted**

4.2. New Text Command Settings

Text command settings that are new to FLUENT 14.0 are listed in the following tables:

- *Table 4.5: New Text Command Settings for the* `define/` *Menu* (p. 19)

Table 4.5 New Text Command Settings for the `define/` Menu

FLUENT 14.0 TUI Menu Command
`define/boundary-conditions/modify-zones/copy-move-cell-zone`
`define/dynamic-mesh/controls/six-dof-parameters/motion-history-file-name`
`define/dynamic-mesh/controls/smoothing-parameters/spring-on-deformable-shapes?`
`define/dynamic-mesh/controls/smoothing-parameters/skew-smooth-face-skew-max`
`define/dynamic-mesh/controls/smoothing-parameters/skew-smooth-all-deforming-boundaries?`
`define/dynamic-mesh/controls/smoothing-parameters/diffusion-coeff-function`
`define/dynamic-mesh/controls/remeshing-parameters/prism-layer-parameters/`
`define/dynamic-mesh/controls/remeshing-parameters/prism-layer-parameters/first-height`
`define/dynamic-mesh/controls/remeshing-parameters/prism-layer-parameters/growth-rate`
`define/dynamic-mesh/controls/remeshing-parameters/prism-layer-parameters/number-of-layers`
`define/dynamic-mesh/controls/remeshing-parameters/zone-remeshing`
`define/mesh-morpher-optimizer/optimizer-parameters/monitor/`
`define/mesh-morpher-optimizer/optimizer-parameters/monitor/clear-opt-hist`
`define/mesh-morpher-optimizer/optimizer-parameters/monitor/plot-hist`

FLUENT 14.0 TUI Menu Command
define/mesh-morpher-optimizer/optimizer-parameters/monitor/plot?
define/mesh-morpher-optimizer/optimizer-parameters/monitor/write?
define/models/dpm/collisions/
define/models/dpm/collisions/collision-mesh
define/models/dpm/collisions/collision-pair-settings/
define/models/dpm/collisions/collision-pair-settings/contact-force-normal
define/models/dpm/collisions/collision-pair-settings/list-pair-settings
define/models/dpm/collisions/collision-pair-settings/ contact-force-tangential
define/models/dpm/collisions/collision-partners/
define/models/dpm/collisions/collision-partners/copy
define/models/dpm/collisions/collision-partners/create
define/models/dpm/collisions/collision-partners/delete
define/models/dpm/collisions/collision-partners/list
define/models/dpm/collisions/collision-partners/rename
define/models/dpm/collisions/dem-collisions?
define/models/dpm/collisions/list-all-pair-settings
define/models/dpm/collisions/max-particle-velocity
define/models/dpm/numerics/vaporization-limiting-factors
define/models/dpm/spray-model/ssd-model
define/models/dpm/options/vaporization-options
define/models/species/reacting-channel-model?
define/models/species/reacting-channel-model-options
define/models/species/surf-reaction-netm-params

FLUENT 14.0 TUI Menu Command
define/models/viscous/curvature-correction?
define/models/viscous/les-subgrid-wmles?
define/models/viscous/sa-enhanced-wall-treatment?
define/models/viscous/turbulence-expert/curvature-correction-coefficient
define/phases/iac-expert/iac-pseudo-time-step

Table 4.6 New Text Command Settings for the `display/` Menu

FLUENT 14.0 TUI Menu Command
display/plot-reacting-channel-vars
display/set/particle-tracks/sphere-settings/
display/set/particle-tracks/sphere-settings/vary-diameter?
display/set/particle-tracks/sphere-settings/diameter
display/set/particle-tracks/sphere-settings/auto-range?
display/set/particle-tracks/sphere-settings/minimum
display/set/particle-tracks/sphere-settings/maximum
display/set/particle-tracks/sphere-settings/smooth-parameter
display/set/particle-tracks/sphere-settings/scale-factor
display/set/particle-tracks/sphere-settings/size-variable
display/set/particle-tracks/vector-settings/
display/set/particle-tracks/vector-settings/style
display/set/particle-tracks/vector-settings/vector-length
display/set/particle-tracks/vector-settings/vector-length-variable
display/set/particle-tracks/vector-settings/scale-factor
display/set/particle-tracks/vector-settings/length-variable?
display/set/particle-tracks/vector-settings/length-to-head-ratio

FLUENT 14.0 TUI Menu Command
display/set/particle-tracks/vector-settings/constant-color
display/set/particle-tracks/vector-settings/color-variable?
display/set/particle-tracks/vector-settings/vector-variable
display/set/particle-tracks/filter-settings/
display/set/particle-tracks/filter-settings/enable-filtering?
display/set/particle-tracks/filter-settings/inside?
display/set/particle-tracks/filter-settings/filter-variable
display/set/particle-tracks/filter-settings/minimum
display/set/particle-tracks/filter-settings/maximum

Table 4.7 New Text Command Settings for the `file/` Menu

FLUENT 14.0 TUI Menu Command
file/em-mapping/volumetric-energy-source

Table 4.8 New Text Command Settings for the `mesh/` Menu

FLUENT 14.0 TUI Menu Command
mesh/modify-zones/copy-move-cell-zone
mesh/polyhedra/convert-hanging-nodes
mesh/polyhedra/options/preserve-interior-zones

Table 4.9 New Text Command Settings for the `parallel/` Menu

FLUENT 14.0 TUI Menu Command
parallel/partition/set/layering
parallel/partition/set/stretched-mesh-enhancement

Table 4.10 New Text Command Settings for the `report/` Menu

FLUENT 14.0 TUI Menu Command
report/surface-integrals/uniformity-index-area-weighted

FLUENT 14.0 TUI Menu Command
report/surface-integrals/uniformity-index-mass-weighted

Table 4.11 New Text Command Settings for the `solve/` Menu

FLUENT 14.0 TUI Menu Command
solve/initialize/levelset-auto-init
solve/monitors/force/delete-monitors
solve/monitors/force/list-monitors
solve/set/pseudo-transient-expert
solve/set/coupled-vof-expert
solve/set/convergence-acceleration-for-stretched-meshes/
solve/set/number-of-iterations
solve/set/number-of-time-steps
solve/set/poor-mesh-numerics-quality-based?
solve/set/vof-numerics
solve/set/high-order-term-relaxation/
solve/set/high-order-term-relaxation/enable?
solve/set/high-order-term-relaxation/options/
solve/set/high-order-term-relaxation/options/relaxation-factor
solve/set/high-order-term-relaxation/options/variables/
solve/set/high-order-term-relaxation/variables/select