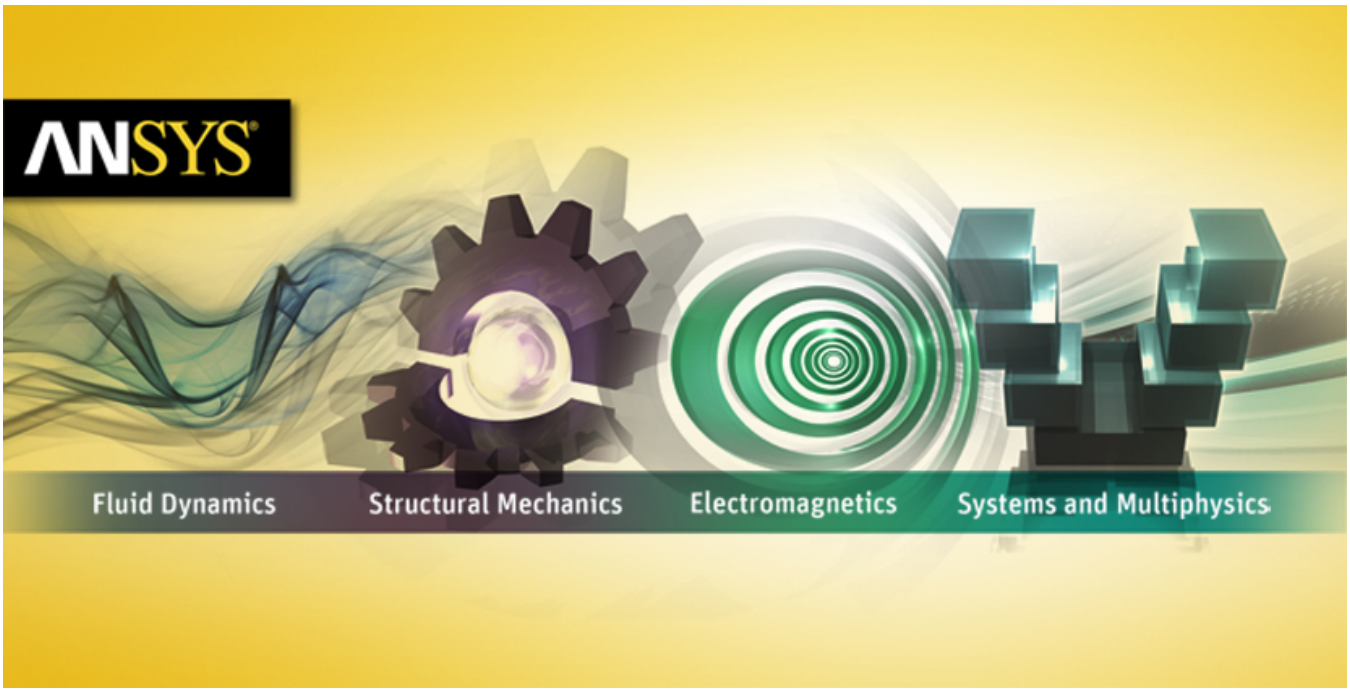

Release Notes

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 15.0
November 2013
000504

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

Copyright and Trademark Information

Disclaimer Notice

U.S. Government Rights

Third-Party Software

Published in the U.S.A.

Table of Contents

Global Release Notes

The release notes are specific to ANSYS, Inc. Release 15.0 and arranged by application/product, with the exception of:

- Advisories (p. ix)
- Installation (p. ix)
- Licensing (p. x)
- ANSYS Customer Portal (p. xi)

Note that installation- and licensing-specific information is detailed in some application and product sections. The release notes are available in printable format (PDF) via the product media, and accessible in the ANSYS Help Viewer or online via the ANSYS Customer Portal (p. xi).

To view release notes specific to previous ANSYS, Inc. releases, go to ANSYS Customer Portal> Downloads> Previous Releases> Documentation. The release notes files are named "ai_rn (followed by the inventory number) in the following zip file links:

- ANSYS 14.5.7 Release Documentation (applies to release 14.5)
- ANSYS 14.0 Release Documentation
- ANSYS 13.0 Release Documentation

1. Advisories

ANSYS 15.0 will be the last release to support the Windows XP operating system, Windows 7 (32-bit) operating system, and the Windows 32-bit platform. Subsequent service packs' and verification testing packages' discontinuance of support of Windows XP will concur with Microsoft's discontinuance of the same, anticipated to be April 2014. For detailed and current support information, see the Platform Support section of the ANSYS, Inc. website (www.ansys.com).

In addition to the incompatibilities noted within the release notes, known non-operational behavior, errors and/or limitations at the time of release are documented in the **Known Issues and Limitations** document, although not accessible via the ANSYS Help Viewer. See the ANSYS Customer Portal for information about the ANSYS service packs and any additional items not included in the **Known Issues and Limitations** document. First-time users of the customer portal must register to create a password.

2. Installation

The following features are new or changed at Release 15.0. Please review these items carefully.

- The Windows and Linux installation guides have been rewritten and reorganized to enhance ease of use.
- Installs and Uninstalls can now be performed from a single unified installation interface.
- An installation launcher is now available for the Linux platform. From this launcher, you can run the ANSYS product, License Manager and EKM installs.

- For Windows systems, ANSYS, Inc. Product installation must be performed by users with administrative privileges.
- Enhancements to the installation user interface include:
  - Improved installation speed
  - Streamlined content display during the installation process
  - Progress bar tracks the installation process

3. Licensing

The following enhancements have been made to ANSYS, Inc. Licensing for Release 15.0:

- The ANSYS, Inc. License Manager now supports the use of IP addresses & LM_PROJECT in the FlexNet options file.
- DesignPoint studies and ANSYS HPC Parametric Pack has been enhanced in the following ways:
  - ANSYS Icepak licenses may now be reserved for DesignPoint studies and can be used with HPC Parametric Packs.
  - ANSYS Structural, ANSYS Structural Solver and ANSYS Professional NLS licenses may now be reserved for DesignPoint studies and can be used with HPC Parametric Packs
- In the **License Preference** dialog box, the **PrePost** tab now includes the product licenses that provide both pre/post processing & solver capability. Previously, this class of product licenses (example ANSYS Mechanical) were only available in the **Solver** tab.
- The ANSYS ASAS SPLINTER functionality is now enabled in these products: ANSYS Multiphysics, ANSYS Mechanical, ANSYS Professional NLS, ANSYS Structural and any ANSYS Academic products that contain ANSYS Mechanical simulation capabilities.
- Graphics Processing Unit (GPU) licensing has been enabled for both ANSYS HPC and ANSYS HPC Pack licenses.
- EKM no longer needs to check out application licenses when extracting data for reports.

4. Documentation

The Help Viewer has been updated with a more modern browser look-and-feel. It offers several enhanced and new capabilities:

- Improved full-text searching, including filters for searching by physics, application, type, and book.
- Boolean searching using a plain English query builder.
- Table of Contents filtering, so you can hide any books that you don't use on a regular basis and then save the filtered version as your default view.
- Annotations, allowing you to add your own comments to the existing help pages. Annotations can be saved and later imported into a future version of the help.

- Improved highlighting and display of your current position in the documentation, so you can easily see where you are at a glance.
- Bookmarking ability.

5. ANSYS Customer Portal

If you have a password to the ANSYS Customer Portal (support.ansys.com), you can view additional documentation information and late changes. The portal is also your source for ANSYS, Inc. software downloads, service packs, product information (including example applications, current and archived documentation, undocumented commands, input files, and product previews), and online support.

All the product documentation is available in printable format (PDF). Note that the content of the files can be copied into word processing programs.

Customer Portal access points:

- **Tutorials and input files** To access tutorials and their input files on the ANSYS Customer Portal, go to http://support.ansys.com/training.
- **Documentation** To access documentation files on the ANSYS Customer Portal, go to http://support.ansys.com/documentation.
- **General information** For further information about tutorials and documentation on the ANSYS Customer Portal, go to http://support.ansys.com/docinfo.

Part I: ANSYS Structural Products

Release notes are available for the following ANSYS Structural products:

Chapter 1: Mechanical Application Release Notes

This release of the Mechanical application contains all of the capabilities from previous releases plus many new features and enhancements. Areas where you will find changes and new capabilities include the following:

1.1. Incompatibilities and Changes in Product Behavior from Previous Releases

Release 15.0 includes several new features and enhancements that result in product behaviors that differ from previous releases. These behavior changes are presented below.

- **Adjustment for Bolt Pretension**. Mechanical now applies the pre-adjustment from the solved deformation value of the previous step to the specified adjustment value. In previous releases, the pre-adjustment was applied from zero to the specified adjustment value.
- **Exporting Results for Fluid Solid Interface**. Compared to the pervious release, the **Export Results** property of the **Fluid Solid Interface** boundary condition is now, by default, set to **No**. In order to export thermal results to an axdt.file, change this setting to **Yes**.
- **Structural Linear Periodic Symmetry**. The MAPDL **CE** command is now used to solve the Linear Periodic boundary condition.
- **Trim Contact**. For Bonded Contact pairs, the **Program Controlled** option for the **Trim Contact** property now performs trimming for all instances of automatically generated contact. In prior versions of Mechanical, trimming was not performed if the **Large Deflection** property was turned on.
- **Mode Reuse** – **Solver Controls**. The **Mode Reuse** property has been moved from the **Rotordynamics** category to the **Solver Controls** of the **Analysis Settings** object.
- **Harmonic Response Analysis**. For Harmonic Response Analysis result type objects, the property **Phase Angle** has been changed to **Sweeping Phase**.

- **Analysis Settings Output Controls**. The maximum number of sets stored in the results file is now dynamic and as a result, the **Max Number of Result Sets** property for the Analysis Settings **Output Controls** category was removed.

- **Analysis Settings Radiosity Controls**. The **Solver Tolerance** property of the **Radiosity Controls** category is now dependent upon the user-defined unit of measure.

- In previous versions, if a high frictional contact was present (friction > 0.2), an unsymmetric stiffness matrix was automatically used during a static or transient structural analyses. Now with the introduction of the new **Newton-Raphson Option** property, you must specify an unsymmetric matrix by selecting the **Unsymmetric** option in the **Newton-Raphson Option** property.

- For Imported Displacements from External Data Import, the **Ignore** identifier was used to ignore a component, i.e. not apply any load component if the data was not available. This resulted in a component at a step being fixed or free based on if load at the component was applied or not in any of the previous steps. Now explicit options for **Fixed** (*displacement = 0.0*) and **Free** components are available, and **Ignore** option is removed.

- **Pre-load the Mechanical editor**. The default setting to pre-load the Mechanical editor has been changed. This option (via Workbench **Tools**>**Options**) is no longer active by default when launching the application.

- You can now pre-select a vertex or node or two vertices or nodes and then insert the following objects to automatically create a directly attachment without the need of an intermediate remote point.

  – Joints

  – Springs

  – Beam Connections

  – Point Mass

  – Thermal Point Mass

- The **Plot Elements Attached to Named Selections** option (selected in the **Annotation Preference** dialog box) is active (checked) by default. In prior releases, it was unchecked by default which caused only the geometry to be highlighted when you selected a Named Selection.

- The Linear Buckling analysis now supports only the linear contact types **Bonded** and **No Separation** with the **MPC** formulation method.

- The Design Assessment system no longer supports upstream Response Spectrum or Random Vibration systems when the assessment type is **Solution Combination Only**.

- **Random Vibration Analysis**. You can now link a Random Vibration analysis system to a fully solved Modal analysis without causing the solution of the Modal system to become obsolete. This new behavior may, however, result in zero equivalent stress results in the Random Vibration analysis. Re-solving the Modal analysis corrects this condition.

1.2. General Enhancements

The following general enhancements have been made at Release 15.0:

- **Direct Scoping**. The following objects can now be scoped directly to a vertex or node of a model instead of using an intermediate remote point.

  - Joints
  - Springs
  - Beam Connections
  - Point Mass
  - Thermal Point Mass

  This is accomplished with the new property **Applied By** that is available in the Details of the object. The **Direct Attachment** option allows you to scope directly to a node of the model while the **Remote Attachment** option uses a **Remote Point** as a scoping mechanism.

- **Filtering the Tree Outline**.
  - You can now remove objects from the tree display using the new Filter feature, **Remove**. The Filtering system works the same for finding and displaying objects, but this new feature allows you to also exclude filtered objects from the display.
  - You can now filter the tree by **Boundary Condition** objects, **Connections** objects, or **Command** objects in addition to **Results**.
  - **Coordinate System** has been added as a new filtering option.
- **Mechanical Shortcut Keys**. Certain selection and view shortcut keys are now available in Mechanical.
- **Coordinate System Object Suppression**. You can now suppress coordinate system objects.
- **Create Element-based Named Selections**. You can now create Named Selections based on element selections on a mesh. You may then use the element-based Named Selections with the MAPDL Solver via a Command snippet.
- **Solving Units**. Mechanical now supports a new unit quantity, Temperature Gradient, for temperature changes per unit length.
- **Creating Named Selections through Promotion**. A new option, **Promote to Named Selection**, is available that allows you create a Named Selection based on the scoping of a Contact Region, Boundary Condition, Remote Point, Spring, Joint, and Result objects.

1.3. Analysis Enhancements

The following analysis enhancements have been made at Release 15.0:

- **Higher Order Thermal Shells**. You can now analyze surface bodies meshed with midside nodes for thermal analyses.
- **Composite Solids**. Steady State Thermal and Transient Thermal analyses are now supported for composite solids defined using ACP and imported into Mechanical.
- **Harmonic Response (Full) Analysis Using Pre-Stressed Structural System**. A Harmonic Response analysis using the Full solution method can now be linked to an existing Static Structural System analysis on the Project Schematic. In this way, multiple full harmonic analyses with different loading conditions could effectively reuse the pre-stressed effect without re-solving the structural analysis.

- **Contact Debonding**. **Contact Debonding** is a new Mechanical feature for structural analyses that allows you to define contact regions along an interface that will separate.

- **Interface Delamination**. **Interface Delamination** is a new Mechanical feature for structural analyses that allows you to simulate the separation of two materials across an interface.

1.4. Geometry Enhancements

The following geometry enhancements have been made at Release 15.0:

- **Importing Mesh-Based Geometry**. You can now import mesh-based geometries that are in the Mechanical APDL mesh common database (.cdb) format using the **External Model** system provided on the Workbench Project page.

- **Assembling Mechanical Models**. You can now assemble and import one or more individually meshed models from the Workbench Project page into Mechanical to create larger models by using the **Mechanical Model** component system, the **External Model** component system, and/or an analysis system.

1.5. Contact and Connection Enhancements

The following contact and connection enhancements have been made at Release 15.0:

- **Connections Worksheet**. The Connections Worksheet now provides information for the properties of Spring Connections and Beam Connections.

- **Bearings**. The **Bearing** connection is now supported by all Mechanical ANSYS Parametric Design Language (MAPDL) Solver analyses. Prior releases supported Modal analyses only.

- **Nonlinear Spring Stiffness**. The MAPDL Solver now supports tabular loading for longitudinal spring stiffness (Force vs. Deflection).

- **Geometric Modification**. Mechanical now provides a Contact Region settings category called **Geometric Modification**. The **Interface Treatment** property (formerly include in the Advanced contact settings category) has been moved to this category. In addition, the category includes a new feature/property called Contact Geometry Correction that allows you to model bolt threading. Please see the Help section for all of the specific property definition requirements.

- **Joint Stiffness**. For the MAPDL Solver, Mechanical now supports Tabular format entries to define **Stiffness Coefficients** using the **Worksheet**.

- **Contact Scope Settings**. You can now utilize the new Contact Region **Scoping Method** option, **Pre-Generate Interface**, to address unexpected penetration results when you are performing fracture analyses using the Interface Delamination feature in combination with the ANSYS Composite PrepPost (ACP) application.

- **Penetration Tolerance**. The contact property, **Penetration Tolerance**, now supports the **Formulation** setting **Pure Penalty**.

1.6. Graphics Enhancements

The following graphical enhancements have been made at Release 15.0:

- The Mesh is now colored to indicate meshing failure, or mesh that is out of date. See Recommended First Course of Action for Meshing Failures for more information.

- **Graphical Tools** - **Select Mode**. You can now change your selection mode from **Single Select** to **Box Select** by holding the right mouse button and then clicking the left mouse button. In addition, given a generated mesh and the **Mesh Select** option is active, holding the right mouse button and then clicking the left mouse button scrolls through the available selection options (single section, box selection, box volume, lasso, lasso volume).

1.7. Loads/Supports/Conditions Enhancements

The following loads/supports/conditions enhancements have been made at Release 15.0:

- **Incremental Adjustment for Bolt Pretension**. An **Increment** option (incremental adjustment) is now available for the **Bolt Pretension** boundary condition. When applied, the specified value is added to the solved deformation value from the previous step.
- **Radiation**. The **Radiation** boundary condition now allows you to specify a perfectly enclosed system using the new property, **Enclosure Type**.
- For a **Harmonic Analysis**, the following boundary conditions now supports **Phase Angle** allowing you to phase shift the loading conditions.
  - **Remote Force**
  - **Moment**
- For a **Harmonic Analysis**, the following boundary conditions now support frequency dependent tabular loading.
  - **Acceleration**
  - **Force**
  - **Moment**
  - **Pressure**
  - **Remote Force**
- **Fluid Solid Interface**. The **Fluid Solid Interface** boundary condition is now supported for Transient Thermal Analyses.
- **Convection**. The **Convection** boundary condition now provides an option to use either a diagonal or consistent film coefficient matrix.
- **Compression Only Support**. The **Compression Only Support** boundary condition now supports the properties **Normal Stiffness** and **Update Stiffness**.
- **Hydrostatic Pressure**. You can now deactivate the Hydrostatic Pressure boundary condition on a per step basis during a multi-step analysis. In addition, this boundary condition can now be parameterized.

1.8. Finite Element (FE) Enhancements

The following FE enhancements have been made at Release 15.0:

- **Path (Construction Geometry)**. When defining a path, you can now define the start and end locations by selecting one or more nodes. If multiple nodes are selected to set the **Start** or **End Location**, an averaged location of the selected nodes is used.

- **Remote Point Scoping**. You can now scope a Remote Point to a single node or to multiple nodes on the geometry.

- **Importing Mesh-Based Geometry**. You can now import mesh-based geometries that are in the Mechanical APDL mesh common database (.cdb) format using the **External Model** system provided on the Workbench Project page.

- **Element Selections**. The ability to select Elements is now available in Mechanical and includes the following capabilities:

  - **Graphical Selection** . You can select Elements in the graphics window and corresponding Element information is available from the **Selection Information** Window.

  - **Named Selection**. You can created Element-based Named Selections using graphical selection or criterion-based selection using the Worksheet. Criterion such as Element ID, Type, Location, and Mesh Metrics are supported. Element-based Named selections are written into the MAPDL input file.

  - **Results Scoped To Element Selections**. You can scope Results, including User Defined Results, to Element-based Named Selections.

- **Exporting Node Results**. You can now select and export results for one or more nodes on a result plot.

1.9. Mapping Enhancements

The following mapping enhancements have been made at Release 15.0:

- **Support for Velocity Data**. Velocities can now be imported from text files and applied as loads in a Harmonic Response Analysis.

- **Support for Stress and Strain Data**. Stress and Strain can now be imported from text files and applied as loads in a structural analysis.

- **Support for Body Force Density Data**. Body Force density can now be imported through External Data, and applied as loads in a structural analysis.

- **Imported Pressure** loads from External Data can now be applied to nodes as well as elements.

- **Imported Boundary Conditions** now support time/frequency dependent tabular loading.

- **Submodeling**. You can now perform a structural or thermal submodeling analysis, on surface as well as solid bodies, by linking systems in the Project Schematic. Several mapping options are available to transfer displacements, rotations, and temperatures at the cut-boundaries.

- **One-way Acoustic Coupling**. You can now import velocities in an acoustic analysis from a linked Harmonic Response Analysis.

- **Mapping Validation**. Mapping validation objects can now be inserted under imported data objects from linked analyses (Thermal Stress, Submodeling, and Acoustic Coupling)

- **Maxwell - Mechanical Coupling**. Frequency dependent Force and Moment data, generated from a transient electromagnetic simulation, can now be imported and applied in a Harmonic Response analysis.

- Legend controls are now supported for all imported loads. The minimum and maximum values of source data are now also available for Thermal Stress, Submodeling, and **One-way Acoustic Coupling** analyses.

- **UV Mapping** is now available to transfer data between non-coincident surfaces.

1.10. Solution Enhancements

The following solution enhancements have been made at Release 15.0:

- **Options for Analyses**. Based on your Harmonic Analysis **Options** category settings, a harmonic environment stores only results requested at the time of solution to minimize data storage. This feature now also supports **Force Reaction** and **Moment Reaction** results.

- **Radiosity Controls**. The **Analysis Settings** group, **Radiosity Controls**, now provides the property, **Radiosity Solver**, allowing you to specify the solver for the radiosity calculation: selections include Gauss-Seidel iterative solver (**Program Controlled** default), **Direct** solver, or the **Iterative** Jacobi solver.

- **Analysis Settings - Nonlinear Controls**. A new Nonlinear Controls property has been implemented, **Newton Raphson Options**, and is now available for use with Static Structural and Full Transient Structural analysis types (only). The **Newton Raphson Options** property allows you to specify how often the stiffness matrix is updated during the solution process.

- **Analysis Settings - Output Controls**. A new **Output Control**, **Keep Modal Results**, is available for use in Random Vibration analyses. It provides the option to include or exclude modal results in the result file once the analysis is solved.

- **Solve Process Settings**. Updates to the Advanced Properties of the **Solve Process Settings** include:

  – The addition of the new property **Number of utilized GPU devices**, which allows you to specify the number of GPU accelerator devices when the **Use GPU acceleration** property is set to use a valid accelerator type.

  – The Graphics Processing Unit (GPU) acceleration capability now supports the Intel acceleration card.

- **Solver Controls** - **Solver Type**. The **Subspace** solver type option is now available for Modal and Linear Buckling analysis types.

- **Solving**. Users can now make multiple attempts to download results from a remote machine given the possibility of a network connection loss to the Remote Solve Manager.

1.11. Rigid Body Solver Enhancements

The following Rigid Body Solver enhancements have been made at Release 15.0:

- **Follower Load** - The new **Follower Load** field on Remote Force objects allows the force direction to be updated with body movements during a simulations. Refer to Remote Force in the *ANSYS Mechanical User's Guide* for more information.

- **FFT based load** - A new fitting method based on Fast Fourier Transform is available for tabular loads. This new fitting method allows the use of complex table definitions for displacement and velocities. This method can be used in place of cubic spline fitting. Refer to Preparing a Rigid Dynamics Analysis in the *ANSYS Mechanical User's Guide* for more information.

1.12. Explicit Dynamics Solver Enhancements

The following Explicit Dynamics Solver enhancements have been made at Release 15.0:

- **Explicit Dynamics analyses now always use double precision.** Using double precision will generally provide the most robust solution. There is typically only a limited increase in CPU time (on the order of 10% to 20%) and memory consumption (on the order of 10% to 30%).
- **Enhanced crushable foam support for Explicit Dynamics analyses.** The full stress/strain curve from Engineering Data is now passed to the Explicit solver for crushable foam.
- **Explicit Dynamics analyses now support springs**. Explicit Dynamics analyses now support springs (tension, compression, both). A stiffness curve can be defined for nonlinear springs.
- **Body Interaction Go To option**. A new Go To option is available for Explicit Dynamics analyses that identifies Body Interaction objects in the tree that are associated with selected bodies.
- **Pressure Initialization option for Pre-Stress**. A Pressure Initialization option that allows you to initialize pressure from the deformed state or from the stress state has been added to the Pre-Stress object for Explicit Dynamic analyses.
- **Spring Result Tracker**. You can use a spring tracker to display the longitudinal results from a spring in an Explicit Dynamics analysis.

1.13. Results Enhancements

The following results enhancements have been made at Release 15.0:

- **Reactions: Forces and Moments**.
  - A new Harmonic Analysis setting now allows you to control the **Phase Increment** for Reaction and Moment results for the **Maximum Over Phase** and **Phase of Maximum** properties.
  - Force and Moment Reaction results can now be scoped to a Contact Region in a Modal analysis or a Harmonic Response Analysis (Full and MSUP).
- **Averaged vs. Unaveraged Contour Results**. You can now averages the results across separate bodies on your model using the new **Average Across Results** property.
- **Result Summary Worksheet**. The **Solution** Object now provides a context menu (right-click) option to display a summary of result information in tabular format.
- **Results Graphical Display**. The new Display tool on the Results Context Toolbar allows you to view only the specific part or parts that you have scoped a result to.
- **Results Scoped To Element Selections**. You can now scope Results, including User Defined Results, to Element-based Named Selections.
- **Velocity Response in Harmonic Analysis**. The **Frequency Response** result type now supports **Velocity**. All three variations of the Harmonic Response analysis (MSUP, linked MSUP, and Full) support this result.
- **Results Definition Parameterization**. You can now parameterize the **Display Time** property for a Static Structural and Transient Structural analyses and the **Frequency** property during a Harmonic Response analysis.

- **Bearing Probe**. Bearing results are now supported for the following Mechanical systems: Static Structural, Transient Structural, Modal, Harmonic Response, Random Vibration, and Response Spectrum.
- **Response Spectrum Analysis**. The Response Spectrum analysis now supports the **Total Deformation** result type.

Chapter 2: Mechanical APDL Release Notes

Release 15.0 of the Mechanical APDL application offers most of the capabilities from prior releases plus many new features and enhancements. Areas where you will find changes and new capabilities include the following:

- Structural (p. 13)
- Coupled Physics (p. 20)
- Solvers (p. 23)
- Commands (p. 25)
- Elements (p. 29)
- Documentation (p. 30)

Also see Known Incompatibilities (p. 32) and ANSYS Customer Portal (p. xi) for important information about this release.

2.1. Structural

Release 15.0 includes the following new features and enhancements for structural analyses:

2.1.1. Contact

Release 15.0 includes the following enhancements for structural analyses involving contact:

2.1.1.1. Bolt Thread Modeling

When simulating bolts, detailed modeling of bolt threads can be computationally expensive and may not be practical to simulate. The new bolt thread modeling technique eliminates the need for a detailed mesh discretization of the threads. Instead, you model smooth cylindrical surfaces on both the bolt and the bolt hole geometries and define a contact pair between the surfaces. The contact normal is computed internally based on user-specified thread geometry data and the two end points of the bolt axis (input via the **SECTYPE** and **SECDATA** section commands). This feature offers simplified modeling with the near accuracy of a true threaded-bolt model.

Bolt thread modeling is applicable to 3-D models and 2-D axisymmetric models and is available for contact elements CONTA171, CONTA172, CONTA173, CONTA174, and CONTA175. For more information, see Bolt Thread Modeling in the *Contact Technology Guide* and Bolt Thread Simulation in the *Technology Demonstration Guide*.

2.1.1.2. Contact Surface Wear

A new modeling capability is available to simulate wear at contact surfaces. Wear at the surface results in material loss. Material loss due to wear is approximated by repositioning the contact nodes at the contact surface. The new coordinates of the nodes are determined by the Archard wear model or a user-defined wear model (USERWEAR subroutine). Wear results in redistribution of contact pressure and a loss of material from the underlying solid elements.

Wear is available for the following contact elements: CONTA171, CONTA172, CONTA173, CONTA174, and CONTA175. For more information, see Contact Surface Wear in the *Contact Technology Guide*.

2.1.1.3. Traction-Based Model for Line-to-Line and Line-to-Surface Contact

A new contact traction-based option is available for the 3-D line-to-line contact element (CONTA176) and the 3-D line-to-surface contact element (CONTA177). When the traction-based option is set, the program determines the area (based on the beam element length and beam section radius) associated with the contact node. Because it is less sensitive to element size and mesh discretization, the traction-based option offers better convergence and solution results than the original force-based option.

2.1.1.4. Enhancements for Contact Interfacial Behavior Subroutines

The user-programmable subroutines USERCNPROP and USERINTER allow you to define your own contact properties and contact interaction behaviors, respectively. The following enhancements are available for these subroutines:

- The USERCNPROP and USERINTER subroutines can now be called at a given frequency value for a harmonic analysis.
- As one of the output arguments from USERINTER, the user-defined contact damping matrix can be programmed for a complex modal analysis, a full transient structural analysis (time domain), or a harmonic analysis (frequency domain).

2.1.1.5. Other Contact Changes and Enhancements

The new **CNTR** command allows you to redirect contact pair output quantities to a text file (`Jobname.CNM`).

The new *Lkey* = FIXED option on the **FDELE** command applies the current relative displacement value to the specified degree of freedom in addition to deleting the force.

There is now no limit on the number of temperature points that can be used when defining a coefficient of friction specification (**TB**,FRIC) or a user-defined contact interaction specification (**TB**,INTER).

2.1.2. Elements and Nonlinear Technology

Release 15.0 includes the following enhancements to elements and nonlinear technology used in structural analyses:

2.1.2.1. Rezoning

Support for rezoning now includes surface-effect elements SURF153 and SURF154. For a complete list of supported elements, see Rezoning Requirements in the *Advanced Analysis Guide*.

When remeshing using the mesh-splitting option, support is now available for 3-D mesh splitting. For more information, see Remeshing Using Manual Mesh Splitting in the *Advanced Analysis Guide*.

2.1.2.2. Mesh Nonlinear Adaptivity (Automatic Rezoning)

Mesh nonlinear adaptivity is now available for modifying the mesh during solution, based on certain criteria either to simulate some problem which otherwise cannot be simulated or to improve the accuracy of simulation results. Unlike rezoning, nonlinear adaptivity is completely automatic, requiring no user input during solution. Mesh adaptivity through refinement can improve solution accuracy in general. It can help to capture local deformations in more detail, useful in applications such as rubber sealing for small cavities, diffuse or local necking, and local buckling. The program offers contact-based, energy-based, and position-based criteria to determine whether the mesh needs to be modified and, if so, what parts of the mesh should be modified.

Mesh nonlinear adaptivity supports static analyses with large deformation (**NLGEOM**,ON). Based on defined nonlinear adaptivity criteria, it allows splitting or remeshing for the following elements: SURF153, SURF154, TARGE169, TARGE170, CONTA172, CONTA173, CONTA174, PLANE182, PLANE183, and SOLID285. Any other element types can also be included in the model.

For more information, see Mesh Nonlinear Adaptivity in the *Advanced Analysis Guide*.

2.1.2.3. Enhanced Ocean Loading

It is now easier to apply ocean loading to a structure, as ocean-loading input is simpler and assumes a single ocean environment.

Ocean loading support has been added for the SURF154 3-D structural surface-effect element, the LINK180 3-D spar (or truss) element, and the BEAM188 and BEAM189 beam elements.

Internal fluid mass and external insulation can now be accounted for when using pipe sections referenced by the ELBOW290 element.

For more information, see Applying Ocean Loads in the *Basic Analysis Guide*.

2.1.2.4. Section Integration for Pipes

A new section-integration scheme for current-technology pipe elements has been implemented. In prior releases, each section was assumed to be an assembly of a predetermined number of nine-node cells, with each cross-section cell having four integration points. Now, each section is assumed to be an assembly of a predetermined number of cells having any number of nodes and is numerically integrated. For more information, see the documentation for the PIPE288 and PIPE289 elements.

2.1.2.5. Arc-Length Method

The arc-length algorithm, a tool for solver unstable problems such as nonlinear buckling, has been enhanced. It is now based on the Crisfield theory. The primary differences between this algorithm and the one used in previous releases are:

- The Crisfield method is less sensitive to the initial number of substeps (**NSUBST**).
- Driftback (when the analysis retraces its steps along the load-deflection curve) is generally avoided.

The new method supports:

- MPC bonded contact
- Values of time (**TIME**) different from one
- Elements with mixed formulation
- Non-zero displacement boundary conditions
- Models with mixed solid and shell elements
- Tabular loads as a linear function of time
- Remote loads
- Distributed memory processing

2.1.3. Material and Fracture Modeling

Release 15.0 includes the following enhancements to material modeling technology used in structural analyses:

Some material properties are not available via the material property menus of the GUI. For a list of such material properties, see GUI-Inaccessible Material Properties in the *Material Reference*.

2.1.3.1. Material Force Evaluation

Support is now available for material force (also known as configurational force) calculation, used primarily for analyzing material defects such as dislocations, voids, interfaces and cracks. Material force refers to a force exerted by a body on any such imperfections within the body. It is calculated in the reference configuration based on a stress-like tensorial quantity, the energy-momentum tensor (Eshelby stress tensor).

The material force calculation is performed only for cracks in homogeneous materials.

The following elements support material force evaluation: PLANE182, PLANE183, SOLID185, SOLID186, and SOLID187.

For more information, see Material Force in the *Structural Analysis Guide*.

2.1.3.2. Material Curve Fitting

The material curve-fitting tool now allows you to include the power hardening law option for the nonlinear isotropic hardening specification (**TB**,NLISO,,,POWER) to curve fit plasticity data with Chaboche kinematic hardening. For more information, see Chaboche Material Curve Fitting in the *Material Reference*.

Also, the curve-fitting tool now allows you to input experimental data in the frequency domain to fit the Prony viscoelasticity material model. The tool also supports temperature dependency, allowing you to generate multiple sets of Prony parameters at different temperatures or by using the shift functions. For more information, see Viscoelastic Material Curve Fitting in the *Material Reference*.

2.1.3.3. Material Damage

In addition to the existing progressive damage evolution method based on simple material stiffness reduction (MPDG), the progressive damage model (**TB**,DMGE) now supports a method based on continuum damage mechanics (CDM). With the new method, damage variables increase gradually based on the energy amounts dissipated for the various damage modes. For more information, see Material Damage in the *Material Reference*.

2.1.3.4. T-Stress Calculation for Fracture Analysis

An additional tool is now available for evaluating fracture mechanics parameters. Besides support for stress-intensity factor, energy-release rate, and J-integral calculations, support is now available for calculating T-stress, the stress acting parallel to the crack faces. For more information, see T-Stress and Numerical Evaluation of Fracture Mechanics Parameters in the *Structural Analysis Guide*.

2.1.3.5. Shape Memory Alloy (SMA)

Support for SMA material models with the memory-effect option (**TB**,SMA,,,,MEFF), previously available only for elements with a 3-D stress state, is now available for current-technology beam elements, shell elements, and elements with plane stress. For more information, see Shape Memory Alloy (SMA) and Material Model Element Support in the *Material Reference*.

2.1.3.6. Cohesive Zone Material (CZM)

Interface elements (INTER*nnn*) now allow viscous regularization to be used for stabilizing interface delamination. For more information, see Viscous Regularization for Cohesive Zone Material in the *Material Reference*.

Support is now available for creating user-defined cohesive zone materials via the new `UserCZM` subroutine. The subroutine supports interface elements (INTERnnn) only, and state variables can be used. For more information, see User-Defined Cohesive Material Law in the *Programmer's Reference*.

2.1.3.7. Chaboche Model with Creep

It is now possible to model Chaboche nonlinear kinematic hardening with implicit creep. A Chaboche model with creep can itself be combined with other material models, such as nonlinear power law isotropic hardening, anisotropic plasticity, and multilinear isotropic hardening. For more information, see

Material Model Combinations in the *Material Reference* and Material Model Combination Examples in the *Structural Analysis Guide*.

2.1.3.8. Plasticity Yield Stress for Beams and Shells

Plasticity yield stress output (NL:SEPL) is now available for the following current-technology beam and shell elements: SHELL181, PLANE182, PLANE183, SOLID185 (homogenous and layered forms), SOLID186 (homogenous and layered forms), SOLID187, BEAM188, BEAM189, SOLSH190, and SHELL281.

2.1.3.9. Strain Energy Density Output

The strain energy density can now be output via the following commands: **ESOL**, **ETABLE**, **PLESOL**, **PRNSOL**, **PRESOL**, **PRNSOL**.

2.1.3.10. Field Variables

Support for user-defined field-dependent material models and functionally graded composite materials has been expanded from elasticity and thermal expansion to include Extended Drucker-Prager (**TB**,EDP), hyperelasticity (**TB**,HYPER), rate-dependent plasticity (**TB**,RATE), creep (**TB**,CREEP) and plasticity (**TB**,PLASTIC) material models.

Subroutine `UserFld` is now available for editing field-variables initialized via the node-based **INISTATE** command and referenced by material tables defined via the **TB** and **TBFIELD** commands. User-defined field variables can therefore be much more powerful, as you can query internal variables and update the field variables during solution at each integration point.

Specified material parameters (**TBDATA**) can now be interpolated (**TBIN**) in both the linear scale and natural log scale. Logarithmic interpolation is available for material table data specified as a function of any single field variable type.

For more information, see Understanding Field Variables in the *Material Reference*.

2.1.4. Linear Dynamics

Release 15.0 includes the following enhancements for structural analyses involving linear dynamics:

2.1.4.1. Mode Selection

After the extraction of the eigensolutions in a modal analysis, you can use mode selection to reduce the number of expanded solutions. It is particularly useful if a mode superposition analysis is intended. The selection is based on user input, modal effective mass, mode coefficient, or the DDAM procedure. For more information, see Using Mode Selection in the *Structural Analysis Guide*.

2.1.4.2. Structural Damping in a QRDAMP Modal Analysis

Modal analysis is now supported with the QRDAMP eigensolver when structural damping is present. For more information, see QR Damped Method in the *Mechanical APDL Theory Reference*.

2.1.4.3. Structural Damping in a Harmonic Analysis

Harmonic (**ANTYPE**,HARMIC) analyses using the FULL, MSUP, or VT method now support a constant structural (or hysteretic) damping coefficient specified with the **DMPSTR** command. For more information, see Damping in the *Structural Analysis Guide*.

2.1.4.4. Mode-Superposition Harmonic Analysis for Cyclic Structures

Harmonic analyses of cyclic structures can now be performed using the mode-superposition method, including support for prestressed structures using linear perturbation. Loading occurs via engine order excitation, and mapping of complex pressures from a CFD analysis is supported.

Full postprocessing is available, including expansion of results (**/CYCEXPAND**) in POST1 and viewing results for any sector in POST26. Because only the base sector results are kept on the MODE and RST files, disk space requirements are reduced by half.

For more information, see Solving a Mode-Superposition Harmonic Cyclic Symmetry Analysis in the *Advanced Analysis Guide*.

2.1.4.5. Closely Spaced Modes (CSM) Method for Dynamic Design Method (DDAM)

The closely spaced modes (CSM) method is now available for analyses using the dynamic design analysis method (DDAM). For more information, see Closely Spaced Modes (CSM) Method in the *Mechanical APDL Theory Reference*.

2.1.4.6. Precise Mass Summary

A precise mass summary is now output for all analyses of 3-D models. For more information, see Mass Related Information in the *Mechanical APDL Theory Reference*.

2.1.4.7. Damping Support for Component Mode Synthesis (CMS) Superelement Generation

Damping is now supported when generating a superelement using component mode synthesis. For more information, see Component Mode Synthesis (CMS) in the *Mechanical APDL Theory Reference*.

Damping ratios can also be added to the CMS reduced damping matrix for the fixed interface method.

The gyroscopic matrix of an axisymmetric rotating structure can now be included in the generation pass of a CMS superelement to perform rotordynamics modal and harmonic analyses.

2.1.4.8. Results Comparison with RSTMAC

The **RSTMAC** command can now compare results (from `.rst` or `.rstp` file) against experimental data in a universal format file. For more information, see Comparing Nodal Solutions From Two Models or From One Model and Experimental Data (RSTMAC) in the *Basic Analysis Guide*.

2.1.4.9. Mode Superposition Harmonic Analysis Based on the Unsymmetric Eigensolver

A mode superposition harmonic analysis can now be performed following an unsymmetric modal analysis (**MODOPT**,UNSYM). This procedure is of particular interest when running a large fluid-structure interaction (FSI) harmonic analysis that includes structural loads. For more information, see Mode-Superposition Harmonic Analysis in the *Structural Analysis Guide* and Harmonic Analysis Settings in the *Acoustic Analysis Guide*.

2.1.4.10. Brake Squeal Analysis Using Superelements

In a brake squeal analysis, superelements (generated by substructuring or CMS) are now supported. They can be used for the static and/or the rotating parts (**CMROTATE**). See Brake Squeal (Prestressed Modal) Analysis in the *Structural Analysis Guide* for a description of the brake squeal analysis procedure.

2.2. Coupled Physics

Release 15.0 includes the following enhancements for structural analyses involving coupled physics:

2.2.1. Coupled-Field

This release includes the following enhancements in the area of coupled-field analysis:

2.2.1.1. Multiframe Restart

Multiframe restart is now supported for nonlinear static and transient electric, magnetic, diffusion and coupled-field analyses. For more information, see the documentation for the **RESCONTROL** command.

2.2.1.2. Linear Perturbation

Support for linear perturbation is now available for the piezoelectric analysis option of the coupled-field solid elements PLANE223, SOLID226, and SOLID227. For more information, see Linear Perturbation Analysis in the *Structural Analysis Guide* and the documentation for the **ANTYPE** and **PERTURB** commands.

2.2.1.3. Nonlinear Stabilization

Support for nonlinear stabilization is now available for coupled-field solid elements PLANE223, SOLID226, and SOLID227 in a coupled-field analysis with structural degrees of freedom. For more information, see the documentation for the **STABILIZE** command.

2.2.2. Low-Frequency Electromagnetics

This release includes the following enhancements in the area of low-frequency electromagnetics:

2.2.2.1. Velocity Effects

Velocity effects can now be taken into account in a static, time-harmonic, or time-transient electromagnetic analysis using current-technology electromagnetic solid elements PLANE233, SOLID236, and SOLID237. For more information, see Velocity Effects in the *Low-Frequency Electromagnetic Analysis Guide* and the documentation for the **BF** command (VELO label).

2.2.2.2. Linear Perturbation

Support for linear perturbation is now available for the electromagnetic analysis option of the electromagnetic solid elements PLANE233, SOLID236, and SOLID237. For more information, see Electromagnetic Linear Perturbation Analysis in the *Low-Frequency Electromagnetic Analysis Guide* and the documentation for the **ANTYPE** and **PERTURB** commands.

2.2.2.3. Magnetic Energy, Co-energy, Apparent, and Incremental Energy

Support for magnetic energy, co-energy, apparent, and incremental energy calculation is now available for the PLANE233, SOLID236, and SOLID237 electromagnetic solid elements. For more information, see the documentation for those elements.

2.2.3. Acoustics

This release includes the following enhancements in the area of acoustic analysis.

For further information, see the *Acoustic Analysis Guide*.

2.2.3.1. Sophisticated Acoustic Materials

The Delany-Bazley equivalent fluid model for perforated material is now supported. The Miki equivalent fluid model for perforated material is also supported, allowing for improved accuracy at low frequency.

Support has been added for the equivalent fluid model defined with complex mass density and sound speed, and for the equivalent fluid model defined with a complex propagating constant and impedance.

The equivalent fluid model for perforated material now supports frequency-dependent input parameters. Frequency-dependent acoustic material properties have been added.

2.2.3.2. Acoustic Loads and Boundary Conditions

A two-port admittance transfer matrix model connecting to the acoustic domain or FSI interface is available to trim sophisticated structures or materials.

Surface impedance can be frequency-dependent. Normal surface velocity (acceleration) can be frequency- or time-dependent. Velocity or acceleration defined by six components can be frequency- or time-de-

pendent. An absorption coefficient can now be specified (**SF**,,ATTN) for frequency-dependent acoustic surface loading.

2.2.3.3. Visco-Thermo Acoustics

The boundary layer impedance (BLI) model is now available for acoustic wave propagation for visco-thermo fluid.

The low reduced frequency (LRF) model is available for wave propagation in a visco-thermo fluid domain for thin layer, rectangular, and circular pipe or slit.

2.2.3.4. Acoustic Analysis

Linear perturbation analysis is now available for nonlinear static prestressed structural solution and coupled FSI linear acoustics with morphed mesh.

To control file sizes, an element key option (KEYOPT(1)) is now available for defining the coupled symmetric algorithm in a modal or harmonic analysis using FLUID30, FLUID220, or FLUID221 elements.

Acoustic elements now support various systems of units, including user-defined units.

Octave frequency bands, or a general logarithm frequency span, can be defined (**HARFRQ**) in an acoustic harmonic analysis.

Efficient one-way coupling from the structure to acoustics with multiple frequencies or time steps for conforming FSI interface mesh (**ASIFILE**).

You can now perform a multiframe restart for static and transient analyses with acoustic degrees of freedom.

A general one-way coupling is now available from the structure to acoustics using Workbench External Data with mapping for a nonconforming FSI interface mesh. See the release notes for ANSYS Workbench.

2.2.3.5. Acoustic Postprocessing

A-weighted sound pressure level (dBA) output is now available in postprocessing.

Sound power parameters, including input sound power level, output sound power level, return loss, absorption coefficient and transmission loss, are available in postprocessing (**SPOWER** and **PLST**).

For more information, see Postprocessing Acoustic Analysis Results in the *Acoustic Analysis Guide*.

2.2.3.6. Acoustic Commands

Several new commands for use in an acoustic analysis have been added (**ASCRES**, **ASIFILE**, **ASOL**, **SPOWER**). For more information, see Commands Used in an Acoustic Analysis in the *Acoustic Analysis Guide*.

2.2.4. Thermal

This release includes the following enhancements for structural analyses involving radiation analysis:

2.2.4.1. Thermal Analysis with Radiosity

An iterative QUASI option (**THOPT**,QUASI) is now available for transient thermal analyses where radiosity is present. The QUASI solution method can now solve for temperature and radiosity flux using either a multipass or an iterative process. For more information, see Solution Algorithms Used in Transient Thermal Analysis in the *Thermal Analysis Guide*.

2.2.4.2. View Factor File

You can now specify formatting options for binary and ASCII view factor files. File-compression options are available for binary files, and low- or high-precision options are available for ASCII files. For more information, see the documentation for the **VFOPT** command.

2.2.4.3. User-Defined Heat Generation Loading

Support is now available for defining custom heat generation loading for the MASS71 thermal mass element via the `usrefl` subroutine. For more information, see Subroutine `usrefl` (Changing Scalar Fields to User-Defined Values) in the *Programmer's Reference*.

2.2.5. Unidirectional Pressure Mapping

The new **/MAP** processor facilitates the import and mapping of pressure data (real or complex) from a CFD analysis for subsequent use in a static, transient, or harmonic analysis. The pressure data can be from a CFX transient blade row analysis, CFD-POST output, a generic text file, or a CSV file.

The pressure data is mapped to specified target nodes and written to a text file as **SFE** commands. Additional functionality includes plotting the source and target geometries and pressures, aligning the geometries, and modifying the mapped pressures.

For more information, see Unidirectional Pressure Mapping: CFD to Mechanical APDL in the *Coupled-Field Analysis Guide*.

2.3. Solvers

Release 15.0 includes the following new enhancements that improve solution procedures and features:

2.3.1. Sparse Solver Enhancements

The sparse solver has been improved to perform better detection and handling of singular (or nearly singular) matrices. Typically, such matrices indicate modeling issues such as rigid body motion or the introduction of hourglass modes for some element formulations (for example, a course mesh of higher-order brick elements with reduced integration). The new logic may result in some poorly constructed models no longer solving, when they may have solved in prior releases.

In a nonlinear analysis using the Newton-Raphson method, the error detection for a singular matrix is performed at the first iteration of the first load step. Review the message(s) and take corrective action to fix the model accordingly. See Singular Matrices in the *Basic Analysis Guide* for a list of conditions

that may trigger this message and suggestions on how to isolate the portion of the model that may not be properly constrained.

It is possible to bypass the error detection logic for singular matrices (**PIVCHECK**,OFF).

2.3.2. Distributed Solver Enhancements

The following enhancements are available for Distributed ANSYS:

- The domain decomposition algorithm has been improved. Better performance and scaling (particularly at higher core counts) may be noticed, especially for problems with a large number of contact pair definitions.
- Support is now available for thermal transient analyses which use the quasi matrix reform method (**THOPT**,QUASI).
- The full matrix file (`Jobname.FULL`) is now automatically combined after a distributed modal analysis that uses the Subspace eigensolver. See the **DMPOPTION** and **COMBINE** commands for more information.
- The algorithm for handling multipoint constraint equations and coupling equations (including related usages by contact and MPC184 elements) has been improved to automatically identify slave degrees-of-freedom and remove redundancy in global solution matrices.
- For performance reasons, the program no longer allows Distributed ANSYS to be launched with more processors (per simulation) than the number of physical CPU cores that exist in the system.

2.3.3. GPU Acceleration Enhancements

The following enhancements are available for the GPU Accelerator capability:

- Intel Xeon Phi coprocessors (models 7120, 5110, and 3120) are now supported.
- The latest generation of NVIDIA GPUs (code-named Kepler) is now supported.
- The sparse solver performance has been significantly improved when running on the latest NVIDIA GPUs (code-named Kepler).

2.3.4. Modal Eigensolver Enhancements

The subspace mode extraction method (**MODOPT**,SUBSP) is now available for modal analysis. This eigensolver is based on an iterative algorithm and is applicable to problems in which the system matrices are symmetric. It performs well when the goal is to obtain a moderate number of eigenvalues on large models run in distributed parallel mode (Distributed ANSYS). Memory management and Sturm sequence check options can be set for the Subspace eigensolver (**SUBOPT**).

2.3.5. Other Solver Changes and Enhancements

The performance of the program has been enhanced when running on the current generation of Intel Xeon processors (code-named Sandy Bridge) due to the use of the Intel 12.1 compiler. In some cases, the performance can be up to 30 percent faster than the previous release.

The performance of the sparse solver has been enhanced when solving harmonic analyses which use the full method (**HROPT**,FULL). In some cases, the solver performance can be up to 40 percent faster than the previous release.

The QR Damp eigensolver calculations have been improved to use shared-memory parallelism whenever possible.

2.4. Commands

This section describes changes to commands at Release 15.0.

Some commands are not accessible from menus and are available via the command input area or batch file input only. The documentation for each command indicates whether or not a menu path is available for that command operation.

2.4.1. New Commands

The following new commands are available in this release:

- **ASCRES** -- Specifies the output type for a acoustic scattering analysis.
- **ASIFILE** -- Writes or reads one-way acoustic-structural coupling data.
- **ASOL** -- Activates a specified acoustic solution.
- **CNTR** -- Redirects contact pair output quantities to a text file.
- **CYCFILES** -- Specifies the data files where results are to be found for a cyclic harmonic mode-superposition analysis.
- **CYCFREQ** -- Specifies solution options for a cyclic symmetry mode superposition harmonic analysis.
- **DMPSTR** -- Sets a constant structural damping coefficient.
- **FTYPE** -- Specifies the file type and the pressure type for the subsequent import of source points and pressures.
- **MAP** -- Maps pressures from source points to target surface elements.
- **/MAP** -- Enters the mapping processor.
- **MODSELOPTION** -- Specifies the criteria for selecting modes to be expanded.
- **NLADAPTIVE** -- Defines the criteria under which the mesh is refined or modified during a nonlinear solution.
- **OCZONE** -- Specifies the type of ocean zone data to follow.
- **PLGEOM** -- Plots target and source geometries.
- **PLMAP** -- Plots target and source pressures.
- **PLST** -- Plots sound power parameters vs. frequency.
- **RCYC** -- Calculates cyclic results for a mode-superposition harmonic solution.

- **READ** -- Reads coordinate and pressure data from a file.
- **SPOWER** -- Calculates sound power parameters.
- **SUBOPT** -- Specifies subspace (SUBSP) eigensolver options.
- **TARGET** -- Specifies the target nodes for mapping pressure onto surface effect elements.
- **TBIN** -- Sets parameters used for interpolation of the material data tables.
- **WRITEMAP** -- Writes interpolated pressure data to a file.

2.4.2. Modified Commands

The following commands have been enhanced or otherwise modified in this release:

- **ANHARM** -- Produces an animated sequence of time-harmonic results or complex mode shapes. This command can now be used after a CMS analysis use pass or expansion pass for modal and harmonic analyses.
- **BF** -- Defines a nodal body force load. In an electromagnetic analysis, the new VELO label defines translational or angular nodal velocities. In an acoustic analysis, *Lab* = EF no longer defines the arbitrary velocity; use *Lab* = VELO instead.
- **BFE**, **BFK**, **BFL**, **BFV** -- Body force loading commands. Support for high-frequency electromagnetic analysis has been undocumented.
- **COMBINE** -- Combines distributed memory parallel (Distributed ANSYS) files. You can now use this command to combine the full matrix file (.FULL) after a distributed solution.
- **CYCOPT** -- Specifies solution options for a cyclic symmetry analysis. The new MSUP option controls how results are written from a modal cyclic symmetry analysis for use in a subsequent mode superposition-based analysis.
- **DDOPTION** -- Sets the domain decomposer option for Distributed ANSYS. The default decomposition algorithm has changed from GREEDY to AUTO. The legacy *ContKey* argument has been undocumented.
- **DMPOPTION** -- Specifies distributed memory parallel (Distributed ANSYS) file combination options. You can now use this command to suppress automatic file combination for the full matrix file (.FULL) during a distributed solution.
- **ESOL** -- Specifies element data to be stored from the results file. The command now supports strain energy density output.
- **ETABLE** -- Fills a table of element values for further processing. The command now supports strain energy density output. A new output option controlling how element table data is stored has also been added; besides storing the averaged element centroid value of the specified item component (the default behavior), the command can now store the minimum or maximum element nodal value of the specified item component.
- **FDELE** -- Deletes force loads on nodes. The new *Lkey* argument can be used to apply the current relative displacement value to the specified degree of freedom in addition to deleting the force.
- **HFANG**, **HFSYM** -- Definition commands for acoustic analysis. Support for high-frequency electromagnetic analysis has been undocumented.

- **MODOPT** -- Specifies modal analysis options. The subspace eigensolver (*Method* = SUBSP) is now available.
- **MXPAND** -- The *ModeSelMethod* argument has been added to allow users to specify the number of modes to expand based on the modal effective masses, the mode coefficients, or the Dynamic Design Analysis Method (DDAM) procedure. (For more information, see Mode Selection Based on DDAM Procedure in the *Structural Analysis Guide*.)
- **NLDIAG** -- Sets nonlinear diagnostics functionality. When contact diagnostic information is requested, total volume loss due to wear is now included in the `Jobname.CND` file.
- **NLHIST** -- Specifies result items to track during solution. Total volume loss due to wear can now be tracked for contact pairs.
- **NRLSUM** -- The new *LabelCSM* argument is used to specify the use of the closely spaced modes (CSM) method for DDAM spectrum analysis. See Closely Spaced Modes (CSM) Method in the *Mechanical APDL Theory Reference*

  for more information.
- **OUTRES** -- Controls the solution data written to the database. A new option allows you to specify the number of user-defined state variables to be written to the results file.
- **PLESOL** -- Displays the solution results as discontinuous element contours. The command now supports strain energy density output.
- **PLNSOL** -- Displays results as continuous contours. The command now supports strain energy density output.
- **PRESOL** -- Prints the solution results for elements. The command now supports strain energy density output.
- **PRNSOL** -- Prints nodal solution results. The command now supports strain energy density output.
- **QRDOPT** -- The new *CMCCoutKey* argument allows complex modal contribution coefficients to be output to the `jobname.CMCC` file.
- **REMESH** -- Specifies the starting and ending remeshing points, and other options, for rezoning. The command now supports 3-D mesh splitting.
- **RESWRITE** -- The new *cFlag* argument is available for flagging complex results.
- **RSTMAC** -- Calculates modal assurance critera (MAC) and matches nodal solutions from two results files or from one results file and one universal format file. Results (`.rstp` or `.rst` file) can now be compared against experimental data in universal format files (`.unv`) files. The new *UNVscale* argument supports the scaling of nodal coordinates when *File2* is a universal format file (`.unv`). *KeyMass* argument allows the inclusion of the mass matrix diagonal in calculations.
- **SECDATA** -- Describes the geometry of a section. The command now includes data to describe the new BOLT section for bolt-thread modeling.
- **SECTYPE** -- Associates section type information with a section ID number. A new contact section, *Subtype* = BOLT, allows you to define a bolt thread surface in a contact analysis.
- **SF** -- In an acoustic analysis, *Lab* = CONV no longer defines the attenuation coefficient. Use *Lab* = ATTN instead.

- **TB** -- Activates a data table for material properties or special element input. A new WEAR label is available for modeling contact surface wear. Also, there is now no limit on the number of temperature points (*NTEMP*) for coefficient of friction (FRIC) and user-defined contact interaction (INTER) specifications.

- **TBFIELD** -- Defines values of field variables for material data tables. Creep (**TB**,CREEP) and plasticity (**TB**,PLASTIC) material models are now supported, for which temperature-dependent and/or user-defined field-variable-based material data tables can be defined. Contact surface wear (**TB**,WEAR) is also supported, for which temperature-dependent and/or time-dependent material data tables can be defined.

- **THOPT** -- Sets the nonlinear transient thermal solution option. A new option is available for specifying whether the FULL or QUASI solution method should use a multipass or an iterative algorithm. The legacy LINEAR solution method has been undocumented.

- **USRCAL** -- Allows user-solution subroutines to be activated or deactivated. Two new arguments have been added: UTIMEINC for overriding the program-determined time step, and UCNVRG for overriding the program-determined convergence.

- ***VFILL** -- Fills an array parameter. A new RIGID argument generates the rigid body modes with respect to the reference point coordinates.

- **VFOPT** -- Specifies options for the view factor file. You can now specify file-compression options for a binary file (*Filetype* = BINA), and low- or high-precision options for an ASCII file (*Filetype* = ASCI).

2.4.3. Undocumented Commands

The following capabilities have been undocumented at this release:

- FLOTRAN analysis

- High-frequency electromagnetic analysis

The following legacy commands have therefore been undocumented:

FLDATA1-40	HFPCSWP	MSDATA	MSVARY	QFACT
FLOCHECK	HFPOWER	MSMASS	PERI	SPADP
FLREAD	HFPORT	MSMETH	PLFSS	SPARM
FLOTRAN	HFSCAT	MSMIR	PLSCH	SPFSS
HFADP	ICE	MSNOMF	PLSYZ	SPICE
HFARRAY	ICEDELE	MSPROP	PLTD	SPSCAN
HFDEEM	ICELIST	MSQUAD	PLTLINE	SPSWP
HFEIGOPT	ICVFRC	MSRELAX	PLVFRC	
HFEREFINE	LPRT	MSSOLU	/PICE	
HFMODPRT	MSADV	MSSPEC	PLWAVE	
HFPA	MSCAP	MSTERM	PRSYZ	

Because input for ocean loading has been simplified in this release, the SOCEAN command has also been undocumented. For more information about ocean loading, see Applying Ocean Loads in the *Basic Analysis Guide*.

For information about commands that have been undocumented in prior releases, see the archived release notes on the ANSYS Customer Portal (p. xi).

2.5. Elements

This section describes changes to elements at Release 15.0.

Some elements are not available from within the GUI. For a list of those elements, see GUI-Inaccessible Elements in the *Element Reference*.

2.5.1. Modified Elements

The following elements have been enhanced in this release:

- BEAM188, BEAM189 -- These beam elements now support ocean loading.
- CONTA171, CONTA172, CONTA173, CONTA174, CONTA175 -- These contact elements now support bolt thread modeling via the BOLT contact section type. In addition, they now support modeling contact surface wear.
- CONTA176, CONTA177 -- The line-to-line and line-to-surface contact elements now support a contact traction-based option that offers better convergence and solution results than the original force-based option.
- ELBOW290 -- Internal fluid and external insulation can now be accounted for when using this element.
- FLUID30, FLUID220, FLUID221 -- Key options KEYOPT(2) = 2 and 3 are no longer supported for defining a symmetric coupled modal or symmetric coupled harmonic analysis, respectively. To control file sizes, KEYOPT(1) is now available when defining the coupled symmetric algorithm in a modal or harmonic analysis using these elements.
- LINK180 -- This 3-D spar (or truss) element now supports ocean loading.
- MATRIX50 -- In transient dynamic analyses that include large deflections (**NLGEOM**,ON), the inertia calculations for the superelement have been improved in order to achieve more accurate results, especially when simulating rotating shafts.
- PLANE223, SOLID226, SOLID227 -- These coupled-field solid elements now support nonlinear stabilization. Support for linear perturbation for piezoelectric analysis has also been added.
- PLANE233, SOLID236, SOLID237 -- These electromagnetic solid elements can now take velocity effects into account in a static, time-harmonic, or time-transient electromagnetic analysis. Support for linear perturbation has also been added. New output labels (MENE, COEN, AENE, IENE) have been added to account for magnetic energy, co-energy, apparent energy and incremental energy, respectively.
- PIPE288, PIPE289 -- A new section-integration scheme has been implemented for these pipe elements. Support for internal fluid and external insulation has been added. Additional SMISC output is now available for the layered versions of the elements.
- SOLID90 -- For transient thermal solutions using the quasi-iterative option (**THOPT**,QUASI) with this thermal solid element, the program no longer removes the midside nodes from any face with a convection load.
- SURF153, SURF154 -- These surface-effect elements now support rezoning and mesh nonlinear adaptivity.

2.5.2. Undocumented Elements

The following legacy elements have been undocumented at this release:

HF118
HF119
HF120
FLUID141
FLUID142

For information about other elements that have been undocumented in prior releases, see the archived release notes on the ANSYS Customer Portal (p. xi).

2.6. Documentation

ANSYS, Inc. continues to refine the Mechanical APDL documentation set. To that end, the following changes and enhancements to the documentation have occurred with this release:

2.6.1. *Technology Demonstration Guide*

The following example problems have been added or updated in the *Technology Demonstration Guide*:

2.6.1.1. Elastoplastic Creep Analysis of Lead-Free Solder Bumps

A thermo-mechanical analysis of lead-free solder bumps subjected to cyclic thermal loading. The problem shows how to obtain implicit creep material constants using experimental data, uses creep and plasticity material models to simulate viscoplastic behavior, and determines accumulated creep strain due to thermal loading.

2.6.1.2. Large-Deformation Neo-Hookean Analysis (via UserMat Subroutine)

Formulates a 3-D large deformation, hyperelastic material to demonstrate the user-defined material capability in nonlinear geometry analyses. Details are given for stress and material tangent calculations and formulation in a co-rotated frame, as well as conversion of tensor quantities to Voigt notation. The problem also serves as a usage example of the `UserMat` subroutine, a user-programmable feature for developing a custom, constitutive model to define the stress-strain behavior of a material.

2.6.1.3. Evaluation of Mixed-Mode Stress-Intensity Factors for 3-D Surface Flaws

Demonstrates the linear elastic fracture mechanics of 3-D structures. Besides showing how fracture mechanics can be used to evaluate mixed-mode stress-intensity factors and J-integrals, the problem has been updated to include T-stress evaluation. Analyses of a simple semicircular surface flaw in a rectangular block and a warped flaw along a tubular joint are discussed.

2.6.1.4. VCCT-Based Crack Growth Simulation of a Composite Laminated T-Joint

A simulation of interfacial crack growth occurring in a laminated T-Joint using the virtual crack closure technique (VCCT). The simulation involves growing an existing crack along a predefined path using interface elements.

2.6.1.5. Dynamic Simulation of a Nuclear Island

A power spectral density (PSD) analysis of a nuclear island component of a nuclear power plant. The problem shows the analysis methods available to simulate the response of the nuclear island to a seismic event, taking into account motion incoherency and wave-passage effects.

2.6.1.6. Bolt Thread Simulation

A comparison of three modeling techniques for bolt threading: true threading simulation, the bolt section method, and the MPC method. The comparison demonstrates that the bolt section method offers accurate results and a while requiring significantly less computational time.

2.6.1.7. Wire Bonding Ultrasonic Transducer

Simulates electrical excitation of an ultrasonic transducer used for wire bonding applications. The model includes piezoelectric material definition as well as prestressed modal and harmonic response analyses.

2.6.2. New *Acoustic Analysis Guide*

Documentation for acoustic analysis has been extensively updated. A new *Acoustic Analysis Guide* has been added to the product documentation set.

2.6.3. *Feature Archive*

Legacy features, commands, elements, and theory information continue to move to the *Feature Archive*. While ANSYS, Inc. continues to support these legacy capabilities for the immediate future, some may be undocumented in future releases. Consider moving to their recommended replacements.

2.6.4. Documentation Updates for Programmers

The following documentation updates are available for programmers:

2.6.4.1. Routines and Functions Updated

Routines and functions documented in the *Programmer's Reference* have been updated to reflect the current source code. To see specific changes in a file, ANSYS, Inc. recommends opening both the old and current files (using a text editor that displays line numbers), then comparing the two to determine which lines have changed. You can copy the updated files to your system by performing a custom installation of the product.

2.6.4.2. Subroutines Added

The following subroutines have been added to the *Programmer's Reference*:

`UCnvrg` - Override the program-determined convergence
`userCZM` - User-defined cohesive zone material

`userfld` - Update user-defined field variables
`userwear` - User-defined contact surface wear model
`UTimeINC` - Override the program-determined time step

2.7. Known Incompatibilities

The following incompatibilities with prior releases are known to exist at Release 15.0.

2.7.1. High-Frequency Electromagnetic Analysis

Support is no longer available for high-frequency electromagnetic analysis. Instead, ANSYS, Inc. recommends using the ANSYS HFSS product.

2.7.2. Tabular Real Constants Using X,Y,Z Primary Variables

The usage of primary variables X, Y, and Z for defining contact element real constants in tabular format has changed. In the previous release, these variables represented the current location of contact detection, causing the associated real constants to be updated as the model deformed. Now, primary variables X, Y, and Z represent the location of contact detection at the beginning of the solution (the undeformed configuration).

2.7.3. Normal Contact Stiffness

In prior releases, the internal contact stiffness in the normal direction did not vary with the user-defined penetration tolerance (contact element real constant FTOLN). In this release, the internal stiffness is inversely proportional to the user-defined penetration tolerance; that is, the tighter the tolerance, the higher the normal contact stiffness.

2.7.4. Tangential Contact Stiffness

When an absolute value is input for tangential contact stiffness (negative value input for contact element real constant FKT), this value remains unchanged throughout the analysis regardless of the contact status or the value of real constant FKOP (opening contact stiffness). In previous releases, tangential contact stiffness could change if an absolute value was input for opening contact stiffness (negative value input for FKOP).

2.7.5. Maximum Allowable Tensile Contact Pressure

The default value for maximum allowable tensile contact pressure (contact element real constant TNOP) has been reduced by a factor of 10. Used as a chattering control parameter for the Lagrange multiplier methods, TNOP now defaults to 10 percent of the force convergence tolerance divided by the contact area at contact nodes.

2.7.6. Stream Function Wave Loading

Tabular input for Dean's stream function wave loading (*KWAVE* = 3 on the **OCDATA** command for waves) has been replaced by the usual wave height, period, and depth. To facilitate a smooth transition to the new logic, however, the prior logic is still available via *KWAVE* = 14 (used with prior-release documentation describing the former usage of *KWAVE* = 3).

2.7.7. Resuming Ocean Loading Analyses

Because input for ocean loading has been simplified in this release, the SOCEAN command has been undocumented. Analyses that applied ocean loads via the SOCEAN command in prior releases cannot be resumed in this release and must be completely restarted.

For more information about ocean loading, see Applying Ocean Loads in the *Basic Analysis Guide*.

2.7.8. Acoustics

The *Lab* = EF argument on the **BF** command no longer defines the arbitrary velocity. Use *Lab* = VELO instead.

The *Lab* = CONV argument on the **SF** command no longer defines the attenuation coefficient. Use *Lab* = ATTN instead.

FLUID30, FLUID220, and FLUID221 element key options KEYOPT(2) = 2 and 3 are no longer supported for defining a symmetric coupled modal or symmetric coupled harmonic analysis, respectively. To control file sizes, KEYOPT(1) is now available when defining the coupled symmetric algorithm in a modal or harmonic analysis using those elements.

2.7.9. Preloads in Modal and Harmonic Analyses are Ignored

In modal and harmonic analyses, spring preloads, initial strains, and pretension loads are now ignored and no longer produce loads.

Chapter 3: Autodyn Release Notes

The following enhancements are available in release 15.0. Please refer to the product specific documentation for full details of the new features.

3.1. Incompatibilities and Changes in Product Behavior from Previous Releases

Release 15.0 includes new features and enhancements that result in product behaviors that differ from previous releases. These behavior changes are presented below.

- **Platform MPI Support**. For the Windows and Linux operating systems, the explicit solvers now use the default Platform MPI 9.1 version installed using the ANSYS Installer. Refer to Windows Systems using Platform MPI for more information.

3.2. New Features

The following new features are exposed in ANSYS Autodyn for Release 15.0:

- The following are now fully supported in parallel solutions:
  - **NBS Tetrahedral Elements**
  - **Trajectory Contact including Erosion and Zero or Non-Zero Shell Thickness**
  - **Remote Point/Remote Displacement**
  - **Reinforcement Beams**
  - **Breakable/Unbreakable Bonds**
  - **Breakable Spotwelds**
- **Double precision.** The Autodyn solver now always uses double precision. Using double precision will generally provide the most robust solution. There is typically only a limited increase in cpu time (on the order of 10% to 20%) and memory consumption (on the order of 10% to 30%).

Chapter 4: Aqwa Release Notes

This release of the Aqwa related products contains all capabilities from previous releases plus many new features and enhancements. The following enhancements are available in release 15.0. Please refer to the product specific documentation for full details of the new features.

4.1. Aqwa Solver Modules

The following new features provide extended capabilities in the Aqwa Solver modules:

- **64-bit solver version.** The Aqwa solver is now available in a 64-bit version, which will be installed by default on 64-bit machines. This version allows for higher limits on elements and nodes in a model.
- **Hull Drag Linearization.** Hull Drag Linearization has been added to Aqwa-Fer, which will enable the inclusion of hull drag in the dynamic behavior of a vessel where this may contribute a significant load effect.
- **Increased user-defined spectrum frequencies.** The number of frequencies allowed when specifying a user-defined wave or wind spectrum has been increased to 50.
- **Multiple simultaneous spectra in Aqwa-Naut.** Multiple simultaneous spectra can now be included in the nonlinear time-domain module Aqwa-Naut, allowing more realism in modeling of wave conditions. The multiple simultaneous wave spectra capability is now available in all the relevant modules in Aqwa.
- **Second Order Correction of Linear Irregular Waves**. A new perturbation approach for the nonlinear correction of incident wave properties has been introduced. It improves the accuracy of the wave property estimation, especially for the fluid pressure near the instantaneous wave surface. The detailed information on this approach can be found in Second Order Correction of Linear Irregular Waves in the *Aqwa Theory Manual*.

  Aqwa-Naut irregular wave analysis employs this new method, unless a **WHLS** option is added for an analysis of linear irregular waves with Wheeler stretching.

4.2. Hydrodynamic Analysis Systems

The following new features provide extended capabilities in the Hydrodynamic Analysis Workbench systems:

- **Improved Meshing.** New meshing capabilities have been exposed for the Hydrodynamic Diffraction system in Workbench that should make it easier to mesh complex shapes.

4.3. Documentation

The following documentation changes have been made for Aqwa for Release 15.0:

- **Aqwa Theory Manual.** A new Aqwa Theory Manual has been added to the Aqwa documentation set.

Chapter 5: Beamcheck Release Notes

There are no new features or enhancements in ANSYS Beamcheck in Release 15.0.

Chapter 6: Fatjack Release Notes

There are no new features or enhancements in ANSYS Fatjack in Release 15.0.

Chapter 7: ANSYS Composite PrepPost (ACP)

The following enhancements are available in release 15.0. Please refer to the product specific documentation for full details of the new features.

7.1. New Features

- Easy Submodeling for composites
- Enhanced interlaminar shear stress computation for solid composites
- Temperature dependent material model in ACP Post
- Easy modeling of interface delaminations (VCCT, CZM, Debonding) in ACP and Mechanical
- Solid model extrusion enhancements
  - Composite solid model workflows in Workbench now support the use of SOLSH190 elements
  - New extrusion methods
    - → Sandwich-wise with max. element thickness
    - → Material-wise with max. element thickness
  - Adjustable warping limit to control removal of bad elements
  - Reorganized solid model GUI
- New failure criteria: Hoffman and LaRC 04
- 1D look up table
- Geometry export of ply and laminate surfaces in STEP and IGES format
- Sharing ACP composite definitions
  - within a Workbench Project using new shared mode of ACP
  - across projects using new import/export functionality in ACP
- Reorganized 3D visualization of composite layups and analysis results in ACP
  - New plot objects can be created for the model and each solution, similar to Mechanical
  - Each plot has a user defined computation scope
  - Plot data is computed on update enabling faster ply-wise and element-wise visualizations of all results
- Other enhancements

- Improved transfer of unconnected Named Selections to ACP Edge Sets
- Edge tapers in modeling plies can now be defined by angle and offset
- ACP documentation moved to ANSYS Help

Part II: ANSYS Fluids Products

Release notes are available for the following ANSYS Fluids products:

Chapter 1: Fluent Release Notes

The following sections contain release information for ANSYS Fluent 15.0.

1.1. New Features in ANSYS Fluent 15.0

New features available in ANSYS Fluent 15.0 are listed below. Where appropriate, references to the relevant section in the User's Guide are provided.

Solver-Numerics

- Mesh interfaces between solid zones of the same material are now created as interior-type interfaces rather than coupled-wall interfaces. This improves mesh-read performance for cases with many solid-solid zone interfaces.
- A new option has been added to use Laplace coefficients for coarsening in the AMG solver. (Coarsening Parameters)
- A limiter filter can be applied with the Standard and Differentiable gradient limiters. This aims to preserve higher-order accuracy of the main flow variables by suppressing limiter intervention resulting from numerical noise while still allowing the limiter to intervene when large gradients or discontinuities appear in the solution. (Selecting Gradient Limiters in the *Fluent User's Guide*)
- You can now specify the time step size for the density based explicit solver. (User Inputs for Time-Dependent Problems)
- Second order in time can now be used with layering, remeshing, and mesh swapping to improve solution accuracy.
- The **Conservative Coarsening** AMG option for coupled equations is now fully supported. This was a beta feature in R14.5. (Coarsening Parameters in the *Fluent User's Guide*)

Solver-Meshing

- A node-based algorithm is now used for diffusion-based smoothing offering improved robustness. In addition, a new smoothing method is available which is based on the equations for a linearly elastic solid. (Linearly Elastic Solid Based Smoothing Method)
- The local face remeshing method can now remesh wedge/prism boundary layer cells adjacent to a deforming face zone. (Local Face Remeshing Method)

- The **Adjacency** dialog box, which allows you to identify, display, and rename face zones that are adjacent to selected cell zones, is now supported as a full feature. It has been enhanced with the capability to rename face zones based on a wildcard pattern. (Managing Adjacent Zones)

- A new TUI command, `mesh/redistribute-boundary-layer`, is available to redistribute the nodes in 3D boundary layer zones to enforce a desired growth rate following anisotropic adation. (Boundary Layer Redistribution)

Models

- Turbulence

  – A new subgrid-scale model is available for the large eddy simulation (LES) model, called the Wall-Modeled LES S-Omega approach. It is an enhancement of the WMLES approach. WMLES does not provide zero eddy-viscosity for flows with constant shear. Consequently, it does not allow the computation of transitional effects, and can produce overly large eddy-viscosities in separating shear layers. The new WMLES S-Omega formulation overcomes these deficiencies and provides better results for cases where the near-wall grid is composed of polyhedral prism layers.(Algebraic WMLES S-Omega Model Formulation)

  – Scale-Adaptive Simulation (SAS) can now be enabled in combination with any omega-based URANS turbulence model, with the exception of the k-kl transition model which is not compatible with the SAS formulation. (Setting Up Scale-Adaptive Simulation (SAS) Modeling)

  – Detached Eddy Simulation (DES) can now be enabled in combination with the transition SST turbulence model. (Setting Up the Detached Eddy Simulation Model)

  – Two formulations are available for limiting turbulent kinetic energy production when using two-equation turbulence models. This can reduce the excessive generation of turbulent kinetic energy in stagnation regions when using these models. (Including Production Limiters for Two-Equation Models)

  – A new laminar-turbulent transition model called the Intermittency Transition model is available. While it is similar to the Transition SST model, it has certain advantages, such as the ability to include the effects of crossflow instability. Also, the model requires the solution of only one additional transport equation, as opposed to the two that are solved for the Transition SST model. (Intermittency Transition Model)

- Heat Transfer and Radiation

  – The ability to use a UDF to define inhomogeneous anisotropic thermal conductivity for solid materials (for example, composites), where the components of the conductivity matrix are not constant and vary independently. (User-Defined Anisotropic Thermal Conductivity)

  – The ability to use multi-layer shells to model heat transfer in the normal and planar directions of walls made up of layers of different materials. Note that the multi-layer shell mode will be enabled by default; if you need to revert to the single-layer shell mode of previous versions of Fluent, you can use the `define/models/shell-conduction/multi-layer-shell?` text command. (Shell Conduction Considerations)

  – The S2S model now supports non-conformal fluid-fluid interfaces with the **Coupled Wall** option enabled, where the wall zones created participate in the S2S calculations. Similarly, non-conformal fluid-solid interfaces are now supported as well. Previous S2S implementations were only compatible with non-conformal fluid-fluid interfaces that did not involve a coupled wall.

- Combustion and Species Transport

  - A reactor network model is available for modeling detailed chemical kinetics as a network of stirred reactors. (Reactor Network Model)

  - A new spark model has been implemented that is much less sensitive to numerical parameters such as mesh and time-step size. The spark kernel growth is now more realistic because it is governed by spark flame speed models rather than the previously assumed exponential relationship. Also, the user input has been simplified for clarity and ease of use. (Spark Model)

  - An additional chemistry acceleration method, Dynamic Mechanism Reduction, is available for improving computational time. (Using Dynamic Mechanism Reduction)

  - The Eulerian and Diesel unsteady flamelet models have been extended to allow multiple flamelets. The multiple diesel unsteady flamelet model includes the ability to simulate multiple cycles in internal combustion engines using the flamelet reset capability. (Setting Up the Steady and Unsteady Diffusion Flamelet Models)

  - You can now simulate reacting flows of up to 500 chemical species.

  - The ability to define the reacting channel inlet conditions using the DEFINE_REACTING_CHANNEL_BC user-defined function has been added. (`DEFINE_REACTING_CHANNEL_BC`)

  - The ability to model Surface Reactions in the Reacting Channel. (Surface Reactions in the Reacting Channel in the *Fluent Theory Guide*)

  - The Porous Media model can now be used inside Reacting Channel. (Porous Medium Inside Reacting Channel in the *Fluent Theory Guide*)

  - You can now use the DEFINE_SOOT_OXIDATION_RATE user-defined function hook to specify soot oxidation rate. (`DEFINE_SOOT_OXIDATION_RATE` in the *Fluent UDF Manual*)

  - You can now use the DEFINE_SOOT_PRECURSOR user-defined-function hook to specify soot precursor. (`DEFINE_SOOT_PRECURSOR` in the *Fluent UDF Manual*)

  - The ability to model diffusion Flamelet Generated Manifolds in the partially premixed combustion model. (Diffusion FGMs in the *Fluent Theory Guide*)

  - You can now specify your own the backward reaction parameters. (Defining Reactions)

- Discrete Phase Model

  - Ability to calculate unsteady time statistics (mean and RMS) for discrete phase particle quantities. (Reporting of Unsteady DPM Statistics)

  - Ability to specify discrete phase density as a function of temperature promoted from beta to full-feature. (Description of the Properties)

  - Ability to use user-defined functions for discrete phase particle specific heat, density, binary diffusivity, swelling coefficient, and latent heat. (`DEFINE_DPM_PROPERTY`)

  - Ability to use composition-dependent density and/or specific heat char properties when using the multiple surface reactions combustion model. (Description of the Properties)

  - Ability to set drag and breakup laws and parameters individually for each injection. (Specifying Injection-Specific Physical Models)

  - Ability to visualize and report vectors and components of DEM collisional force, total force, and acceleration when displaying particle trajectories. (Displaying of Trajectories)

  - Ability to have DPM particle tracking performed on a separate domain when using the Hybrid method for parallel particle tracking. This offers improved load balancing and scalability. (Parallel Processing for the Discrete Phase Model)

  - DPM source term linearization is now supported as a full feature. This was a beta feature in R14.5. (Linearized Source Terms)

- Volume of Fluid

  - Ability to specify transient profiles for free surface and bottom level for all supported open channel or open channel wave boundaries. (Determining the Free Surface Level and Determining the Bottom Level)

  - Ability to specify density interpolation method at open channel pressure inlet and mass flow inlet boundaries. (Choosing the Density Interpolation Method)

  - Ability to specify a sub time step size for VOF calculations to improve convergence in time-dependent VOF simulations. (Setting Time-Dependent Parameters for the VOF Model)

  - Ability to prescribe velocity components for oblique waves as well as specify a reference direction from which wave heading angle is measured. (Modeling Open Channel Wave Boundary Conditions)

- Eulerian Multiphase Model

  - New methods for specifying/computing interphase heat-transfer coefficients in the Eulerian multiphase model. (Including Heat Transfer Effects)

  - A new parameterized formulation of the Syamlal-O'Brien drag model is available for incompressible gas-solid flows that addresses a tendency of the conventional Syamlal-O'Brien model to under/over-predict bed expansion. (Specifying the Drag Function)

  - Ability to compute Surface Integrals and Mass Flux Reports for degassing boundary conditions.

  - Adaptive time stepping is now available with the mixture and Eulerian multiphase models.

- Eulerian Wall Film Model

  - The phase change option of Eulerian wall film model is now compatible with the mixture and the eulerian-eulerian multiphase models.

  - You can now generate reports for the film heat transfer and the film mass flow at each external boundary.

- Population Balance

  - A new option to initialize bin-fractions at **velocity-inlet** boundaries according to a user-specified log-normal distribution is available when using either of the discrete population balance models. (Defining Population Balance Boundary Conditions)

Material Properties

- You can now define and edit materials in the **Create/Edit Materials** dialog box while using the single-species and multi-species NIST real gas models.

- You can now set the phase (liquid or vapor) for a specific fluid zone for the real-gas models.
- You can now specify that the binary diffusivity for discrete phase materials be calculated from the assumption of unity Lewis number (Le=1). (Description of the Properties)
- You can now set the phase (liquid or vapor) for a specific fluid zone for the real-gas models.
- You can now specify the mass diffusion coefficients using the assumption of unity Lewis number (Le=1) for all species in the mixture. (Unity Lewis Number)

Cell Zone and Boundary Conditions

- When using the **Non-Reflecting Boundary Condition** option at a **pressure-outlet** with the pressure-based solver, you can now specify whether to use **Static Pressure** or **Total Pressure** for **Backflow Pressure Specification**. (Using General Non-Reflecting Boundary Conditions)
- You can lock (or "freeze") the temperature values for all the cells in solid zones and in walls that have shell conduction enabled, so that the values do not change during further solver iterations. This option can be useful when simulating underhood transient soaking. (Locking the Temperature for Solid and Shell Zones)
- The following text user interface commands will support a wildcard (*): `define/boundary-conditions/zone-type` and `define/boundary-conditions/modify-zones/zone-type`. You may include a wildcard (*) within the zone names. With this wildcard (*) option, setting up a specified "zone type" to many zones of a given mesh becomes easier.
- You can now model anisotropic species diffusion in porous media. (Anisotropic Species Diffusion)

Mesh Morpher/Optimizer

- Ability to select multiple control points and multiple parameters when defining the scaling factor settings for the mesh morpher/optimizer. (Setting Up the Mesh Morpher/Optimizer)
- The NEWUOA optimizer has been added, which produces faster results for problems that have a large number of parameters or degrees of freedom. (The NEWUOA Optimizer)
- Ability to limit how much each parameter is allowed to deform. (Setting Up the Mesh Morpher/Optimizer)
- The processing speed for the mesh morpher/optimizer has been significantly increased (typically being 10–20 times faster than the previous release) and memory usage has been reduced, in order to allow it to more easily handle large industrial cases.
- Ability to have the solution variables set to values obtained from a data file at the beginning of every design stage. (Setting Up the Mesh Morpher/Optimizer)

Parallel Processing

- The **Conservative Coarsening** AMG option for coupled equations is now fully supported. This was a beta feature in R14.5. (Coarsening Parameters in the *Fluent User's Guide*)
- Algebraic Multigrid acceleration on General Purpose Graphics Processing Units (GPGPUs) is now supported as a full feature. This was a beta feature in R14.5. (Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver)

- Ability to have DPM particle tracking performed on a separate domain when using the Hybrid method for parallel particle tracking. This offers improved load balancing and scalability. (Parallel Processing for the Discrete Phase Model)

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

- User-defined scheme variables created with `rp-var-define` are now saved and loaded with the case file. (Defining a Scheme Variable in the Text Interface in the *Fluent UDF Manual*)

Data Import and Export

- Tecplot export now supports polyhedral meshes. (Exporting Solution Data after a Calculation)

Graphics, Postprocessing, and Reporting

- A new field variable, Cell Acoustic Courant Number, is available for compressible transient simulations using the pressure-based solver. The existing Cell Courant Number has been renamed Cell Convective Courant Number. (Field Function Definitions)

- A new option, **Average Over**, has been added for force, moment, surface, and volume monitors, which calculates a running average over a user-specified number of iterations/time steps for the selected coefficient. (Setting Up Force and Moment Coefficient Monitors in the *Fluent User's Guide*)

- The definitions have changed for various functions that are available from the **Y Axis Function** drop-down list when postprocessing time-dependent data using the Fast Fourier Transform (FFT) algorithm (Specifying a Function for the y-Axis):

  – The **Power Spectral Density** function has changed for cases when you are plotting the detailed spectrum (not the octave or 1/3-octave bands) with either **Frequency**, **Strouhal Number**, or **Fourier Mode** selected from the **X Axis Function** drop-down list. According to the new definition, the **Power Spectral Density** is normalized per frequency unit of 1 Hz, so that it is statistically invariant to the total signal length / spectrum frequency step. Consequently, in such cases its units have changed (for example, from Pa^2 to Pa^2/Hz.

    Note that the **Sound Pressure Level (dB)** function has been correspondingly changed, as it is derived from the **Power Spectral Density**.

  – The **Magnitude** function is now calculated using a more conventional definition. As a result, for $n > 0$ the **Magnitude** value is higher by a factor of the square root of 2 when compared to the value calculated by previous releases of ANSYS Fluent (that is, version 14.5 or lower).

    Note that the **Sound Amplitude (dB)** function has been correspondingly changed, as it is derived from the **Magnitude**.

- When using the Eulerian wall film model, you can generate reports for the film heat transfer rates and the mass flow rates at each external boundary.

- You can now create **Output Parameters** for surface and volume integrals using data-integrated Fluent interface. For additional information, see Overview of Defining Surface Monitors, Overview of Defining Volume Monitors, and Save Output Parameter Dialog Box.

- It is now possible to enable spectrum smoothing when postprocessing time-dependent data with the Fast Fourier Transform (FFT) algorithm, in order to suppress spurious amplitude fluctuations. (Spectrum Smoothing Through Signal Segmentation)

User Interface

- When viewing multi-selection lists, you can arrange the list in a hierarchical tree view, based on the prefixes used in the names of each item. (Multiple-Selection Lists)

- Clicking the **Cancel** button or typing **Ctrl**+**c** while running a transient simulation will now stop the calculation at the end of the current time step instead of after the current iteration. If you want to interrupt a time step you can type **Ctrl**+**c** a second time.

Workbench

- Workbench solution monitoring capability has been extended to include surface, volume, and force monitors. (Monitoring Fluent Solutions in Workbench)

- A new **Solution Monitoring** option, available in the **Properties** view for the **Solution** cell, allows you to enable solution monitoring for your current project, while the **Enable Solution Monitoring** option in the Fluent Tab of the **Option** dialog box provides you with the ability to enable solution monitoring for new Fluent systems. (Specifying Other Setup and Solution Cell Settings, Fluent)

- The **Delete Old Solutions at Start of New Calculation** option has been replaced by the **Automatically Delete Old Solutions On Start Of New Calculation**. This option is enabled by default.

- The Fluent tab of the **Options** dialog box is reorganized to group together the general options and the default options for newly created Fluent systems. Under **Default Options for New Fluent System**, the options are organized in three categories related to the Launcher, the **Setup** cell, and the **Solution** sell.

- You can now generate an output case file that allows you to bypass loading the mesh and executing the pre-set mesh operations when the **Setup** cell is edited, resulting in faster performance. This feature is particularly efficient for multiple design point simulations if the upstream mesh component has not being updated. The feature also allows you to perform setup changes more promptly. You can control the output case file generation for your current project and new Fluent-based systems using the **Generate Output Case File** option in the **Properties** view for the **Setup** cell and the **Enable Generation of Setup Output Case file** option under the **Default Options for New Fluent System** in the Fluent configuration **Options**, respectively. (Specifying Other Setup and Solution Cell Settings, Fluent)

- New data interpolation functionality provides you with the ability to automatically initialize a Fluent session using data interpolated from the current or previous simulation. This feature is particularly useful for multiple design point simulations involving geometry and mesh changes. For your current project, this functionality is implemented via the **Data Interpolation** option in the **Properties** view for the **Solution** cell. For new Fluent systems, it is implemented via the **Enable Generation of Interpolation File** option under **Solution Cell** in the **Options** dialog box. (Specifying Other Setup and Solution Cell Settings, Fluent)

Add-Ons

- Adjoint Solver

  – A new modal stabilization scheme has been added and is now the default stabilization approach. (Modal Stabilization Scheme)

  – The energy adjoint equation can now be solved for flows with heat transfer. (Fluent Adjoint Solver Module Manual)

- Battery Model

– A new Dual Potential MSMD battery model add-on is now available. The model is based on the Multi Scale Multi Domain approach and has an ability to simulate a single battery cell or complex battery packs. The model includes three submodels of different complexity (the NTGK model, the Electric Circuit model, and the Newman P2D model) to meet various design and analysis needs. (Dual-Potential MSMD Battery Model)

Beta Features

- There are also some exciting new enhancements available as beta features that you may be interested in trying out. Detailed documentation is in the *Fluent 15.0 Beta Features Manual* which is available on the ANSYS Customer Portal.

1.2. Supported Platforms for ANSYS Fluent 15.0

Platform/OS levels that are supported in the current release are posted on the ANSYS website.

1.3. New Limitations in ANSYS Fluent 15.0

The following is a list of new or recently discovered limitations known to exist in ANSYS Fluent 15.0. Where possible, suggested work-arounds are provided.

- Models

  – The junction of a wall with shell conduction enabled and a non-conformal coupled wall is not supported. Such a junction will not be thermally connected, that is, there will be no heat transfer between the shell and the mesh interface wall.

  – In some cases involving periodic boundary conditions and the DEM model, a loss of particle mass may be observed.

  – For cases with highly stretched cells, convergence difficulties may arise in parallel. In such cases, the Laplace partitioning beta feature may address the difficulties. Please refer to the Beta Features Manual for details.

- Platform support and drivers

  – On Windows 7 and later, installing ANSYS Fluent on any driver other than `C:` may result in issues arising from spaces in the pathname not getting converted to short file names. This is the result of a change in the default value for `NtfsDisable8dot3NameCreation` starting with Windows 7. If you need to install ANSYS Fluent on any drive other than `C:` you must run the following command *prior* to installing ANSYS Fluent:

    ```
    fsutil 8dot3name set <driveletter> 0
    ```

    where <*driveletter*> is the target drive letter including the colon (e.g. **`D:`**).

  – When running ANSYS Fluent on a Linux machine directly or through a remote connection (such as NICE DCV or RealVNC), some dialog boxes may open in the background and be obscured from view. If you encounter this issue, you can define the environment variable `KEEP_DIALOG_ON_TOP`. This forces new dialog boxes to open in the foreground of your display.

- Remote display

For proper mouse-button functionality with a two-button mouse when running ANSYS Fluent through a Remote Desktop Protocol (RDP) connection to a Windows 8 machine, define the environment variable `CX_WIN8_MB_TWO` on the remote machine.

- Solver
  - The AMG cycle for scalar models using the pseudo-transient method is dependent upon the order in which the models and pseudo-transient formulation are set up. Models added or changed after pseudo-transient is enabled will not use F-cycle as documented. This can lead to differences in convergence behavior depending on the sequence of set up steps. A workaround is to disable and then re-enable pseudo-transient after any changes to models and before iterating.
- Parallel processing
  - Parallel simulations involving non-conformal interfaces and node-based gradients may show a jump in residuals when restarted on a different number of processors. You use the following Scheme command to invoke an improved treatment of interface gradients that minimizes this jump. Once saved with the case file, this setting will be persistent in future solver sessions.

    ```
    (rpsetvar 'parallel/add-ext-si-bridge-nodes-to-int-or-corner? #t)
    ```

  - Newer releases of Linux are known not to have rsh installed which is used by default to spawn remote processes in parallel. This will cause remote processes to fail to start unless you either select **SSH** in the Fluent Launcher or use the `-ssh` command line option.
  - Platform MPI does not support running in parallel in an overload condition (using more cores than are physically available on a machine). For performance reasons, running in an overload condition is generally not recommended. However, if it is required to run overloaded than you can use Intel MPI (`-mpi=intel`) or OpenMPI (`-mpi=openmpi`).
- Other
  - When running Fluent from within Workbench, it is not possible to create new parameters after running calculations.
  - When using the `opengl` display driver, displaying filled contours in a 2D case will cause a crash if the surfaces selected in the Contours dialog box contain more than 2000 1D facets. As a workaround, you can use the `msw` driver (Windows) or `x11` driver (Linux) or disable **Filled** in the Contours dialog box.
  - Connecting or disconnecting a VPN network while running a Fluent simulation will result in a failure because of changes to the network interface.
  - ANSYS Fluent Meshing fails to start with `-g` or `-gu` specified when using Fluent LM. As a workaround, you can use ANSYS LM.
- For a complete list of known limitations, including those which exist from previous releases, refer to Known Limitations in ANSYS Fluent 15.0 in the *Fluent Getting Started Guide*.

1.4. Limitations That No Longer Apply in ANSYS Fluent 15.0

This section lists limitations which have existed in previous releases, but which have been removed in ANSYS Fluent 15.0.

- Models

  - The Eulerian wall film model is now compatible with moving walls and moving reference frames.

  - The S2S model now supports non-conformal fluid-fluid interfaces with the **Coupled Wall** option enabled, where the wall zones created participate in the S2S calculations. Similarly, non-conformal fluid-solid interfaces are now supported as well. Previous S2S implementations were only compatible with non-conformal fluid-fluid interfaces that did not involve a coupled wall.

  - Fluent now allows specification of Arrhenius reverse rate parameters (Chemkin keyword REV) for gas phase reactions.

- Materials

  - Access to the **Create/Edit Materials** dialog box and menus is no longer restricted when you are using the NIST real gas models.

- Solver

  - Memory for monitors is no longer allocated based on the zone id. A large value for zone id on a zone with a monitor will no longer result in significant memory overhead.

1.5. Updates Affecting Code Behavior

The sections in this chapter contain a comprehensive list of the code changes implemented in ANSYS Fluent 15.0 which may affect the ANSYS Fluent 14.5 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

- The symmetry boundary condition for flow has been corrected to include normal viscous diffusion and compute proper velocity limits. To revert to the previous behavior, execute the following commands in the TUI console:

```
(rpsetvar 'mom/symmetry-bc-v15? #f)
(rpsetvar 'recon/r15-sym-limit? #f)
```

- In previous releases, when using 2nd order in time the flow equation treated the first 2 timesteps as 1st order whereas scalar equations treated only the 1st timestep as 1st order. Both flow and scalar equations now behave consistently. Flow only cases will be unchanged. However cases which use flow together with energy, turbulence, etc. may show differences in their transient histories.

- The default setting for pressure discretization in single-phase flows is now **Second Order** instead of the **Standard** scheme used in previous releases. This change will affect only newly set up cases. Existing case file settings are respected and clicking **Default** in an existing case, or in any R15 case derived (i.e. saved) from a pre-R15 case will result in the R145 default being chosen.

- The default pressure discretization scheme for flows using the mixture multiphase model is now **PRESTO**. In addition the available methods for mixture multiphase models are now restricted to **PRESTO** and **Body Force Weighted**. Exiting case file settings will be respected and will present the previous list of available pressure discretization methods.

- For the density-based solver, pressure-outlet boundary conditions now enforce the specified pressure as a constant value by default. This is consistent with the default treatment when using the pressure-based solver. Previously, when using the density-based solver the specified pressure was enforced as a weak-

average at pressure-outlet boundary conditions. Existing case files that used the old default will be respected. New cases will be created with the new default treatment.

- A change has been made to the computation of node-based gradients at boundaries to better match experimental data in isolated cases. In general, no significant differences have been observed in most of the experimental cases evaluated, nor in most regression test matrix cases. However, a TUI command has been added to revert to the legacy node-based-gradient treatment if desired:

  ```
  solve/set/nb-gradient-boundary-option? no
  ```

- For robust startup from **Standard Initialization** with pressure schemes using **Linear**, **Body Force Weighted**, or **Second order**, the solver switches internally to PRESTO! for the first iteration only. Thereafter, the user/case specified pressure scheme is used. This fix was found to be especially useful for zero velocity initial conditions and it helps prevent the development of unphysical pressure fields. This fix is controlled via rpvar `pressure/presto-at-startup?`, this rpvar is on by default. To recover the old behavior, set the rpvar to false in the journal file or directly in the active Fluent session. Note that this rpvar is not saved to the case file. **NITA**, **Hybrid Initialization**, and **FMG Initilization** cases are unaffected along with all cases using **PRESTO!** or **Standard** pressure schemes.

- A change has been made to the handling of porous jump boundary conditions in single phase flows. The porous jump face zone is now reoriented based on the bulk flow direction. This removes a solution dependence on the orientation of the porous jump zone. To revert to the previous behavior, use the following Scheme command:

  ```
  (rpsetvar jump/orient-zone-based-on-flow? #f)
  ```

Convergence Optimization

- Several changes in default settings have been made for the AMG linear solver. Existing case file settings will be respected.

  – Maximum number of AMG cycles for coupled equations is changed to 10 from 30.

  – The energy equation now uses F-cycle by default (was Flexible cycle).

- When Pseudo-Transient is enabled, the pseudo-transient method is no longer applied for combustion variables or species by default. Additionally, for the premixed, partially-premixed, and PDF combustion models, pseudo-transient is no longer applied to the energy equation by default. Existing case settings are respected.

- The domain reordering operation has been removed from FMG initialization. This may change the convergence behavior during FMG initialization, but the final results should be the same. You can manually perform the reordering before starting the FMG initialization by using the **Mesh** → **Reorder** → **Domain** menu command. You can also use the TUI command `mesh/reorder/reorder-domain`.

- Porous and/or solidification/melting models that solve the energy equation using pseudo transient method will now use a solid pseudo transient time scale to achieve faster convergence for the energy equation. To revert to the old behavior use the following command:

  ```
  (rpsetvar 'pseudo/energy-solid-dt-in-melt-porous-zone? #f)
  ```

Solver-Meshing

- The diffusion-based mesh smoothing method has been changed to use a finite element discretization rather than a finite volume discretization when the mesh does not include polyhedral or hanging-node

elements. As a result, small differences in the smoothed element shapes can be expected which may lead to small solution changes in mesh-dependent problems.

- When using the VOF model with remeshing volume and mass conservation is now enforced on a per-phase basis rather than only at the mixture level.

- A source term correction is added to the energy equation for moving and deforming meshes (MDM) where pressure work must be included. This source is designed to prevent non-physical build up of temperature that commonly occurs with 1^{st} order in time mdm cases for compressing cylinders. It is active for all 1^{st} order cases where pressure work is needed. It is also active for 2^{nd} order in time mdm with remeshing, when remeshing occurs and the solver switches to 1^{st} order in time. To disable the addition of the source term, enter the following command:

```
(rpsetvar 'mdm-press-work-correct? #f)
```

- The algorithm for conversion to polyhedral cells has been modified in order to reduce the memory consumption for the conversion. As a result of this change, there may be small differences in nodal positions when compared to meshes converted using older versions of Fluent. The legacy conversion algorithm can be invoked using the following Scheme command:

```
(rpsetvar 'polyhedra/lean-conversion? #f)
```

Turbulence

- The effects of compressibility on turbulence are no longer included by default. To enable them you must either enable **Compressibility Correction** in the **Viscous Model** dialog box or use the TUI command `define/models/viscous/turb-compressibility?` (This TUI command was previously called `define/models/viscous/kw-compressibility?`)

- A small lower limit is imposed on U^* when using non-equilibrium wall functions in order to avoid zero wall shear.

- The Wall Modeled LES (WMLES) S-Omega formulation that was a beta feature in Fluent R14.5 has been further optimized and is now released in R15.0. To revert to the previous (beta) formulation, use the scheme command:

```
(rpsetvar 'les-wmles-smnsomg/version-r145? #t)
```

- You can no longer combine the SST model with SAS using the TUI command: `define/models/viscous/turbulence-expert/kw-add-sas?`. In order to combine SAS with SST in R15.0 your only option is to select **Scale-Adaptive Simulation (SAS)** from the **Model** list in the **Viscous Model** dialog box. If you used the TUI command to combine SAS with SST in R14.5, you must switch to the **Scale-Adaptive Simulation (SAS)** model in the **Viscous Model** dialog box in order to continue the simulation in R15.0 (note that the solution may change as a result).

Heat Transfer

- The dual-cell heat exchanger model has been improved to eliminate problems when the target heat rejection was specified as a negative value. Cases in which target heat rejection is a positive value are not affected.

- The S2S model now supports non-conformal fluid-fluid interfaces with the **Coupled Wall** option enabled, where the wall zones created participate in the S2S calculations. Similarly, non-conformal fluid-

solid interfaces are now supported as well. Previous S2S implementations were only compatible with non-conformal fluid-fluid interfaces that did not involve a coupled wall.

- The multi-layer shell mode will be enabled by default, which may affect the results of solutions that involve shell conduction. If you need to revert to the single-layer shell mode of previous versions of Fluent, you can use the `define/models/shell-conduction/multi-layer-shell?` text command.

- The default surface clustering algorithm for cluster-to-cluster (C2C) has changed to the BSP from the Greedy algorithm in R15.0. The Greedy algorithm was expensive for problems with > 15-20M cells, although the overall cluster quality is better. Use the following command to revert to the Greedy algorithm:

```
(rpsetvar 's2s/c2c-clustering-algo 2)
```

Reacting Flow

- For ISAT, Jacobians are now calculated analytically. No solution changes are expected with the exception that cases where the ODE solver was previously not converging may now converge using the analytically-calculated Jacobians. Run-times in general should decrease.

- The stoichiometric mixture fraction required when discretizing flamelets and PDF tables is now calculated by minimizing reactant mole fractions rather than by using peak temperature. Small changes in interpolated values can be expected due to different discretization points.

- A new spark model has been introduced which is much less sensitive to numerical parameters such as mesh size and time-step. By default, Fluent will revert to the R14.5 spark model when reading R14.5 case files.

- A more accurate algorithm is used for the stiff chemistry solver with second order temporal discretization. Some changes in results can be expected compared with previous releases.

- The G-equation premixed model incorporates numerical enhancements for curvature calculation.

- The DPM source term is now calculated correctly for Non-Premixed and Partially-Premixed combustion with DPM, which may cause a difference of a few Kelvin from pre- Release 15.0 versions. To revert to the older formulation:

```
(rpsetvar 'pdf/dpm-h-source-r145? #t)
```

Discrete Phase Model

- The functionality of **Start Crank Angle** and **Stop Crank Angle** used for In Cylinder flows in injection Point Properties, has changed to be compatible with the definition of Dynamic Mesh Events. The injection crank angles are defined for one complete engine cycle and repeated periodically in all engine cycles. The inputs for **Start Crank Angle** and **Stop Crank Angle** in cases that were set up in versions of Fluent prior to R15 need to be reviewed and possibly modified to take into account the new functionality.

- A correction to the calculation of the turbulence source terms in the **Two-Way Turbulence Coupling** feature may result in changes in solution and convergence behavior. Previously, the computed source terms were too small by several orders of magnitude. If desired, the previous behavior can be achieved by disabling the **Two-Way Turbulence Coupling** feature.

- When the **Particle Radiation Interaction** option is enabled the boiling rate of the multicomponent particle is augmented by the radiation term.

- The default density for the material particle-mixture-template has been changed to volume-weighted-mixing-law from a constant value of 1 kg/m^3.

Volume of Fluid Model

- Linear damping has been added to the VOF numerical beach treatment for open channel flow. This may marginally affect the damping behavior in the numerical beach zone, but should not affect the results in the non-damping zone.

Acoustics

- The definitions of Y Axis Functions when using the Fast Fourier Transform tool have been changed. Refer to Graphics, Reporting, and Postprocessing in this section for further details.

- The "Overall sound pressure level," which is printed to the Fluent console after a FFT of the acoustic signals, may deviate from Release 14.5 results. The old value resulted from summing up the Fourier modes. The new value is based on the raw signal variance, without the possible effect of the window function. This change is a result of the newly implemented Welch's method (FFT averaging using the multiple overlapping segments.

Graphics, Reporting, and Postprocessing

- The definitions of the available Y Axis Functions for the Fast Fourier Transform tool have been changed.

  – **Magnitude** has been scaled up by a factor of SQRT(2) for n>0 making it consistent with the conventional definition. The **Sound Amplitude (dB)** function has been changed correspondingly, as it is derived from **Magnitude**.

  – **Power Spectral Density** has changed for cases when the user plots the detailed spectrum (as opposed to the octave or 1/3–octave bands) with **Frequency**, **Strouhal Number**, or **Fourier Mode** selected as the X Axis Function. With the new definition, the **Power Spectral Density** is normalized per frequency unit of 1 Hz. It is thus statistically invariant to the total signal length / spectrum frequency step. As a result, the units have changed in such cases (for example, from Pa^2 to Pa^2/Hz). The **Sound Pressure Level (dB)** definition has likewise changed as it is derived from **Power Spectral Density**.

- The step-by-step particle report (`.his`) for DPM steady particle tracking no longer contains a column labeled **Particle Number in Parcel**. Data in subsequent columns will shift by one column.

User Interface

- Solver interrupts have been improved to maintain better solver state consistency. The new behavior is as follows:

  – Pressing **Ctrl**+**c** during a transient simulation no longer interrupts the solver immediately, but waits until the end of the current time step before stopping the solver. If desired, you can interrupt the solver immediately by pressing **Ctrl**+**c** a second time.

  – For steady state simulations the behavior of **Ctrl**+**c** remains unchanged and it interrupts the solver immediately.

  – The **Cancel** button in the **Working** dialog box behaves similarly, interrupting immediately for steady state and at the end of the time step for transient.

UDF Programming Interface

- The argument `real T` has been added to the following DPM UDF macros. Existing UDF's using these macros will need to be updated or they will fail to compile in Fluent 15.0.

  DEFINE_DPM_VP_EQUILIB

  ```
  #define DEFINE_DPM_VP_EQUILIB(name, p, cvap_surf,Z) \
   void name(Tracked_Particle *p, real *cvap_surf, real *Z)
  ```

  has changed to:

  ```
  #define DEFINE_DPM_VP_EQUILIB(name, p, T, cvap_surf, Z) \
   void name(Tracked_Particle *p, real T, real *cvap_surf, real *Z)
  ```

  DEFINE_DPM_PROPERTY

  ```
  #define DEFINE_DPM_PROPERTY(name, c, t, p) \
   real name(cell_t c, Thread *t, Tracked_Particle *p)
  ```

  has changed to:

  ```
  #define DEFINE_DPM_PROPERTY(name, c, t, p, T) \
   real name(cell_t c, Thread *t, Tracked_Particle *p, real T)
  ```

- The argument `cxboolean vapor_phase` has been added to the User Defined Real Gas Density Macro.

  double ANYNAME_density

  ```
  double ANYNAME_density(double T, double P, double yi[])
  ```

  has changed to:

  ```
  dbout ANYNAME_density(cxboolean vapor_phase, double T, double P, double yi[])
  ```

  The variable `vapor_phase` passes to your UDF the setting of the text-interface command `define/user-defined/real-gas-models/set-phase` or, if applicable, the zone phase set by the text-interface command `define/boundary-conditions/modify-zones/change-zone-phase`.

- In the Fuel Cell and Electrolysis model, the arguments `cell_t c` and `Thread *t` have been added to the Fuel Cell UDF macro to enable you to specify the water vapor saturation pressure as a function of temperature and user-specified values.

  real Get_P_sat

  ```
  real Get_P_sat(real T)
  ```

  has changed to:

  ```
  real Get_P_sat(real T, cell_t c, Thread *t)
  ```

Chapter 2: CFX Release Notes

This section summarizes the new features in ANSYS CFX Release 15.0.

2.1. New Features and Enhancements

New features and enhancements to ANSYS CFX introduced in Release 15.0 are highlighted in this section.

2.1.1. Parallel Processing Enhancements

The following parallel processing improvements have been introduced in ANSYS CFX:

- Platform MPI is the default parallel start method.
- Intel MPI is an available parallel start method.
- For parallel runs, the maximum number of mesh partitions has been increased to 16384.

2.1.2. ANSYS CFX-Solver

New features and enhancements to the CFX-Solver introduced in Release 15.0 are highlighted in this section.

- The `Surface of Revolution` mesh motion boundary condition no longer requires that the surface has a single radial value for each axial position. For example, surfaces may run perpendicular to the axis of revolution.
- Enhancements to the mesh motion defaults and automatic options provide greater ease of use and control in mesh deformation cases.
- Primarily for turbomachinery cases, there is a new outlet boundary condition that enables you to sweep through the complete operational range of mass flow at the outlet, including machine operating points from choked flow to stall conditions. For details, see Exit Corrected Mass Flow Rate in the *CFX-Solver Modeling Guide* and Exit Corrected Mass Flow Rate in the *CFX-Solver Theory Guide*.
- When defining a run using an initial values file that contains mesh displacements, you can more easily specify whether to use the initial or final mesh from the initial values file. For details, see the description for "Initial Values File containing Mesh Displacements" in Continuing the History in the *CFX-Solver Modeling Guide*.
- You can reposition and replicate solution data in order to initialize a simulation. For details, see Initial Values Tab in the *CFX-Pre User's Guide* and Fourier Transformation Method for an Inlet Disturbance Case in the *CFX Tutorials*.
- Changes have been made to the controls that affect the interpolation of particle data from an initial values file. For details, see Adjusting the Bounding Box Tolerance in the *CFX-Solver Modeling Guide*.

- You can more easily specify whether the interpolator matches fluid definitions between the Initial Values File(s) and the Solver Input File. By default, when interpolating from multiple Initial Values files onto a single Solver Input File, if each Initial Values File contains just one fluid, then the interpolator regards all of these fluids as being the same fluid, regardless of the Fluid Definition name.

  You can make the interpolator match each fluid from each Initial Values File with the appropriate fluid in the Solver Input File by selecting the **Enforce Strict Name Mapping for Phases** setting, which is described in Interpolator Tab in the *CFX-Pre User's Guide*.

- If Lagrangian Particle Tracking information is included in the simulation, you can now monitor particle energy and mass flow rates in addition to particle force, momentum, and sources.

- Particle track variables are now available in histograms. In addition, two new particle track variables are available: `<Particle Type>.Velocity Magnitude`, `<Particle Type>.Particle Mass Flow Rate`. For details, see Particle Track Variables in the *CFX Reference Guide*.

- The expert parameter `porous cs discretisation option` has a new option: 3. For details, see Discretization Parameters in the *CFX-Solver Modeling Guide*.

- Expert parameter `realeos liquid prop` is now accessible from the user interface. This parameter controls whether liquid saturation properties are extracted using a function of temperature or a function of both temperature and pressure. For details, see Physical Models Parameters in the *CFX-Solver Modeling Guide* and Real Gas Liquid Properties in the *CFX-Solver Theory Guide*.

- Transient Blade Row model expert parameters are available in CFX-Pre under the **Expert Parameters** details view. For details, see Transient Blade Row Models Parameters in the *CFX-Solver Modeling Guide*.

- Refrigerant material `R-1234yf` is now available for selection as a fluid material in the ANSYS CFX materials database.

- You can now enable Beta features from the options menu. For details, see CFX-Solver Manager Options in the *CFX-Solver Manager User's Guide*.

- Several bug fixes and improvements have been made to the calculation and reporting of equation imbalances. The changes relate to the way that imbalance percentages are calculated, and can therefore affect the number of iterations the solver performs when percentage imbalances are used to determine convergence.

  Specific areas where imbalance percentages may have changed from previous versions include:

  – Multiphase cases without domain interfaces

  – Particle transport cases

  – Cases with point, boundary, or subdomain sources

  Note that in all cases, only the way the that the normalized (percentage) imbalances are calculated has been changed; the way that the equation imbalances are calculated is unchanged. For details, see Global Conservation Statistics in the *CFX-Solver Manager User's Guide*.

- The calculation of solver work units in parallel runs has been modified to be more consistent with the calculation of solver work units in serial runs. The calculation of solver work units in parallel runs is now independent from the number of partitions. There was formerly a problem affecting all parallel runs (especially runs with a large number of partitions), whereby the calculated work units were too large, causing

the linear solver to return prematurely. This former problem resulted in different convergence histories depending on the number of partitions. The convergence histories are now more consistent.

- There is a new expert parameter, `parallel optimization level`, to help optimize parallel performance. For details, see Physical Models Parameters in the *CFX-Solver Modeling Guide*.

2.1.3. ANSYS CFX-Pre

New features and enhancements to CFX-Pre introduced in Release 15.0 are highlighted in this section.

- For blade flutter cases, parametric studies of nodal diameter can be done in Workbench.
- You can select the **Profile Vector** check box to visualize vectors defined from the variables in the profile file. For details, see Viewing and Editing Data Imported from a File in the *CFX-Pre User's Guide*.
- Aerodynamic damping can be monitored. For details, see Aerodynamic Damping: Frame Overview in the *CFX-Pre User's Guide*.
- There is a new turbulence model option for the Detached Eddy Simulation model. For details, see Turbulence: Option in the *CFX-Pre User's Guide*.

2.1.4. ANSYS CFX Documentation

New features and enhancements to the ANSYS CFX documentation introduced in Release 15.0 are highlighted in this section.

- The documentation viewer has been changed.
- PDF files can no longer be opened from the help menus of ANSYS CFX software. PDF help is accessible from the customer portal.
- The Fourier Transformation Method for an Inlet Disturbance Case tutorial has been changed significantly.
- The Time Transformation Method for a Transient Rotor-Stator Case tutorial has been changed significantly.

2.2. Incompatibilities

This sections highlights differences in the behavior between Release 14.5 and Release 15.0 of ANSYS CFX.

2.2.1. CFX-Solver

- File Type Support

  FLOTRAN files with the following extensions are no longer supported: .rfl, .brfl, .brmg, .brst, and .brth.

- Numerics Changes

  The algorithm for the `Surface of Revolution` option for Mesh Motion has changed to allow surfaces to run perpendicular to the axis of revolution. If necessary, you can revert some aspects of the old algorithm by setting the expert parameter `meshdisp sor override` to `f`.

- Discretization changes (that affect the converged solution):

The default value of the expert parameter `porous cs discretisation` has changed from 2 to 3. For details on this expert parameter, see Discretization Parameters in the *CFX-Solver Modeling Guide*.

- MPICH is no longer a (readily available) parallel start method.
- Immersed solid cases that utilize the `General Motion` option (for **Domain Motion**) to rotate both the body and the origin will not orientate correctly after a restart if the initial values file was created with a version earlier than 15.0.

Chapter 3: TurboGrid Release Notes

There are no new features or enhancements in ANSYS TurboGrid in Release 15.0.

Chapter 4: ANSYS BladeModeler Release Notes

This section summarizes the new features in BladeModeler Release 15.0. Topics include:

4.1. BladeGen

BladeGen is a geometry-creation tool for turbomachinery blade rows.

4.1.1. BladeGen New Features and Enhancements

There are no new features or enhancements in BladeGen.

4.2. BladeEditor

ANSYS BladeEditor is a plugin for ANSYS DesignModeler for creating, importing, and editing blade geometry.

4.2.1. BladeEditor New Features and Enhancements

- There is a new property of the Blade and Splitter features, named Operation, that can be used to freeze some features while you are designing the blade or splitter, in order to reduce feature generation time.
- The ThroatArea feature now has options for controlling the throat location and for choosing between speed and accuracy of the throat calculation.
- There is a new feature, SectorCut, that can be used to create a 3D hub sector. For details, see SectorCut Feature in the *TurboSystem User's Guide*.
- There is a new advanced property for the FlowPath feature, named Interpolation Mesh Size, that can be used to increase the interpolation mesh size for cases that require it. For details, see FlowPath Feature in the *TurboSystem User's Guide*.

4.2.2. BladeEditor Incompatibilities

This section describes the operational changes, the procedural changes (actions that have to be done differently in this release to get an outcome available in previous releases), and the support changes (functionality that is no longer supported) in the current release of BladeEditor.

- When you resume work on a database that was saved from a previous version, you will have to regenerate all ExportPoint and VistaTFExport features.

Chapter 5: CFD-Post Release Notes

This chapter summarizes the new features and incompatibilities in CFD-Post Release 15.0. Topics Include:

5.1. New Features and Enhancements

- Boundary Condition profiles

  You can export connectivity data for boundary condition profiles using the **Export Geometry Information** > **Line and Face Connectivity** option. For details, see Line and Face Connectivity Check Box in the *CFD-Post User's Guide*.

- Crank Angle Option for Timestep Selector

  For internal combustion engine simulations, a column appears in the **Timestep Selector**, which displays the `Crank Angle` at each associated time step. You can synchronize the time steps used for multiple internal combustion engine files by setting the **Sync Cases** option to `By Crank Angle`. For further details, see Timestep Selector in the *CFD-Post User's Guide* and Multiple Files in the *CFD-Post User's Guide*. To insert auto-annotation for a crank angle in internal combustion engine cases, see Type in the *CFD-Post User's Guide*.

- Ability to select multiple locations when creating objects

  You can select multiple domains, boundaries, mesh regions, or user locations and insert a new user plot with the selection as the new plot's location. For details, see Outline Tree View Shortcuts in the *CFD-Post User's Guide*.

- You can produce instances of a domain with data available in them for use in post-processing. This feature is for transient blade row cases only. For details, see Data Instancing Tab in the *CFD-Post User's Guide*.

- When post-processing time-dependent data using the Fast Fourier Transform (FFT) algorithm, various functions that are available under **Y Axis** > **Data Selection** (see Y Axis: Data Selection in the *CFD-Post User's Guide*) have changed.

  – The function for power spectral density has changed for cases in which you plot a detailed spectrum (**X Function** set to `Frequency`, `Strouhal Number`, or `Fourier Mode`). Power spectral density is now normalized per hertz ([Hz], a frequency unit), so it no longer varies based on the total signal length or spectrum frequency step. Consequently, its units have changed (for example from Pa^2 to Pa^2/Hz). Note that the function for sound pressure level has been changed accordingly, since it is derived from the function for power spectral density.

  – The function for magnitude is now calculated using a more conventional method. As a result, for $n > 0$, the magnitude is higher than the values calculated by previous versions of the software by a factor of $\sqrt{2}$. Note that the function for sound amplitude has been changed accordingly, since it is derived from the function for magnitude.

Chapter 6: Polyflow Release Notes

The following sections contain release information for ANSYS Polyflow 15.0.

6.1. Introduction

ANSYS Polyflow 15.0 is the fifth version of ANSYS Polyflow to be integrated into ANSYS Workbench. Starting in version 12.1, ANSYS Polyflow users were able to create interlinked systems with geometry, meshing, solution setup, solver and postprocessing inside ANSYS Workbench, using shared licensing and HPC. Blow molding and extrusion application-specific versions of ANSYS Polyflow were introduced to allow specific industrial processes to be simulated. With regard to modeling, two new models were introduced: the volume of fluid (VOF) model for free surface modeling in a fixed domain; and the discrete ordinates (DO) model for radiation.

In ANSYS Polyflow 15.0, the ANSYS Workbench integration, licensing, and modeling capabilities have been further enhanced to meet the needs of ANSYS Polyflow users.

Note

ANSYS Polyflow 15.0 is installed under `ANSYS Inc\v150\polyflow` on Windows and `ansys_inc/v150/polyflow` on Linux platforms.

ANSYS Polyflow 15.0 is available within ANSYS Workbench for Windows and Linux platforms.

6.2. New Features

The new features in ANSYS Polyflow 15.0 are as follows:

- Various graphical user interface enhancements have been implemented, in order to speed up user interactions.
- The subdivision of shell elements in blow molding and thermoforming simulations has been enhanced: to preserve the quality of finite elements, several subdivision patterns are available, and the solver picks up the subdivision that leads to the best subelements.
- It is now possible to use Polydata to convert Fluent format mesh files into Polyflow native meshes, as long as the mesh does not contain non-supported elements (polyhedra). 2D PMeshes in 3D and 1D PMeshes in 2D are supported.
- The slipping condition applied on solid moving parts involved in MST simulations has been improved, in order to ensure that mass is conserved. To achieve this result, the available slipping law in MST calculations obeys an asymptotic behavior characterized by an upper limit of the slipping stress.

- For nonisothermal problems, it is now easier to enable appropriate evolution schemes related to three typical sources of nonlinearities (that is, viscous heating, thermal convection, and the temperature dependence of the viscosity), and thus reduce the risk of convergence difficulties.

- When using ANSYS Polymat to fit material parameters for a multi-mode differential viscoelastic model, it is now possible to specify that the corresponding nonlinear parameters for the material model be assigned the same value among all modes. This may improve the fitting procedure by increasing the sensitivity of the model properties with respect to nonlinear parameters.

- The guidelines for using visoelastic models have been expanded and improved in the Polyflow User's Guide and the Polymat User's Guide.

- It is now possible to make contact parameters dependent upon contact time via the PMAT feature. The contact time is now part of the list of available fields when defining a PMAT function for a parameter affecting contact.

- The import of CutCell meshes has been improved such that there are fewer configurations of non-conformal elements that are not supported.

- The following solver-related improvements have been made:

  – The solver speed has increased for diagonal matrices, allowing the solver to calculate postprocessors (for example, shear rate) faster.

  – The solver memory management is improved.

  – The algorithm for the calculation of the "inside" field as part of the mesh superpostion technique has been improved, leading to a five-fold increase in speed for small screws, and an up to ten-fold increase for long screws, with only a minor increase in the memory footprint. This algorithm also scales well in parallel.

  – The time-step management has been improved when adaptive meshing is used for large deformations: larger time steps are used, leading to significant reductions in computational time. Note that the reason why the reported number of steps can be larger is because a time step can be skipped when mesh deformation is too important.

  – The global CPU time has been reduced through the computation of postprocessors only when required; some postprocessors are only executed when an output is triggered.

  – Improvements have been made relating to IO: the amount of data written in files has been reduced through the elimination of unnecessary or redundant data; the free format is now used, which allows the data to be written with less characters (about 30% less); and asynchronous writes are performed when possible.

  – When adaptive meshing is activated, calculated quantities must be mapped from the previous mesh onto the improved mesh. This mapping has been significantly accelerated.

  – The evaluation of curvatures and angles between elements of the mold for adaptive meshing for contact has been optimized, in order to reduce the number of calculations. Adaptive meshing is thus much faster.

- The default thickness interpolation for shell elements has been changed from linear (continuous) to constant (discontinuous).

- It is now possible to define slip conditions that are not only velocity and temperature dependent, but also pressure dependent. This pressure dependence can be used to model a slipping law that is quite similar to the Coulomb law. Pressure-dependent slip is not available for contact problems.
- The contact detection algorithm has been further improved, making it even more robust.
- An example demonstrating the inverse prediction capability (die design) for fiber spinning has been added.
- The loading of large mesh files in Polydata has been significantly accelerated.

6.3. Defect Fixes

The defect fixes in ANSYS Polyflow 15.0 are as follows:

- The exporting of IGES files has been corrected so that you can approve the system of units.
- A fix was introduced to eliminate limitations on the number of equations allowed by the Polyflow solver.
- A fix was introduced to avoid Polyflow crashes when the viscosity is defined as a function of pressure through the PMAT feature.
- A fix was introduced to avoid Polyflow crashes when running integral viscoelastic calculations in parallel.
- A fix was introduced so that adaptive meshing via TGrid can be applied on more than one sub-domain
- A fix was introduced to avoid Polyflow crashes when the output triggering is incorrectly defined.
- A fix was introduced to avoid Polydata crashes due to the definition of too many sub-tasks.
- It is now possible to import temperature data into ANSYS Mechanical from a steady-state Polyflow simulation.
- A fix was introduced to avoid Polydata crashes when specifying a UDF for the rate constant of a chemical reaction.
- A fix was introduced to avoid Polydata crashes when accessing the element distortion check for an adaptive meshing problem.
- With regard to boundary conditions for a moving interface that is by definition shared by two sub-tasks, it is no longer possible to attempt to modify them except in the sub-task in which they were originally defined. Previously, the relevant menu items were accessible in the other sub-task as well, even though the changes made using such menu items were not recorded by Polydata; these menu items are no longer selectable.
- It is now possible to define the conservation of volume of liquid on each sub-task, as it is no longer imposed on the whole velocity field.
- The limit for the definition of a multi-ramp function has been increased from 100 to 250 pairs of data.

- A fix was introduced to avoid Polydata crashes related to the definition of adaptive meshing for large variations of shear rate.
- The delaying strategy in the solver has been corrected, so that variables are eliminated in the best order. This removes the risk of null pivot and improves the conditioning of the equations system.
- Polyflow now correctly interpolates all fields and correctly evaluates moving parts shape when using adaptive meshing for Mesh Superposition Technique (MST) in 2D. In previous releases of Polyflow, interpolation of the initial coordinates for nodes on the middle of segments of moving parts led to erosion of the moving parts.
- It is now possible to use z-coordinate and z-velocity in PMAT functions in the frame of shell simulations.
- The generation of a sliceable free jet for a non-conformal (CutCell) mesh is now more robust.
- The force postprocessor for differential viscoelastic flows is now more accurate.
- A fix has been introduced to avoid crashes of the parallel solver when using user-defined functions.
- The secant solver is now more robust when handling the transport equation in deforming meshes.

6.4. New Limitations in ANSYS Polyflow 15.0

The following is a list of new or recently discovered limitations known for ANSYS Polyflow 15.0.

- CutCell meshes as well as meshes that have been refined by adaptive meshing cannot be processed by ANSYS Polyfuse.
- In some cases, flow problems with non-moving parts that involve the slip condition may not converge, or may converge but produce nonphysical results. In such cases, it is recommended that you try using the **constant per element** interpolation rule when defining the boundary condition. This rule can be selected by using the **Modify interpolation rule** menu option in the **Slipping : Advanced options** menu (accessed through the **Advanced options** menu option when **Slip conditions** is selected for the flow boundary) or the **Flow boundary conditions** menu (when **Interface with elastic solid** is selected for the flow boundary as part of the FSI model).
- Errors may be generated when you use remote display on a Linux machine if you have not defined the `LIBGL_ALWAYS_INDIRECT` environment variable.
- Meshes created for Polyflow using ANSYS DesignModeler 15.0 and/or ANSYS Meshing 15.0 may not be scaled according to the length unit used within those preprocessors; therefore, you should always check the length unit and dimensions of the mesh using the **Info** button in the **Mesh** tab of Polydata (see Reporting Information about the Mesh in the Polyflow User's Guide), and then ensure that the values you enter as part of your Polydata setup consistently account for this mesh scale.
- For further limitations that are present in ANSYS Polyflow 15.0 but that were discovered during previous releases, see Known Limitations in ANSYS Polyflow 15.0 in the Polyflow User's Guide.

Part III: ANSYS Electronics Products

Release notes are available for the following ANSYS Electronics products:

Chapter 1: Icepak Release Notes

Release 15.0 of the ANSYS Icepak application offers most of the capabilities from previous releases plus many new features and enhancements.

1.1. Introduction

ANSYS Icepak 15.0 is a release of ANSYS Icepak that has new features and defect fixes. New features are listed in the following section of this document.

1.2. New and Modified Features in ANSYS Icepak 15.0

- Graphical User Interface
  - Implemented automatic problem setup wizard for common solution scenarios. See Section Problem Setup Wizard of the User's Guide.
  - Implemented ability to view orthotropic metal fraction maps for traces and vias before executing the solver. See Section The View Menu of the User's Guide.
  - Implemented option to highlight objects by material, property, power value and object type. See Section The Model Menu of the User's Guide.
  - Implemented **New** and **Close** buttons for monitor points. See Section Creating a New Point of the User's Guide.
  - Implemented the **Delete assembly** option to move items within an assembly to the level of the assembly node and moves the assembly to the **Trash** node. See Section The Assembly Node Context Menu of the User's Guide.
  - Implemented the **Default transparency** option to globally control the transparency of objects in your ANSYS Icepak model. See Section Using the Context Menus in the Graphics Display Window of the User's Guide.
  - Implemented material property search to automatically search for materials in the library. See Section The Materials Node Context Menu of the User's Guide.
- ECAD Import/Export
  - Implemented feature to support the import of trace and via information and compute a thermal conductivity map for PCB objects. See Section Adding a PCB to Your ANSYS Icepak Model of the User's Guide.

  - Implemented feature to view trace layers by net information. See Section Importing Trace Files of the User's Guide.

- Model Import/Export

  - Implemented option to publish ANSYS Icepak parameters to the Workbench Parameter Manager. Allows design points, design of experiments and optimization with ANSYS DesignXplorer. See Design Explorer - Icepak Coupling in Workbench of the User's Guide.

  - Implemented one-way coupling between Q3D Extractor and ANSYS Icepak application within ANSYS Workbench. Both volumetric and surface mapping are possible. See Ansoft - Icepak Coupling in Workbench of the User's Guide.

  - Implemented export of distributed Icepak HTC values for package top and substrate. See Section Write Sentinel TI HTC File of the User's Guide.

  - Model summary report now includes material properties for each object.

- Modeling and meshing

  - DesignModeler supports creation of Icepak enclosure object. See Section Icepak Enclosure of the DesignModeler User's Guide.

  - Implemented species transport modeling to enable modeling of multiple species (up to twelve) in the simulations. See Section Species Transport Modeling of the User's Guide.

  - Implemented automatic adaptive time-stepping for transient solutions. See Section User Inputs for Transient Simulations of the User's Guide.

  - Implemented additional specification for solar loading. Solar loads can now be defined by total heat flux and direction vector. See Section User Inputs for the Solar Load Model of the User's Guide.

  - Implemented ability to specify transient currents for variable Joule heating. See Section Adding a Source to Your ANSYS Icepak Model of the User's Guide.

  - Implemented fixed flow rate for fans with 2D CAD shaped geometry. See Section Fixed Flow of the User's Guide.

  - Implemented feature to allow temperatures at boundary nodes of network objects to be transient. See Section Adding a Network to Your ANSYS Icepak Model of the User's Guide.

  - Implemented the kw-sst turbulence model. See Section k-ω SST Model of the User's Guide.

  - Implemented feature to identify gaps between objects that overlap. See Section Meshing Options of the User's Guide.

- Solving

  - Implemented a new coupled pressure velocity solver scheme for robust and efficient solution of steady-state flows. See Section Using the Solve Panel to Set the Solver Controls of the User's Guide.

  - Enabled GPU computing for coupled solver and ray tracing radiation model. See Section Using Graphics Processing Units (GPUs) with Icepak of the User's Guide.

- Postprocessing and reporting

- Implemented transparency for postprocessing objects. See Section Displaying Results on Object Faces of the User's Guide.
- Implemented more user interaction with 3D transient plots. See Section Variation Plots of the User's Guide.
- Implemented variation plot in a selected region. See Section Variation Plots of the User's Guide.
- New summary report options to report conductive, convective and/or radiative heat flow for objects. See Summary Reports of the User's Guide.
- Implemented reporting of facet values in the summary reports panel. See Summary Reports of the User's Guide.
- Implemented feature to allow 3D plotting of variation plots. See Section Variation Plots of the User's Guide.

Part IV: ANSYS Geometry & Mesh Prep Products

Release notes are available for the following ANSYS Geometry & Mesh Prep products:

Chapter 1: Geometry Release Notes

This section summarizes the new features in DesignModeler and CAD Integration Release 15.0. Topics include:

DesignModeler Enhancements

1.1. Electronics

Electronics Enhancements

If you are using Icepak, you may now define a PCB object from a block or polygonal block geometry. Additionally, you are now able to create an Enclosure object using the Electronic Enclosure feature. The enclosure can take as input an enclosure shaped body and generate an Icepak body that defines the Icepak Enclosure object.

For more information, see Electronics> Electronics Menu in the DesignModeler User's Guide.

1.2. Import/Export

IGES/STEP Property Changes

You may now specify Heal and Clean options for IGES imports, and the Stitch Surfaces option for STEP imports. IGES and STEP import options now are consistent with each other.

For more information, see Geometry Options in Import and Attach Options (Menus> File Menu> Import External Geometry File) in the DesignModeler User's Guide.

Attach Feature Source Property Changes

The Attach feature now allows the Source property to be reconnected with an active CAD system, mostly used when resuming work from a previous Workbench session. The new option, "Browse.." launches a file selection dialog box to pick a CAD file.

For more information, see Source Property (Menus> File Menu> Attach to Active CAD Geometry> Attach Properties) in the DesignModeler User's Guide.

Method to Export Cross Section Data to a Script

Though batch scripting commands exist for cross section types, there is now an option to export cross section data to a script. The "Export Cross Sections to Script.." is found in the context menu of the cross section group in the Tree Outline.

For more information, see Export Cross Sections to Script (Menus> Context Menus) in the DesignModeler User's Guide.

1.3. Modeling Feature Enhancements

Body Operation Clean Bodies Option

A new option has been added to the Body Operation feature which allows you to Clean Bodies. Two levels of cleaning are available, normal and high. This cleaning option is also available in the Import and Attach feature options.

For more information, see Body Operation (3D Modeling> Advanced Features and Tools) in the DesignModeler User's Guide.

Refactored Body Operation

Commonly used Body Operation types such as transformations and deletion are now available directly from the menu. Additionally, when performing a Rotate operation using components, you are able to specify the point through which the axis vector passes.

For more information, see 3D Modeling in the DesignModeler User's Guide.

Primitive Feature Target Body Selection

The Primitive features now allow you to select target bodies when the Operation is Cut Material/Imprint Faces/Slice Material.

For more information, see 3D Modeling> Primitives in the DesignModeler User's Guide.

Imprint Feature in Body Operation

Imprinting among bodies has been simplified in that the selected bodies are able to imprint with each other, regardless of whether they are active or frozen. To specify targets and tools, the Boolean feature is still available.

For more information, see Imprint Faces (3D Modeling> Advanced Features and Tools> Body Operation) in the DesignModeler User's Guide.

Non-Uniform Body Scaling

The Scale Type option can be changed to Non-Uniform, which allows for an independent scale factor for each axis. Scale Type can be changed to Uniform if necessary, and a single global scaling factor is then presented.

For more information, see Scale (3D Modeling> Advanced Features and Tools> Body Transformation) in the DesignModeler User's Guide.

Preserve Bodies Feature in Modeling

You are now able to separate face sets and preserve the original body when using the Thin/Surface feature.

For more information, see Thin/Surface (3D Modeling> 3D Features) in the DesignModeler User's Guide.

Plane Definition Added

You can define Plane features with circular or elliptical edges.

For more information, see New Plane (Planes and Sketches> Active Plane/Sketch Toolbar) in the DesignModeler User's Guide.

Component Display in Distance Finder

You can easily compute the XYZ components of the nearest distance computed in addition to the nearest distance when using the Distance Finder utility. XYZ components can also be computed from a particular plane.

For more information, see Distance Finder (3D Modeling> Analysis Tools) in the DesignModeler User's Guide.

1.4. Option Changes

Options Dialog Reorganization

The Options Dialog for DesignModeler has been reorganized for easier use. Additionally, sketching auto constraint settings may now be set in the Options Dialog, which allows the auto constraint settings to persist between sessions.

For more information, see Application Options> Sketching in the DesignModeler User's Guide.

Auto-Save Frequency Changes

Auto-save frequency preference now includes two additional options: 10th (default) and 20th generate.

For more information, see Application Options> Miscellaneous in the DesignModeler User's Guide.

1.5. Parameter Manager

Additional Parameter Manager Functions

The following new functions have been added to the Parameter Manager. The functions and operations they perform, respectively:

- CEILING: rounds up to the nearest integer
- FLOOR: rounds down to the nearest integer
- MIN: minimum value of a set
- MAX: maximum value of a set
- ROUND: rounds to the nearest integer

For more information, see Parameters> Parametric Functions in the DesignModeler User's Guide.

New Parameter Manager User Interface

The Parameter Manager has be redesigned in the latest release so that parameters can be assigned units. Other features include the ability for you to suppress/unsuppress individual design parameters and expressions, and state icons that indicate validity of invalidity of each line in the table.

For more information, see Parameters> Parameters Windows in the DesignModeler User's Guide.

1.6. Shared Topology

Share Topology Performance

Performance of the Shared Topology feature has been significantly improved at Release 15.0. All models created in version 15.0 onwards will use the new, faster algorithm. For older databases, you may control whether to apply the new algorithm using the Optimizations property.

For more information, see Share Topology (3D Modeling> 3D Features) in the DesignModeler User's Guide.

Face Coloring and Display Legends for Shared Topology Type

You now have an additional option in the face coloring dropdown list to color entities by their Share Topology type. The predefined list includes automatic, imprints, edge joints, and none.

For more information, see Face Coloring (Viewing> Model Appearance Controls> Graphics Options) in the DesignModeler User's Guide.

Custom Shared Topology for Bodies

You can now apply a Shared Topology Method to bodies within a multi-body part. This allows you to specify sharing settings for individual bodies that are different from the rest of the part. The property will be set to Default, which means it will inherit its Shared Topology setting from the owning part.

Instancing Support for Multi-Body Parts

Share Topology now supports instancing for multi-body parts using Automatic or Imprint shared topology methods. For parts to be instances of each other, corresponding bodies must have the same shared topology method.

For more information, see Feature Modeling's Effect on Instance Data (3D Modeling> Bodies and Parts> Instancing Support> ANSYS DesignModeler Instancing) in the DesignModeler User's Guide.

Slice Feature Enhancement

The Slice feature now allows the Slice Off Faces option after Share Topology. Faces that are shared between bodies cannot be sliced.

For more information, see Slice Limitations (3D Modeling> Advanced Features and Tools) in the DesignModeler User's Guide.

Face Connect after Shared Topology

You are now able to connect faces that were not shared as a part of the Share Topology automatic operation in order to achieve conformance at desired locations.

1.7. Units

Unit Property for Coordinates Files

You can specify the units of the incoming data when using the Coordinates File option in the Point and 3D Curve features.

For more information, see Coordinates File (3D Modeling> Concept Menu> 3D Curve or 3D Modeling> 3D Features) in the DesignModeler User's Guide.

Support for Tolerance Handling

Model Tolerance is now able to be set in addition to specifying the session unit. This defines the precision to be used for the model. The feature list will be marked to regenerate completely when the tolerance is changed.

For more information, see **Large Model Support and Model Tolerance** in Menus> Units Menu in the DesignModeler User's Guide.

Unit Selection Feature

Both Units and model size settings can be modified from either the Units Menu or the Details View of the model. Large model support or micrometers must be selected, prior to adding features. Any other unit selections may be chosen at any time during the session.

For more information, see Units (Scripting API> Script Features> Functions within Script Features) in the DesignModeler User's Guide.

Ability to Change Units

Session units are now able to be changed without preference control and within the same session.

For more information, see Menus>Units Menu in the DesignModeler User's Guide.

Ability to Set Units from Script

Using Script, the ability to set unit, model size, and tolerance setting is now available. No pop-up windows appear when the options above are changed using script. The following options are now available via script:

- Set/query the session length unit
- Set/query the session angular unit
- Set/query the Large Model Support (if applicable)
- Set/query the model tolerance (if applicable)

For more information, see Units (Scripting API> Script Features> Functions within Script Features) in the DesignModeler User's Guide.

1.8. Usability

Color by Name Selection Legend Scroll Support

You can now access all named selections in the legend using a new scroll bar. This is useful when the number of named selections exceeds the number that can be displayed in the legend at one time.

For more information, see Face Coloring (Viewing> Model Appearance Controls> Graphics Options) in the DesignModeler User's Guide.

Named Selection Legend Inclusion Property

You can control if the Named Selection is included in the Color-by-Named Selection graphics using the Include In Legend property. This allows you to be selective about which Name Selections are shown without needing to suppress/unsuppress them.

For more information, see Named Selection (3D Modeling> Advanced Features and Tools) in the DesignModeler User's Guide.

Addition of Section Planes

The Section Plane feature provides functionality to visually slice the model so that the internal geometry is able to be seen.

For more information, see Section Planes (Viewing> Rotation Modes Toolbar) in the DesignModeler User's Guide.

Hotkey Additions

The new hotkeys are provided below and the options they perform, respectively:

- Ctrl+ F8: Hide All Other Faces
- Ctrl+ F9: Hide All Other Bodies
- F8: Hide/Show Faces
- F9: Suppress/Hide Part and Body
- Shift+ F8: Show All Faces
- Shift+ F9: Show All Bodies

For more information, see Typical Usage> DesignModeler Hotkeys in the DesignModeler User's Guide.

1.9. CAD Integration Enhancements

The following general enhancements have been made in Release 15.0:

Parameter Key Processing Support

The parameter key will now support multiple prefixes/suffixes values entered in the parameter key preference. If you wanted to import parameters which are keyed with DS and others that are keyed with PARAM they could enter DS;PARAM in the Parameter Key field.

For more information, see Parameter Key Property (Menus> File Menu> Attach to Active Geometry> Attach Properties) in the DesignModeler User's Guide.

SolidWorks Parameter Processing Equations

In addition to dimension names, global variables are also processed for parameters. The same filter logic is used to determine which parameters are imported. Parameters that are driven are not imported because they will create conflicting results.

For more information, see SolidWorks Associative Geometry Interface (File Format Support> SolidWorks (*.sldprt, *.sldasm)) in the CAD Integration section of the ANSYS Help.

Geometry Interface Setup for Network Installations

There is no longer a requirement to run the CAD Configuration Manager when setting up a client system that uses the Product Configuration Tool.

For more information, see the CAD Integration section of the ANSYS Help.

Compare Parts on Update Tolerance

There is now added flexibility in the comparison tolerance value. As the tolerance is there to handle small numerical deviations between representations of the import, these numbers are small to avoid marking parts with real change as unmodified.

For more information, see Overview> Compare Parts on Update in the CAD Integration section of the ANSYS Help.

Geometry Interfaces Update for New CAD Releases

Geometry interfaces are updated to support new CAD releases including:

- ACIS 24
- AutoCAD 2014
- Autodesk Inventor 2014
- CATIA V5–6R2013
- CATIA V6 R2013
- JT 9.5
- NX 8.5
- Parasolid 26.0
- Solid Edge ST5
- Solid Edge ST6
- SolidWorks 2013
- Teamcenter Unified 8.1 with NX 7.5, N X8 & NX 8.5
- Teamcenter Unified 8.3 with NX 7.5.2, NX 8 & NX 8.5
- Teamcenter Unified 9.1 with NX 7.5, NX 8 & NX 8.5
- Teamcenter 4–tier support with NX 8.3 & NX 9.1

For more information, see CAD Integration.

Chapter 2: Meshing Application Release Notes

This release of the Meshing application contains many new features and enhancements. Areas where you will find changes and new capabilities include the following:

2.1. Incompatibilities and Changes in Product Behavior from Previous Releases

- There have been significant changes to the size function (especially **Proximity Size Function**) to improve speed and memory usage. The mesh is now more accurate in capturing the number of cells in gap and in accounting for the mesh sizings defined. While this should generally lead to a better quality mesh in a faster amount of time, we have seen some cases where the mesh count changes noticeably from 14.5 to 15.0. These improvements make the Advanced Size function **Proximity Accuracy** unnecessary, so it has been removed.
- When using match controls or Cyclic Symmetry, if the low and high sides of a geometry do not match, the high side geometry is meshed and the low side will be meshed using the transformation from the high side. In this case, the low side geometry and mesh might be slightly different, so a warning is displayed prompting you to check this deviation and decide whether it is acceptable or if you should modify the geometry.

2.2. Ease of Use Enhancements

The following ease of use enhancements have been made in Release 15.0:

- Certain selection and view Shortcut Keys are now available for Mechanical and Meshing tasks.
- You can now create Named Selections based on element and node selections on a mesh.
- The Report Preview tab (from Mechanical) is now enabled for Meshing and enables you to create a report based on analyses in the Tree Outline.

2.3. Meshing Options Enhancements

The following Options enhancements have been made in Release 15.0:

- You can now specify the default number of processors to be used for parallel part meshing using the **Number of CPUs for Parallel Part Meshing** option under **Tools>Options**. You can change this value for a specific project in the **Advanced** group under the **Details** view. Using the default for specifying multiple processors will enhance meshing performance on geometries with multiple parts. For parallel part meshing, the default is set to **Program Controlled** or **0**. This instructs the mesher to use all available CPU cores. The Default setting inherently limits 2 GB memory per CPU core. An explicit value can be specified between **0** and **256**, where **0** is the default. Parallel part meshing is currently only available on 64–bit Windows. Refer to *Parallel Part Meshing Best Practices* for more details.

2.4. Fluent Export Changes and Enhancements

The following changes and enhancements have been made to Fluent Export at Release 15.0:

- You can now control zone type assignments when exporting to ANSYS Fluent. When the ANSYS Fluent export option **Auto Zone Type Assignment** is set to **Off**, all boundary zones are set to WALL as a default. You can then assign zones manually in Fluent.

2.5. Assembly Meshing Changes and Enhancements

The following assembly meshing changes and enhancements have been made at Release 15.0:

- Assembly meshing can now be exported to the CGNS format.

2.6. Assembly Model Changes and Enhancements

The following model assembly enhancements have been made in Release 15.0:

- You can now merge multiple meshing component systems together as mesh-to-mesh connections in ANSYS Workbench. This allows for more flexibility in combining models, or re-using existing mesh models.

2.7. MultiZone Mesh Method Enhancements

The following MultiZone mesh method enhancements have been made at Release 15.0:

- The Multizone Mesh method now supports Body of Influence (BOI) and Sphere of Influence (SOI) controls with the following limitations:
  - The hard-sizing behavior of Sphere of influence (SOI) controls is not supported. The SOI element size is only applicable in the mesh if it is smaller than the local mesh size without SOI.
  - BOI controls are interpreted as an agglomerate of SOI controls that fill the body of influence. Therefore, on the boundaries of that body, the size might be a little larger because the location is not in one of the spheres.
  - The scope is not used for the SOI/BOI definition. The SOI/BOI does not act only on the scoped topology, but on all topologies within the region of the SOI/BOI.
- MultiZone now respects all **Bias Type** options.
- MulitZone now supports mapped face mesh control with internal loops.
- You can now use the **Internal Number of Divisions** option in Mapped Mesh control to specify the number of divisions across annular regions and seamless cylinders.

- The **Preserve Boundaries** option can be set to respect only protected topologies or all features in the model. If set to **All**, the MultiZone mesh method behaves in a patch conforming way, and topology checks are skipped independent of whether this option is **Yes** or **No**. Selecting **All** disables **Meshed Based Defeaturing**.
- You can now choose the **Use Multizone for Sweepable Bodies** option (in **Tools>Options**) to use Multizone in place of Sweep for sweepable bodies.

See MultiZone Meshing for more information.

2.8. Parallel Part Meshing

You can now control three mechanisms in ANSYS Workbench that operate in a parallelized manner:

- Remote Solve Manager Design Point updates. See Options Dialog Box.
- Parallel Part meshing: **Tools**> **Options**> **Number of CPUs for Parallel Part Meshing**
- Individual mesh methods (**MultiZone Quad/Tri**, **Patch Independent Tetra**, and **MultiZone** only): **Tools**> **Options**> **Number of CPUs for Meshing Methods**

Note

Parallel Part meshing is available only on Windows 64–bit systems.

Parallel Part Meshing does not support the following mesh controls:

- Assembly meshing
- Contact Sizing
- Fracture
- Manual Refinement
- Mesh Match via Symmetry
- Morph Service/Morphing
- Pinch
- Post Connection
- Post-Inflation
- Preview Surface Mesh/Preview Inflation
- Retry
- Refinement

For more information, see Parallel Part Meshing Best Practices

2.9. Size Control Enhancements

The following size control enhancements have been made at Release 15.0:

- The new **Reverse Bias** option enhances the bias options for edge meshing by enabling users to select edges to reverse when some of the edges in the original selection do not have the same orientation.
- Local Min Size is now supported for bodies, faces, and edges.
- The **Radial Number of Divisions** control on the Mapped Mesh control has been renamed **Internal Number of Divisions**, and now works for both annular as well as seamless cylinders. See Notes on the Mapped Face Meshing Control for more information.
- The **Curvature and Proximity Refinement** option for the Patch Independent meshing has been renamed **Refinement**, with options for **Proximity and Curvature**, **Curvature**, **Proximity**, or **None**. The default setting is **Proximity and Curvature**.
- The Advanced Size Function setting **Proximity Accuracy** has been removed due to improvements in the size function that more accurately account for the mesh sizings defined, making this setting unnecessary.

2.10. Meshing Failure Troubleshooting

The following enhancements to mesh failure troubleshooting have been made in Release 15.0:

- The new **Topology Checking** option determines whether the mesher performs topology checks for **Patch Independent** and **MultiZone** meshes. If set to **No**, the mesher skips topology checks except where it is necessary to imprint all protected topologies. The default is **Yes**, in which case the mesher performs all topology checks.
- The patch conforming tet, sweep, and hex dominant mesh methods now show the meshing results when the meshing partially fails. To see the entities upon which meshing failed, right-click the **Show Problematic Geometry** message in the **Messages** window.
- Mesh is now colored to indicate meshing failure, or mesh that is out of date. See Recommended First Course of Action for Meshing Failures for more information.

Chapter 3: IC Engine Release Notes

The IC Engine Analysis System is a customized tool that automates many of the required steps for setting up and simulating the flow inside internal combustion engines with moving geometry. It is used to examine the flow rate, swirl and tumble, and other flow parameters during the engine cycle. The IC Engine system uses ANSYS DesignModeler and ANSYS Meshing for geometry decomposition and meshing steps of the simulation. The ANSYS Fluent solver for fluid flow analysis and ANSYS CFD-Post is used for postprocessing. In this release IC Engine System supports combustion simulation along with the cold flow and port flow simulation. The new features in this release are:

IC Engine System Properties

- Option to choose the type of simulation: cold flow, port flow and combustion. Combustion Simulation includes three types: Full Engine Full Cycle, Full Engine Part Cycle (IVC to EVO), and Sector Combustion.
- Choice for Input option for IVC to EVO — By Lift Profile or Direct Values of IVC and EVO

Geometry

- Support for improving geometry before decomposition using Pre Manager.
- Support for geometry having a crevice.
- Compression Ratio Validation.
- Support of geometry preparation for combustion analysis.
  - Preparation of sector geometry from a complete engine geometry.
  - Support for importing sector geometry.
  - Preparation of geometry for closed cycle simulation.

Meshing

- Support of automatic meshing for combustion sector geometry.
- Improvements in meshing to handle closed cycle simulation of the engine geometry.

Solver

- New user interface for solver setting.
- KeyGrid setup with workflow and UI.
- Combustion setup.
  - Diesel combustion simulation for sector geometry.

  – Diesel combustion simulation for full engine full cycle, and closed cycle .

  – SI combustion simulation for full engine full cycle, and closed cycle.

Usability Improvements

- New user interface for solver and keygrid setting.
- Support for improving geometry before decomposition.
- Complete streamlined combustion setup user interface.
- Crank Angle (CA) based file and image saving.
- Automatic time step calculator for the given crank angle period.
- Automatic compression ratio calculation based on the geometry.

Chapter 4: ICEM CFD Release Notes

This section summarizes the new features in ICEM CFD Release 15.0. Topics include:

4.1. Highlights of ANSYS ICEM CFD 15.0

Release 15.0 development efforts included enhancement of ANSYS ICEM CFD as a standalone application as well as continued development of its underlying technology exposed within the ANSYS Workbench-based Meshing application. Specific enhancements are outlined in the following sections.

4.2. Key New Features/Improvements

ANSYS ICEM CFD 15.0 includes the following new features and improvements:

4.2.1. Workbench Add-In Component

- You can now use **File>Import** to import a standalone ICEM CFD project into Workbench.
- You can now use the **Model** cell of Mechanical systems as upstream data.
- The new **Duplicate** on the ICEM CFD system in the schematic copies the entire geometry and mesh data, enabling you to edit a cell in the duplicate system to investigate an alternative modeling approach.
- The ICEM CFD Add-in component now supports the **Create Subset(s) from Named Selection**. If this option is set, then overlapping named selections appear as subsets instead of parts. This is most useful in case of overlapping named selections which are transferred to overlapping subsets instead of to exclusive parts. Subsets can later be converted to parts after sorting out the overlapping entities (point/curve/surface).
- The ICEM CFD Add-in component now supports the geometry cell option, **Use Associativity**. If **Use Associativity** is **On** (the default setting), Part Reference IDs are stored in a Workbench database and remain persistent even if the topology of the geometry is changed.
- You can now set part-specific input parameters, including prism Boolean, using **Part Mesh Setup**.
- Prisms are now automatically created when the project is updated if the `Part PART_NAME: Prism Workbench` Input Parameter is set in the **Part Mesh Setup**.

- You can set additional Workbench Output parameters to include the ability to check the mesh for errors and possible problems during an update, check Hexa PreMesh Quality and number of blocks, and show the results as output parameters in ICEM CFD and Workbench.

- The new **Workbench Replay Control** dialog that is called from the **One-click Replay Recording** menu automatically loads the current Replay script and supports Workbench Input parameters, enabling you to step through Replay scripts that use these parameters. The **Replay Control** dialog called by **File>Replay Scripts>Replay Control** does not support Workbench Input Parameters.

4.2.2. Translate and Rotate

Simultaneous **Translate and Rotate** functionality has been added to the **Transform Block** and **Transform Mesh** Data Entry Zones. This new functionality mimics the **Translate and Rotate** behavior that has long been available in **Transform Geometry**. The new location and orientation is determined by selecting two sets of points, by selecting two curves, or by setting a Local Coordinate System

4.2.3. Smooth Hexahedral Mesh Orthogonal

A new GUI with an additional option for **Edit Mesh > Smooth Hexahedral Mesh Orthogonal** has been released. In the new GUI, the selection of **Smooth Type** is now done by drop-down list, and includes a **Structured** option. The Structured smoother offers two additional choices: Sorenson methods attempt to maintain node distributions (bunching) near the surface boundaries while improving orthogonality. Hilgenstock methods maintain orthogonality and first layer height.

4.2.4. Aspect Ratio (Fluent)

The **Quality Type** drop-down menu has been changed to include a new option: **Aspect Ratio (Fluent)**. You can now determine mesh quality using the same Aspect Ratio algorithm as is used in ANSYS Fluent, as well as the traditional ICEM CFD method.

4.2.5. Delaunay Smooth Mesh option

An option to smooth the mesh after finishing subdivision has been added to the **Quick (Delaunay)** volume meshing process.

4.2.6. Geometry Import options

The Geometry Import options have been streamlined for better integration with Workbench interfaces, yet maintaining some advantages of the long time ICEM CFD import options.

Additional improvements have been made to improve entity recognition when importing from UG files and accelerated Workbench Readers for a wide range of geometries.

4.2.7. Remote Solver Manager

ICEM CFD is now able to be used with the Remote Solver Manager, Reserved Licensing, and HPC Parametric Packs.

4.3. Documentation

All documentation for **ANSYS ICEM CFD Release 15.0** including several tutorials is accessible using the Help menu. Please visit the ANSYS ICEM CFD website for more information.

4.3.1. Tutorials

To access tutorials and their input files on the ANSYS Customer Portal, go to http://support.ansys.com/training. The Customer Portal also contains links for training, for hard copies of the Tutorial manual, or for PDF format copies of the tutorials.

Chapter 5: Fluent Meshing Release Notes

The following sections contain release information for ANSYS Fluent Meshing Release 15.0:

5.1. Changes in Product Behavior from Previous Releases

- The printing of lists of boundary and cell zones in the transcript window during reading and writing of mesh files has been disabled by default to improve file read and write performance. You can increase the verbosity level to reinstate printing of these lists during reading and writing of files.

5.2. New Features

New features available in the meshing mode in Fluent include enhancements to the object based CAD-to-surface meshing workflow such as improved CAD import, size functions, graphics tools and construction tools, display options, wrapper technology improvements, remeshing and join operations. New diagnostics, hole fixing, and topology creation tools have been introduced. For volume meshing, besides improvements to prism, tet, and hexcore meshing, a new thin volume meshing approach has been introduced. There are also enhancements to many existing features, and improved robustness through defect fixes.

User Experience

The following enhancements have been made:

- A new color palette has been added to improve the visual experience. You can assign random colors to objects/zones using a hot-key.
- A new button for multiple-selection lists listing face zones, edge zones, or objects allows you to toggle the tree view for items listed. This allows you to expand/collapse lists based on items with a common prefix.

  Since the zone names are initialized based on CAD hierarchy at the time of CAD import, you can see the CAD hierarchy to start with. Combined with easy renaming/prefixing of objects/zones, this provides a powerful tool to create the analysis model hierarchy.
- The layout of the **Mesh Generation** task page has been modified.
  - The object list is available on the **Mesh Generation** task page.
  - An option is available to overlay graphics on the objects and zones currently displayed in the graphics window.
  - Clipping plane values can be used to set the bounded display ranges. You can also reverse the direction of the movement of the clipping plane relative to the slider movement using the **Flip Clipping Planes** options.
  - You can specify a separator symbol for the common prefix when using the tree view for lists.

- A number of operations such as showing or hiding objects/zones, showing objects/zones in the neighborhood, displaying objects/zones of similar size based on selections in the graphics window can be performed using hot-keys.
- Operations like merging objects, separating object face zones, creating capping surfaces, aligning objects based on selections in the graphics window can be performed using hot-keys.
- There are toggle options for transparent and exploded view, as well as mesh display and selective edge selection.
- Objects/zones can be renamed by selecting them in the graphics window and using a hot-key.
- Fully connected, manifold/non-manifold, faceted boundary zones or objects can be globally remeshed based on the existing size field using a hot-key.
- Graphically selected faces can be locally remeshed based on the existing size field or surrounding sizes using a hot-key.
- The list of available graphical shortcuts (hot-keys) is now available on-screen.
- You can select, deselect, or add items to selection lists based on selections in the graphics window.
- You can select list items based on graphics selections in dialog boxes with dual selection lists.

Object Based Meshing Workflow

The following enhancements have been made:

- Object transformation operations—scaling, translation, and rotation are available.
- Objects are automatically updated after deleting or merging zones, extracting edges from faces comprising the object, and separating faces comprising the object.
- Material point management has been improved.
- A distance threshold can be used for removing gaps between the wrap objects.
- Wrap objects created using the **Wrap Selected Object(s)** option have the same name as the corresponding geometry objects.
- The **Purge Wrap** option allows you to delete the face zones associated with the wrap object(s) selected and reverts the object(s) to their geometry representations.
- Additional wrapping options are available for the shrink-wrap method:
  - The **Low** option allows you to create a rough wrapped representation of the geometry object.
  - The **Medium** option performs additional refinement, imprinting and further aggressive imprinting iterations to improve the feature recovery. Individual zones are recovered based on the original geometry object and then rezoned.

Note

This option is identical to the default shrink-wrap option in R14.5.

  - The **High** option allows better feature capture and surface remeshing based quality improvement. You can also use sampling coarser/finer than the final surface mesh by specifying an appropriate resolution factor.

- Feature capture can be improved locally using a shortcut (hot-key).

- A new diagnostics tool allows you to find and fix problems in the mesh.

  - Geometry operations assist in finding and fixing gaps, overlaps and intersections in the assembly in preparation for downstream Wrap or Sew operations.

  - Face connectivity operations assist in obtaining a valid surface mesh by finding and fixing problems related to free or multi connected nodes, duplicate faces, islands, and steps. It also assists in prism and volume meshing by identifying and resolving spikes, non-manifold faces, bad normals and leaks. Finally, the wrapper surface can be improved based on the deviation to the original geometry.

  - Quality operations assist in finding and improving the quality of faces based on one or several quality measures.

- A new hole finding and fixing tool has been added.

  - A combination of path tracing between external and internal material points and an auto hole detection mechanism is used to pinpoint the location of the holes.

  - A number of patching operations (automatic and manual) are available.

- A new build topology tool has been added.

  - The build topology supports local sew (i.e., join/intersect) operations to produce a fully connected assembly ready for volume mesh.

  - Each join/intersect pair can be locally examined before executing the local sew.

  - Build topology also supports multibody parts imported from CAD (see below) and multidomain surface mesh from external packages.

  - Many additional tools are available to prepare the topology for volume meshing.

Size Functions

The following enhancements have been made:

- You can read in previously saved size field files using the **File/Read/Size Field...** menu item.

- You can save the computed size field to a size field file using the **File/Write/Size Field...** menu item.

- The size functions user interface indicates when a size field has been read or computed. You can also delete the size field.

- You can display contours of size on selected face zones to visualize the sizes defined after the size field has been read/computed.

- You can also graphically visualize the defined size at the selected probe location when the selection filter is set to **size**.

- You can specify periodicity for the size-field by selecting a source zone for which periodicity has been defined.

- You can apply a scaling factor to filter the size output from the current size-field.

CAD Import

The following enhancements have been made:

- You can specify a size field file to be used to create conformal tessellation during CAD import.

  You can also choose to save the size field file based on the conformal tessellation parameters specified.

- CAD import now uses the fast proximity size function computation.

- You can directly import a mesh object by controlling the object granularity options available in the **CAD Options** dialog box. The object granularity options also allow you to import multibody parts.

Volume Meshing

The following enhancements have been made:

- Prism meshing:

  – Prism zone selection based on mesh object allows selection of baffles.

  – Prism layers can now be split as part of or after prism generation.

  – Additional smoothing and the post-ignore function can be included in the prism generation process, and also when **Auto Mesh** is used.

- Tetrahedral meshing:

  – The global size function can now be used for tet meshing, enabling the use of Body of Influence.

- "Hexcore only" mesh generation is nearly 100% faster.

- Face zones comprising the non conformal interface between cell zones can be separated easily. You can also write a journal file that can then be read into solution mode to create the mesh interfaces automatically.

- Thin volumes can be meshed by selecting source and target zones.

Parallel Meshing

The following enhancements have been made:

- Distributed parallel prism and tet meshing is now available for a pre-decomposed and surface meshed multi-domain.

- The Fluent Launcher contains options to specify the number of meshing processes for parallel prism and tet meshing.

Miscellaneous

The following options are available:

- Local coordinate systems can be created graphically, and are now available in other tools like rotate, revolve, align objects, etc.

- Graphics based vector definition is available for tools like translate, swept surface creation, etc.

- A new tool has been added for creating a revolved surface.

Part V: ANSYS Simulation Products

Release notes are available for the following ANSYS Simulation products:

Chapter 1: Workbench

The ANSYS Workbench platform offers many new features and enhancements. Areas where you will find changes and new capabilities include the following:

1.1. ANSYS Workbench

Enhancements have been made to the following ANSYS Workbench areas:

1.1.1. ANSYS Workbench Enhancements

Non-Parametric Component Updates for External Connection and CFD-Post

For External Connection and CFD-Post-based Results cells, you can specify that a cell will always be updated with a design point update operation, even when the update of the cell will not affect parameter values. To do so, open the cell Properties view and select the **Always Include in Design Point Update** check box.

For more information, see Updating Design Points in the Workbench User's Guide.

Detailed Results from CFD-Post in Workbench Project Reports

The Workbench Project Report can now optionally include detailed results from CFD-Post reports in the Workbench Project report. When you enable the "Publish Report" option in the Properties view for CFD-Post-based Results cells, CFD-Post report content is added to the Workbench project report.

For more information, see Generating Project Reports in the Workbench User's Guide.

Detailed Design Point Results in Workbench Project Reports

It is now possible to get detailed tables, graphs and figures created for each design point in a parametric Workbench project which contain CFD-Post-based cells. This can significantly enrich the value of design points - now you can get detailed results for each design point without resorting to creating "Exported" projects.

Now, when the "Publish Report" option is enabled for projects with design points, the exported report will contain links to sub-reports for each calculated design point. When you select the File > Export

Report menu option, Workbench generates a main project report (including data from the current design point) and a separate sub-report for each of the other design points. You can access these sub-reports via hyperlinks in the new Report column in the Parameter Set Table of Design Points, or in design point tables produced by DesignXplorer.

For more information, see Including Design Point Data in a Project Report and Generating Project Reports in the Workbench User's Guide.

Submit Project Updates from Workbench to Remote Solve Manager

From Workbench, you can submit a project update to RSM by setting the new **Update Option** property (accessible in the Project Update category of the Project Schematic Properties view — right-click on white space in the Project Schematic and select Properties) to **Submit to Remote Solve Manager**. When this option is selected, the RSM-related properties are displayed, allowing you to specify a Solve Manager, queue, and pre-RSM foreground update preferences. The **Project Update** properties are shared with the Parameter Set as **Design Point Update Process** properties. Changes to the values of these properties here will be reflected in the Parameter Set properties, and vice versa.

Note

When a project update is submitted to RSM, only systems above the Parameter Set bar are sent to RSM. DesignXplorer systems can be further updated locally once the remote project update is completed.

For more information, see Submitting Projects to Remote Solve Manager (RSM) for Remote Update in the Workbench User's Guide.

External Model Component System

A new External Model component system allows for specification of an external mesh that can be read into Mechanical as Mesh-Based Geometry.

Faster Execution for Quantity Math Operators

The ANSYS Workbench **Unit Manager** can now cache unit conversions, providing faster execution for large numbers of math operators involving quantities. This enhancement reduced the time required to perform 10,000 calculations on a standard to high-end workstation or laptop by approximately 95%.

SAMCEF CAE Interface

The following model systems have been added to Workbench which support the SAMCEF CAE interface: Linear Buckling (Samcef), Steady-State Thermal (Samcef), Transient Structural (Samcef), and Transient Thermal (Samcef). The SAMCEF CAE interface is not functional in Windows 8, and interfaces with SAMCEF version 14.1-03.

For more information see Linear Buckling Analysis, Steady-State Thermal Analysis, Transient Structural Analysis, and Transient Thermal Analysis in the Workbench User's Guide.

1.1.2. ANSYS Workbench and EKM Integration

Importing Repository Files into a Project

In ANSYS Workbench, a new **Browse from Repository** option is available on the right-click context menu for some software components that also allow the user to attach a file using the **Browse** option. This option is currently available for the import operation for **Geometry** cell, the **Mesh** cell (for imported mesh), and for the **External Data** and **External Model** systems. This option allows you to browse for files in an EKM repository and import the files to the current project. The imported file is then shown in the Workbench Files view, with its repository path distinguishing it from files that are stored only locally. When the file is imported to a project, a version of the file is stored locally so you can continue working on the project without being connected to the repository.

For more information, see Importing Repository Files in the Workbench User's Guide.

New Options for Configuring EKM Repositories

The ANSYS Workbench **Options** dialog (accessed via the **Tools > Options** menu item) now includes a **Repository** section. The options in this section allow you to specify:

- Whether Workbench, upon opening a local repository project, checks the repository for updates
- Whether Workbench, upon closing a repository project, automatically sends changes to the repository

For more information, see Repository and Setting ANSYS Workbench Options in the Workbench User's Guide.

Checking an EKM Repository for Newer File and Project Versions

ANSYS Workbench offers you the option of specifying when the repository will be checked for newer versions of projects, imported files, or projects containing imported files. This behavior corresponds to your selections in the **Repository** section of the Workbench **Options** dialog regarding whether to check for version updates when opening:

- A repository project
- A repository project containing imported repository file(s)
- A local project containing imported repository file(s)

Upon any of these actions, dialogs ask if you would like to check the repository for newer file and project versions. Each dialog includes a **Save my choice and don't ask this question again** check box. If you select this check box, your selection will be saved and the corresponding option in the Options dialog will be updated accordingly. If you select **Yes**, you will be given the option to download any repository changes (to the project and/or imported files) to your local project.

For more information, see Working with ANSYS Workbench Projects Saved in an EKM Repository and Importing Repository Files in the Workbench User's Guide.

Register Workbench Session with EKM Repository

You can register your current ANSYS Workbench session with the EKM Repository. Once registered, the session will display as a job in the EKM **My Jobs** folder. The job allows you to perform and monitor various Workbench operations remotely via the EKM Web Client or EKM Mobile application. Since the

registration is associated with the Workbench session, rather than the project, the registration is valid for the duration of the session; you can open existing or create new projects within the open session without needing to repeat the registration.

For more information, see Registering an ANSYS Workbench Session with an EKM Repository in the Workbench User's Guide.

EKM Individual Workspace

When you connect to an EKM server from Workbench, there is now an **Individual Workspace** in addition to a Shared Workspace. With an Individual workspace, you can only see your own files, which are stored in your **My Data** directory. With all Shared Workspaces, you can see both your own files and shared files within the repository, stored in the **My Data** and **Repository** directories, respectively.

For more information on workspace types and server modes, see the ANSYS EKM User's Guide.

1.2. External Connection

For the 15.0 release, no enhancements have been made to the External Connection add-in:

1.3. Engineering Data Workspace

The following new material models are now available in Engineering Data:

- **Cohesive Zone Material**: The following models are available for modeling interface delamination and debonding. These material models are available for Static Structural and Transient Structural analyses.
  - Exponential for Interface Delamination
  - Bilinear for Interface Delamination
  - Separation-Distance based Debonding
  - Fracture-Energies based Debonding
- **Fracture Criteria** - The following fracture criteria are available for Crack growth specifications. These criteria are available for Static Structural and Transient Structural analyses.
  - Linear Fracture Criterion
  - Bilinear Fracture Criterion
  - B-K Fracture Criterion
  - Modified B-K Fracture Criterion
  - Power Law Fracture Criterion
- Toolbox category "Experimental Stress Strain Data" is renamed to "Hyperelastic Experimental Data".
- Curve Fitting Support for Chaboche Kinematic Hardening
- Curve Fitting Support for viscoelastic models Prony Shear Relaxation and Prony Volumetric Relaxation.

1.4. External Data

The following new features are now available in External Data:

- **Support for Velocity Data**. Velocities can now be imported from text files and applied as loads in a Harmonic Response Analysis.
- **Support for Initial Stress and Initial Strain Data**. Initial Stress and strain can now be imported from text files and used to define the state of a structure at the beginning of a structural analysis.
- **Support for Body Force Density Data**. Body Force density can now be imported from text files, and applied as loads in a structural analysis.
- **Support for CDB Input**. A CDB file can now be used for either data mapping or model creation.

1.5. FE Modeler

The following new features apply to Nastran Import Specifications

New Materials:

- MAT2 - Shell Element Anisotropic Material Property
- MAT9 - Solid Element Anisotropic Material Property

Elements

- CBUSH - Generalized Spring/Damper with element coordinate system option
- CBAR/CBEAM - Beam element offsets

Properties

- PBUSH - Generalized Spring/Damper property

1.6. Remote Solve Manager (RSM)

Enhancements have been made to the following Remote Solve Manager areas:

1.6.1. Cluster Enhancements

Customize Integration with Supported and Non-Supported Clusters

ANSYS Remote Solve Manager (RSM) now provides a mechanism that allows you to customize ANSYS Workbench and RSM's integration with clusters. The custom integration mechanism consists of RSM's ability to execute a set of user-provided scripts that replace RSM functionality for operations such as submitting jobs to the cluster, monitoring job status, interrupting or cancelling job execution, and transferring files to and from the cluster.

RSM allows you to customize integration using either client-side or server-side integration techniques. You can customize cluster integration by modifying (or providing new) scripts that control interaction with the cluster. For custom server integration, you will modify or replace one or more of RSM's job-handling responsibilities, while still taking advantage of RSM's ability to manage and transfer files. For custom client integration, your custom scripts will need to handle all aspects of job handling and must also define how file transfers are done between the client machine and the cluster, as well.

For more information, see Custom Cluster Integration in the Remote Solve Manager User's Guide.

RSM Support for SGE and UGE Clusters

RSM supports the use of SGE and UGE clusters. An SGE option is available on the Cluster tab of the Compute Server Properties dialog. Selection of the SGE cluster type enables the **Parallel Environment (PE) Names** property, which requires you to enter the names of existing **Shared Memory Parallel** and **Distributed Parallel** environments.

Note

RSM does not support the use of SGE clusters for Polyflow.

For more information, see Compute Server Properties Dialog: General Tab in the Remote Solve Manager User's Guide. For specifics on setting up SGE clusters via the wizard, see Prerequisites for the RSM Setup Wizard in the Remote Solve Manager User's Guide or the Readme file for the 15.0 RSM Setup Wizard (accessed via Start > All Programs > ANSYS 15.0 > Remote Solve Manager).

1.6.2. RSM Setup Wizard Enhancements

Start Linux Services via the RSM Setup Wizard without Root Permissions

On Linux, the RSM Setup Wizard allows you to start RSM services with non-root administrative permissions (i.e. with a user account in the **rsmadmins** user group). In this scenario, the service will be run by that account in non-daemon mode. Before running the wizard, your IT department must create the **rsmadmins** user group manually and add any users who will be starting/running non-daemon services.

For more information, see Prerequisites for the RSM Setup Wizard in the Remote Solve Manager User's Guide or "Installation Prerequisites" in the Readme file for the 15.0 RSM Setup Wizard (accessed via Start > All Programs > ANSYS 15.0 > Remote Solve Manager).

Wizard Installs ANSYS Workbench Prerequisites on All MS HPC and Windows LSF Cluster Nodes

For MS HPC and Windows LSF clusters, the Remote Solve Manager Setup Wizard now installs ANSYS Workbench prerequisites on all nodes in the cluster. For version 15.0, the new prerequisites installed include MS .NET Framework 4.0 Redistributable and MS VC++ 2010 Redistributable x64.

For more information, see Overview of the RSM Setup Wizard in the Remote Solve Manager User's Guide.

1.6.3. RSM Enhancements

Submit Project Updates to Remote Solve Manager

You can submit a project update to RSM by setting the new Update Option property (accessible in the **Project Update** category of the Project Schematic Properties view) to **Submit to Remote Solve Manager**. When this option is selected, the RSM-related properties are displayed, allowing you to specify a Solve Manager, queue, and pre-RSM foreground update preferences.

The **Project Update** properties are shared with the Parameter Set as **Design Point Update Process** properties. Changes to the values of these properties here will be reflected in the Parameter Set properties, and vice versa.

For more information, see Submitting Projects to Remote Solve Manager (RSM) for Remote Update in the Workbench User's Guide.

Use SSH Protocol for All Linux Compute Servers

RSM provides an option which allows you to specify that all Linux Compute Servers (not only those in clusters) use SSH for inter- and intra-node communications. This setting is useful for scenarios where RSM jobs are set to run in distributed parallel on a single Compute Server machine. To set this option, open the General tab of the RSM Compute Server Properties dialog and select the **Use SSH protocol for inter- and intra-node communication (Linux only)** check box.

Note

If Fluent, CFX, Mechanical, or Mechanical APDL are configured to send solves to RSM, they will automatically use the same RSH/SSH setting as RSM.

Improved Responsiveness of the RSM User Interface

When a large number of jobs (>100) have been submitted to a Compute Server for execution, the selection of interface items (such as a job, job log, menu or toolbar entry, or any item in the RSM Tree) does not impact the responsiveness of the RSM user interface.

1.6.4. RSM User Interface Enhancements

Line Numbers and Time Stamp for the RSM Job Log View and Job Report

You can now enable or disable the display of line numbers and a time stamp on the RSM Job Log view and Job Reports. For the Job Log view, you can select or deselect the **Line Numbering** and **Time Stamp** options on the right-click context menu. For a Job Report, you can select or deselect the **Include Log Time Stamp** or **Include Line Numbering** check boxes on the **Save Job Report** dialog that appears when you generate the report.

For more information, see Job Log View in the Remote Solve Manager User's Guide.

Status Icon in the RSM Status Bar

The RSM Status bar now indicates the status of the currently running operation by showing either a **Ready** icon or a **Busy** icon in its left corner. When the **Busy** icon is displayed, the corresponding text indicates which operation is running.

For more information, see Status Bar in the Remote Solve Manager User's Guide.

1.6.5. RSM Design Point Enhancements

Immediate Design Point Submission for Pre-RSM Local Updates

When RSM performs a pre-RSM local update (i.e. the **Pre-RSM Foreground Update** property is set to **Geometry**), updated design points are submitted immediately for remote execution, rather than waiting for full completion of the pre-RSM local update.

- If the **Default Job Submission** property is set to **One Job for Each Design Point**, each design point is submitted as soon as the Geometry update for that design point is complete.
- If the **Default Job Submission** property is set to **Specify Maximum Number of Jobs**, each group of design points is submitted as soon as the Geometry update for that group is complete.

For more information, see Updating Design Points via Remote Solve Manager (RSM) in the Workbench User's Guide.

Improved Merging Capabilities for Simultaneous Design Point Updates

When multiple design point jobs are being executed simultaneously via RSM, RSM now merges all available design point results into the current project at once, rather than performing a separate merge operation for each design point result.

For more information, see Updating Design Points via Remote Solve Manager (RSM) in the Workbench User's Guide.

Use Previous Design Point Solutions for Pre-RSM Local Updates

When multiple design points are submitted as part of a job via RSM and the geometry is updated locally (i.e. the **Default Job Submission** property is set to **Specify Maximum Number of Jobs** and the **Pre-RSM Foreground Update** property is set to **Geometry**), the solution for an updated design point can now be reused by subsequent design points.

For more information, see Updating Design Points via Remote Solve Manager (RSM) in the Workbench User's Guide.

Improved Control of Parallel Execution for Design Point Updates

RSM includes new **Parameter Set** and **Solution** (or **Analysis**) component properties that provide improved control of parallel execution of design point updates and allow you to override the number of parallel processes.

For more information, see Updating Design Points via Remote Solve Manager (RSM) in the Workbench User's Guide.

1.7. System Coupling

New features and enhancements to System Coupling introduced in Release 15.0 are highlighted in this section.

- New System Coupling connections in the Workbench schematic:

  - Steady-state and Transient Thermal systems for the Mechanical product, and Fluid Flow (Fluent) systems can provide (in addition to consume) thermal data to other systems via the System Coupling system in the Workbench schematic.

  - Thermal-Fluid-Structural co-simulation is supported by connecting Static and Transient Structural and Fluid Flow (Fluent) systems via the System Coupling system.

- Newly supported connections involving thermal systems enable one- and two-way steady and transient thermal co-simulation. With these new co-simulations, thermal coupling capabilities have been extended from one-way sequential simulation via the External Data system to match the co-simulation capabilities already available for force and motion.

- Support for thermal-fluid-structural co-simulation is enabled within the Structural system's user interface, through the use of coupled field elements (SOLID226 and SOLID227, which are available via a command object) and the use of Fluid Structure Interfaces that support both structural and thermal transfers. For more information, see Thermal-Fluid-Structural Analyses using System Coupling in the Mechanical User's Guide.

- Expert settings are fully supported, and are available in the System Coupling interface. These settings were previously exposed as a Beta feature. For more information, see Expert Settings in the System Coupling User's Guide.

- Sub-stepping is fully supported in the Mechanical application, when Thermal or Structural systems are connected to the System Coupling system in the Workbench Schematic. Sub-stepping allows the Mechanical APDL solver to operate with a smaller time step size than the coupling step.

- Various usage and workflow improvements have also been introduced. Specific examples include:

  - memory and disk usage optimization,

  - restarts facilitated for analyses that experience run-time errors by allowing the selection of previously inaccessible restart points in each of the Fluent, Mechanical and System Coupling systems. For more information, see Recovering from a Workbench Crash in the System Coupling User's Guide.

1.7.1. Numerics Changes

The changes noted below may affect either the convergence to the final solution or the final (fully converged) solution itself.

- In a general analysis, when the solver receiving the motion (such as Fluent) solves before or simultaneously to the solver sending the motion (such as Mechanical), then the value of zero is transferred for incremental displacement in the first coupling iteration of each coupling step. This change will affect motion transfer convergence, but not the final solution.

- When ramping is applied in a transfer of incremental displacement, the reference value used for every coupling step is zero, rather than the value from the previous coupling step. This change will affect motion transfer convergence, but not the final solution.

- Changing from the Intel Fortran 11.1 to 12.1 compiler has led to differences in 4th significant figure of the target values generated by both the conservative and profile preserving mapping algorithms. The changes manifest as a slight redistribution of target data values. Although changes to global values like extrema, averages and sums were not observed during testing, these values could also change slightly. This change will affect the final solution.

- When using Solution Stabilization in the Fluent application, the Scale Factor's default value has changed from 0.1 to 0.0. The previous value of 0.1 was arbitrary since the effect of this setting is extremely problem dependent. Note that the new scale factor of 0.0 is equivalent to not activating stabilization. You need to enter a non-zero value for this setting to have any effect. This change will affect convergence of the coupled analysis, but not the final solution.

1.7.2. Known Limitations

- When part of a coupled analysis, the Mechanical APDL solver will produce incorrect results and/or fail to converge if **Large Deflection** is changed to **Off** (default is **On**) in the **Analysis Settings** within the Mechanical application's user interface. If you would like to turn off Large Deflections, you must also change **Time Integration** to **Off** (default is **On**).

- When part of a transient coupled analysis, the Mechanical APDL solver will produce incorrect results when receiving Heat Transfer Coefficient and Bulk Temperature values. This error occurs for Transient Thermal systems when the default setting **Program Controlled** is used for **Nonlinear Controls** > **Nonlinear Formulation** in the **Analysis Settings** within the Mechanical application's user interface. Correct results will be generated by using the **Full** setting for **Nonlinear Formulation**.

1.8. TurboSystem

TurboSystem is a set of software applications and software features that help you to perform turbomachinery analyses in ANSYS Workbench.

ANSYS TurboGrid is a meshing tool for turbomachinery blade rows. The release notes for ANSYS TurboGrid are provided in TurboGrid Release Notes.

CFX-Pre, a CFD preprocesor, and CFD-Post, a CFD postprocessor, are part of the ANSYS CFX product. Both of these products have Turbomachinery-specific features. The release notes for CFX-Pre are given at CFX Release Notes. The release notes for CFD-Post are given at CFD-Post Release Notes.

Release notes for ANSYS BladeModeler are provided in ANSYS BladeModeler Release Notes (p. 69).

Note

After reviewing these release notes, you are encouraged to see Usage Notes, which describes some known TurboSystem-related workflow issues and recommended practices for overcoming these issues.

1.8.1. Vista CCD

Vista CCD is a program for the preliminary design of centrifugal compressors.

Vista CCD was developed by PCA Engineers Limited, Lincoln, England.

1.8.1.1. Vista CCD New Features and Enhancements

- Vista CCD with CCM can better handle shrouded, process-type compressor impellers.

- The length of the axial impeller is no longer a required input.

- The meridional shape is computed and output using Bezier curves when exporting to BladeGen or BladeEditor. The shape is improved and more consistent between BladeGen and BladeEditor.

- The leading edge location of the impeller is now adjustable, accommodating inlet orientations ranging from axial to radial.
- The dynamic viscosity may be specified. Previously, only the kinematic viscosity could be specified.

1.8.1.2. Vista CCD Incompatibilities

- The Blade Design cell no longer has a **Geometry export style** property. Parametric geometry export is no longer available by default.

1.8.2. Vista CPD

Vista CPD is a program that employs a 1D approach for the preliminary design of pumps.

Vista CPD was developed by PCA Engineers Limited, Lincoln, England.

There are no new features or enhancements in Vista CPD.

1.8.2.1. Vista CPD Incompatibilities

- DesignModeler is unable to change the units for a volute that is generated in Vista CPD. You must select the units in Vista CPD prior to generating the volute.

1.8.3. Vista RTD

Vista RTD is a program for the preliminary design of radial inflow turbines.

Vista RTD was developed by PCA Engineers Limited, Lincoln, England.

1.8.3.1. Vista RTD New Features and Enhancements

- A new impeller loss correlation has been introduced in recognition of recent design trends in automotive turbocharger turbines — in particular, reduced physical size and increased specific flow. The new loss correlation is now the preferred method for Vista RTD. One feature of the new correlation is the ability to specify the tip clearance in order to control the tip clearance losses directly. The old loss correlation is still available as an option.
- You can now directly specify a value for nozzle efficiency.
- The user interface has been simplified for clarity and ease of use.

1.8.3.2. Vista RTD Incompatibilities

- The default settings have changed significantly.

1.8.4. Vista TF

The Vista TF program is a streamline curvature throughflow program for the analysis of any type of turbomachine, but has been developed in the first instance primarily as a tool for radial turbomachinery analysis.

Vista TF is provided by PCA Engineers Limited, Lincoln, England.

1.8.4.1. Vista TF New Features and Enhancements

- Aerodynamics template files (`.aert`), correlation template files (`.cort`), and control template files (`.cont`) now contain a header line that includes the Vista TF solver version. Template files without this additional header line are assumed to be from an earlier version of Vista TF.
- In the control data file, there is additional information in section 2 (`n_points`. `n_speeds`, `n_reserve`) for multi-point calculations, and a new section: section 6. Section 6 is only needed if a change in mass flow distribution across the streamlines is needed (`i_mass` > 0). For details on the format of the control data file, see Specification of the Control Data File (*.con) in the *TurboSystem User's Guide*.
- In the geometry data file, section 5 has new parameters. For details on the format of the geometry data file, see Specification of the Geometry Data File (*.geo) in the *TurboSystem User's Guide*.
- In the aerodynamic data file, sections 10, 11, and 12 have been added. For details on the format of the aerodynamic data file, see Specification of Aerodynamic Data File (*.aer) in the *TurboSystem User's Guide*.

1.8.4.2. Vista TF Incompatibilities

In the correlations data file, the selection of correlations has been renumbered. In particular, `i_loss_type`, `i_dev_type`, and `i_ewb_type` now have different values. For details on the format of the correlations data file, see Specification of Correlations Data File (*.cor) in the *TurboSystem User's Guide*.

1.8.5. Throughflow Analysis System

- You can no longer directly link between Vista RTD and Vista TF by default. Instead, you can use the **Create New** > **Throughflow** capability, which provides a one-click throughflow analysis. This throughflow analysis enables detailed manipulation of the geometry to allow for geometry optimization. The **Create New** > **Throughflow** capability is also available with Vista CCD and Vista CPD.

Chapter 2: ANSYS EKM Release Notes

ANSYS Engineering Knowledge Manager (EKM) 15.0 consists of EKM, the EKM server product, and its companion web application. The following sections provide an overview of new features and enhancements in ANSYS EKM 15.0.

2.1. ANSYS EKM Enhancements

If you have used previous versions of EKM, Release 15.0 offers many significant changes and improvements:

New — EKM Mobile

- EKM Mobile is a new way to access an EKM repository and to control jobs from a mobile phone.

Installation and Configuration

- As of Release 15.0, all servers are shared installations; Individual mode is now a workspace-level configuration. In an individual-mode workspace, non-root users will have access only to their own data (in /My Home/My Data) and to objects in the System folder; they will not have access to the Repository folder or to any other user's My Data folder.
- You can define a default set of Preferences in the `WorkspaceConfig.xml` file that each new user will automatically inherit.
- A new Usage Mode setting in a user's profile permits the display of a minimal interface when running EKM. This simple, streamlined environment is ideal for users who are only interested in running applications and completing work items.
- Partial workspace imports and exports are no longer available. Workspaces must be exported in full.
- The default values for the `maxDirtyObjects` setting in the `ekm.xml` file have been doubled.
- A new `fileTransferService` setting has been added to the `ekm.xml` file. This setting's `rootDir` variable enables you to specify the root directory where files are staged when uploading files to EKM using the File Transfer Client.
- A new `email` setting has been added to the `ekm.xml` file. This setting allows you to specify a configured domain name for all emails, alerts and messages sent by EKM.
- A new `workbenchServerPortRange` setting has been added to the `ekm.xml` file. Using a min-max format, this setting lets you specify the reserved range of ports that the Workbench server will use for listening to command requests from EKM.
- Newer versions of web browsers are supported when running EKM, (Internet Explorer 8+, Firefox 14+, Chrome 21+). Internet Explorer 7 is no longer supported.

Applications

- The **Applications** folder has become a dashboard for launching applications. The following applications are provided by default:
  - Create Custom Application
  - Create Job Template
  - Create Workflow Definition
  - Start Batch Job
  - Start Remote Desktop
- The default applications listed above are also stored in the new **/System/Shared Applications** folder. This public folder is accessible by any user with `Modify` permissions on it. When you create a custom application you have the option of saving it in the **Shared Applications** folder, or the new **My Applications** folder, which is private to you.
- Applications displayed in the **Applications** folder can be bookmarked for easy access.
- You can control the display of applications in the file list window by selecting one of the following filter options: My Applications, Shared Applications, My and Shared Applications, All Applications, or Bookmarked Applications. You can also control the number of columns in which application gadgets are displayed in the file list window.
- The option to create a custom application using a Java class file has been removed, as it is not supported.

Changes to My Home

- Two new folders have been added to **My Home** to facilitate data management:
  - **My Applications** is a built-in folder where you can store custom applications, workflows and job templates that you create.
  - **My Extraction Monitors** is a built-in folder that you can use to monitor and control the extraction of metadata and simulation details from files. Each time that data extraction occurs in EKM, a job monitor object is added to this folder.
- The following folders have been moved from **My Data** to **My Home**:
  - **My Jobs**
  - **My Processes**
  - **My Saved Queries**
- The **My Job Templates** folder (formerly located under **My Data**) has been replaced by the **My Applications** folder, which resides under **My Home**.
- The built-in **My Journals**, **My Reports** and **My Shortcuts** folders have been removed from **My Data**. Journals, reports and shortcuts are now saved directly in **My Data** by default.

Dashboard

- The state of dashboard is persisted on logout and when you move to another view.
- An 'X' icon will not appear for gadgets marked as "essential" in the `WorkspaceConfig.xml` configuration file.
- You can now create a `My Processes` gadget, which displays all active workflow processes.
- A previous restriction that only six gadgets could be defined has been lifted; you can now define any number of gadgets.
- A previous restriction that gadgets could appear in only set places has been lifted; you can now place gadgets anywhere — including overlapping them. To revert gadgets to their default size and layout, use **Reset Layout**.

Queues and External Applications

- Queues are located in the `/System/Servers/Master` folder, and external applications are now listed directly under each queue (previously, **Applications** and **Jobs** folders listed applications).
- The properties of a job submission queue (for example, the job submission system, domain name, operating system and account names) are now set by selecting **Edit** > **Job Submission Settings**. The **Edit Job Submission Settings** dialog also enables you to automatically load default applications for the specified operating system.
- Different icons are provided for the Data Extraction Queue and Job Submission Queues to help differentiate them.
- When defining an external application, you can now specify an **Application Label** that will be used to identify the application in the interface.

Searching the Repository

- In advanced searches, fields that take a user name as input now have an automatically populated option list to make name selection easier. These fields are **Checked-out by**, **Created by**, and **Modified by**.
- Advanced searches now permit searches with sub-expressions based on object types.
- When using a `Date` property in an advanced search, you can now specify a `Within` condition to search within a specific time frame.
- You can now save search queries as reports.

File Transfers

- The EKM Desktop application has been removed in R15. The File Transfer Client is now a Java webstart application that greatly improves performance when uploading files to the repository.
- With EKM Desktop removed, the Windows Explorer Extension is no longer available in R15. Consequently, there will be no EKM-specific menu items in Windows Explorer.
- A new **Upload Files/Folders From Server** tool enables you to transfer files from the EKM server to the EKM repository.

- A new **Download To a Folder on Server** tool enables you to transfer files from the EKM repository to the EKM server.
- You can upload to — or download from — a catalog.
- Object properties are now set *after* an upload rather than during the upload for improved efficiency. Properties for a folder can be applied recursively to the objects in the folder.
- A new user preference enables you to specify whether or not you want archives to be compressed prior to transfer.

Data Extraction and Monitoring

- It is no longer necessary for a solver/application license to be checked out for data extraction.
- Data extraction licensing has been simplified in R15. While data extraction is in progress, an EKM User License will remain checked out for the user who started the extraction, even if the user logs out of the EKM web client.
- Data is now extracted asynchronously when a file is uploaded to the repository, updated using send changes, or created/synchronized if it is a remote file.
- When data is being extracted, a data extraction monitor object is created and displayed in the new **My Extraction Monitors** folder under **My Home**. The monitor reports the current status of the extraction (for example `Running` or `Completed`), the percentage complete, the number of files, and the number of errors. Options are also provided for cancelling data extractions and retrying failed extractions.
- The following data is extracted from MAPDL `.inp`, `.dat` and `.out` files:
  - Analysis Type
  - Element Types
  - ANSYS Version Number

Reports

- A new Process Report can be generated for an active workflow process.
- In WBPZ reports, individual CAE reports have been moved from the **Systems** section to the **Project** section, and the **Systems** section has been removed.

Job Management

- Integration with the EnginFrame application portal changes the way in which interactive queues are defined and interactive jobs are run. EnginFrame manages all aspects of the interactive session and integrates with various job schedulers (LSF, MOAB, Neutro, PBS, Torque) and remote visualization tools (VNC, DCV, HP RGS). An EnginFrame installation package is included with the EKM installation.
- The identification of a queue as "interactive" is no longer specified in the job submission settings. Rather, interactive queues are defined in the `ekm.xml` file, where they are mapped to EnginFrame queues. Every interactive queue that is created contains a predefined `remoteDesktop.app.xml` file that will launch an empty remote desktop session provided that a remote visualization tool is installed on the user's computer. Remote visualization options can be specified in each queue's job submission settings.

- Interactive jobs are launched using the new **Start Remote Desktop** application displayed in **/Applications**. When an interactive session is started, a job object is created in EKM that enables you to connect to the remote session and view detailed information about the job.
- You can now run jobs directly from `*.wbpz` files residing in the EKM repository.
- In the **Execute Job** dialog box, the ability to save job submission settings to a template has been removed. Templates are now created using the **Create Job Template** application.
- A new **Create Job Template** application in the **Applications** folder provides a way to capture job submission settings in a template. You can specify a template name and save it in either **My Applications** or **Shared Applications**. Templates can be created for both Batch and Interactive jobs. General settings include applications, queues, command line arguments, environment variables, native job submission options, and transcript file names. Remote visualization options are provided for interactive job templates. You can define variables to be referenced in command line arguments, job submission options, environment variables and the transcript file using the {$var} notation.
- Double-clicking a job template automatically executes it.
- In process management operations you must specify a queue for batch execution in the following dialog boxes:
  - ProcessPlayer
  - New DX Run
  - New Design Point Run
  - Analysis Execution
  - Execute Batch Job

  If a password is invalid, you are prompted with instructions on how to use RSM to set a new password.
- For **New DP Run** and **New DX Run** actions, new job objects are created in **My Jobs**, while the output folder continues to be used for storing outputs.
- When uploading files to the working directory from the server or repository, you can select multiple files and folders to upload.
- The Status view of a job is more detailed and functional in R15. Information is divided into 3 sections:
  - The **Execution Details** section reports the Status, Start time and End time of the job.
  - The **Inputs and outputs** section provides a link to the Input file coming from EKM as well as a link to the Output folder.
  - The **Job Settings** section displays all of the job submission settings. These include Application, Queue, Command Line Arguments, Number of Cores, Transcript, and Job Submission Options.
- Jobs can be cancelled even if they are queued.

Workbench Integration

- Workbench sessions can now be registered in EKM. When this occurs a job object is created in EKM so that users can monitor and control the Workbench process remotely, even if Workbench has been manually started on a workstation without using EKM.
- When performing a new Design Point run, updating a Workbench project, or running a Workbench server job from EKM, you can choose to run component updates (for example, Fluent updates) on the same machine where Workbench is running (**Serial** mode), or submit component updates to RSM (**Parallel** mode). If you choose parallel processing you can specify additional job submission settings, such as the **Component execution queue** and the **Number of cores**. For parametric jobs you can also specify the **Maximum number of jobs** to be run simultaneously.

EKM Studio

- You can insert a node from the context menu at any time. New nodes are selected automatically when inserted.
- You can insert a transition from the context menu.
- You can duplicate any existing node in the diagram.
- Error messages from operations are tracked on the **Error** tab.
- The scripting language is now set globally in the **Edit Workflow Attributes** dialog. The selected language will be displayed in all dialogs where expressions or macros are defined, ensuring consistent syntax throughout the workflow definition.
- You can disconnect ports from cells.

Batch Node Definitions

Batch Node definitions have changed as follows:

- `nativeOptions` has been removed.
- `inputVar`, `outputVar`, `batchVar`, `batchValidation`, and `outputValidation`, have been removed. To have the same effect as these settings, add simple nodes before and after the batch node.
- The following attributes have also been removed: `batchProcessorClass`, `batchMacro`, and `completionMacro`. These attributes were used to show a custom user interface for batch variables, but you can use a custom dialog node before the batch node to have the same effect.

Workflow Processes

- Workflow processes can now be cancelled after they have been started.
- Job details are now displayed on the **Audit Trail** tab of the Process Player.
- The **Complete** and **Reassign** actions are no longer available in **My Work Items**. These actions can only be performed in the Process Player.
- In the Process Player, the process creator or any `admin` user can reassign the current task from one user to another.

- When reassigning a work item to another user, you can include a note containing instructions or other relevant information that you would like to pass on to the user.
- For input variables that refer to a specific file, a new download icon is displayed next to the file link in the Process Player so that you can download the file directly from the Process Player.

Data Types and Attributes

- The `storeContent` attribute is no longer available as a type attribute.

Usability Enhancements

- All dialogs are now expandable and, when required, the content of dialogs is resized as the dialog window expands.
- The actions you choose from toolbar menus now apply to the selected objects, just as has always been the case with the actions chosen from the right-click (context) menu.
- Objects that normally launch a dialog or process when selected in EKM will also launch automatically when accessed through an email link. Examples include custom applications, workflows, workflow processes, and work items. For example, clicking on a link to a work item will immediately load the work item in the Process Player.
- When you hover over **Repositories to be searched** in the **Expand Search** dialog, a tool tip provides a suggestion to you.
- Object properties are now displayed in a tree, creating a more intuitive display for hierarchical models such as WBPZ and HFSS files.
- Filter options have been added to column headers in the file list window, enabling you to filter the display of contents in the current folder. You can also apply filters in the search results window.
- The time stamps of unchanged files in the `datastore` directory are no longer changed during the garbage collection task. This enables you to use disk space more effectively and perform incremental backups of changes to `datastore`.

Chapter 3: DesignXplorer

Enhancements have been made to the following DesignXplorer areas:

3.1. DesignXplorer Becomes an Open Optimization Platform with ACT Customization

DesignXplorer's open optimization platform uses extensions created with the ANSYS Application Customization Toolkit (ACT) to integrate external optimizers into its own design exploration environment and workflow. An ACT optimization extension defines the capabilities and properties for one or more external optimizers. Once the extension is installed in ANSYS Workbench and loaded to your project, the optimizers defined in it are available as optimization methods that you can select in DesignXplorer. DesignXplorer then delegates the resolution of the optimization problem to the external optimizer.

For information on creating optimization extensions, see the Application Customization Toolkit Developer's Guide and the Application Customization Toolkit Reference Guide. These documents are part of the ANSYS Customization Suite on the ANSYS Customer Portal.

For more information on using optimization extensions in DesignXplorer, see Performing an Optimization with an External Optimizer in the Design Exploration User's Guide.

3.2. Optimization Enhancements

Parameter Relationships in the Optimization Domain

In the Optimization component of DesignXplorer, you now have the ability to add **Parameter Relationships** to the optimization domain. Parameter relationships give you greater flexibility in defining optimization limits on input parameters than the standard single-parameter constraints, enabling you to specify expression-based relationships between multiple input parameters. New parameter relationships can be added by right-clicking Parameter Relationships (under the Domain node) and selecting **Insert Parameter Relationship**. Alternatively, you can select the Domain node or any object underneath it and type details into the New Parameter Relationship row at the bottom of the Table view.

Once a parameter relationship has been created, you can edit the name, left and right expressions, left and right quantity values, and operator. Parameter relationships enabled via the **Outline** view are displayed in the **Candidate Points** table and are eligible to be displayed in the corresponding History charts and sparklines via the **Show Parameter Relationships** property in the Candidate Points **Properties** view.

Note

This feature is not available for Adaptive Single-Objective optimization.

For more information, see Defining Parameter Relationships in the DesignXplorer User's Guide.

Manufacturable Values Parameters for AMO and ASO

For the Adaptive Multiple-Objective and Adaptive Single-Objective optimization methods, you can now use continuous input parameters with Manufacturable Values.

For more information, see Input Parameters in the Design Exploration User's Guide.

Discrete and Manufacturable Values Parameters for MOGA

For the MOGA optimization method, you can now use discrete input parameters and continuous input parameters with Manufacturable Values.

For more information, see Input Parameters in the Design Exploration User's Guide.

New Convergence Criteria Chart

DesignXplorer now provides a Convergence Criteria chart that allows you to view the evolution of the convergence criteria for the selected iterative optimization method. The rendering and logic of the chart varies according to whether a single-objective or multiple-objective optimization method is being used.

The chart is updated iteratively, so you can use it to monitor the progress of the optimization. When the convergence criteria have been met, the optimization stops and the chart remains available in the **Charts** view.

For more information, see Using the Convergence Criteria Chart in the Design Exploration User's Guide.

Project Reporting for Direct Optimization Systems

Direct Optimization systems can now be included in ANSYS Workbench Project reports. When you generate a Project Report for a project containing a DesignXplorer Direct Optimization system, the report will include:

- Information about the optimization, such as properties, objectives and constraints, design points, and candidates
- Charts generated for the optimization
- An Appendix containing a Table of Rating Values and a Raw Optimization Data Table

For more information, see Working with Design Exploration Project Reports in the Design Exploration User's Guide.

Advanced Optimization Options

DesignXplorer now provides advanced options that you can choose to show or hide in the optimization **Properties** view. To display the advanced options, select the **Show Advanced Options** check box in the **Design Exploration** section of the Workbench **Options** dialog (accessed via **Tools > Options**). Advanced properties are displayed in italics.

For more information, see Design Exploration Options in the Design Exploration User's Guide.

New Optimization Properties for MOGA and AMO

The following new Advanced Options are available for the MOGA and Adaptive Multiple-Objective optimization methods:

- **Type of Initial Sampling**
- **Mutation Probability**
- **Crossover Probability**

The following new standard options are also available:

- **Convergence Stability Percentage**
- **Maximum Allowable Pareto Percentage**

For more information, see Performing an Adaptive Single-Objective Optimization or Performing an Adaptive Multiple-Objective Optimization in the Design Exploration User's Guide.

New Optimization Properties for ASO

The following new Advanced Options are available for the Adaptive Single-Objective optimization method:

- **Number of Starting Points**
- **Random Generator Seed**
- **Maximum Number of Cycles**
- **Number of Screening Samples**
- **Maximum Number of Domain Reductions**
- **Percentage of Domain Reductions**
- **Retained Domain per Iteration (%)**

The following new standard option is also available:

- **Convergence Tolerance**

For more information, see Performing an Adaptive Single-Objective Optimization in the Design Exploration User's Guide.

3.3. General Enhancements

Export Design Point Data from DesignXplorer Tables and Charts

The right-click context menu for DesignXplorer tables and charts now includes an **Export Data** menu option. This option allows you to export the parameter values for each design point in the table or chart to an ASCII file, which can then be used by other programs for further processing.

For more information, see Exporting and Importing Data in the Design Exploration User's Guide.

Ability to Exceed the Recommended Number of Input Parameters

In this release, you have the option of proceeding with an update even when you have exceeded the recommended number of input parameters. If more than 20 input parameters are defined and enabled, DesignXplorer will alert you of this by showing a warning message in the **Message** column of the **Outline** view, rather than showing an error message and preventing the update. You can either continue with your update in spite of the message, or can remove the message by reducing project's number of enabled inputs.

Note

Limitations still apply for specific DOE types. Central Composite Design remains limited to 20 input parameters and Box-Behnken remains limited to 12 input parameters.

Trendlines for the Correlation Scatter Chart

In the Parameters Correlation component of DesignXplorer, the **Correlation Scatter** chart now shows the linear and quadratic trendlines. The trendline equations are included in the chart's **Properties** view, chart legend, export data, and the Workbench project report.

For more information, see Using the Correlation Scatter Chart in the DesignXplorer User's Guide.

DesignXplorer Points Indicate Corresponding Design Point

For any point that displays in any of the DesignXplorer **Table** views (such as DOE points, refinement points, direct correlation points, and candidate points), the user interface now indicates the when the point corresponds to a design point in the Parameter Set **Table of Design Points**. When a correspondence exists, the point's **Name** specifies the design point to which it is related, making it unnecessary to create a new design point for a candidate already associated with one.