



Release Notes



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

Release 17.0
January 2016

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

Copyright and Trademark Information

© 2015 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS, AIM and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

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

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, Contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

Global	ix
1. Advisories	x
2. Installation	x
3. Licensing	x
4. Documentation	xi
5. Online Video Access	xi
6. ANSYS Customer Portal	xi
7. Platform Support	xi
7.1. Screen Resolution Recommendations	xii
8. New Product Levels	xii
I. ANSYS Structural Products	1
1. Mechanical Application Release Notes	3
1.1. Incompatibilities and Changes in Product Behavior from Previous Releases	3
1.2. General Enhancements	4
1.3. Performance Enhancements	5
1.4. Graphics Enhancements	6
1.5. Geometry Enhancements	7
1.6. Model Assembly and External Model Enhancements	7
1.7. Contact and Connection Enhancements	8
1.8. Mesh Enhancements	9
1.9. Fracture Enhancements	9
1.10. Composites Enhancements	9
1.11. Analysis Enhancements	9
1.12. Linear Dynamics Enhancements	10
1.13. Loads/Supports/Conditions Enhancements	11
1.14. Solution Enhancements	12
1.15. Rigid Body Solver Enhancements	12
1.16. Explicit Dynamics Enhancements	12
1.17. Results Enhancements	13
1.18. ANSYS Product Improvement Program	14
2. Mechanical APDL	15
2.1. Structural	15
2.1.1. Contact	15
2.1.1.1. Coupled Pore Fluid-Mechanical Contact Interactions	15
2.1.1.2. CONTA177 Line-to-Surface Element Supports 3-D Beam-to-Beam Contact	16
2.1.1.3. General Contact Enhancements	16
2.1.1.3.1. 3-D Beam-to-Beam and Beam-to-Surface Contact in a General Contact Defini-	
tion	16
2.1.1.3.2. Rigid Target Primitives Supported in a General Contact Definition	17
2.1.1.3.3. Improved Preprocessing and Solution Processing for General Contact	17
2.1.1.4. Enhanced Initial Interface Adjustment with Mesh Morphing	17
2.1.1.5. Calculation of Initial Normal Contact Stiffness for Anisotropic Materials	17
2.1.1.6. Debonding Under the Presence of Friction	17
2.1.1.7. Thermal Contact Modeling for Thermal Shells	18
2.1.2. Elements and Nonlinear Technology	18
2.1.2.1. Mesh Nonlinear Adaptivity	18
2.1.2.2. Structural Infinite Solid Element	18
2.1.2.3. 3-D Shell Stress Output	18
2.1.2.4. 3-D Thermal Surface Effect	18
2.1.3. Material and Fracture Modeling	19

2.1.3.1. Gurson Plasticity with Chaboche Kinematic Hardening	19
2.1.3.2. Density as a Function of Temperature	19
2.1.3.3. Mixed u-P Formulation for User-Defined Materials	19
2.1.3.4. Total Strain Energy Density Output	19
2.1.3.5. Fracture Parameters	19
2.1.3.6. Singularity-Based XFEM Analysis	19
2.1.3.7. Fatigue Crack Growth	20
2.1.4. Geomechanics	20
2.1.4.1. Soil Analysis	20
2.1.4.2. Porous Elasticity	20
2.1.4.3. Geomechanical Plasticity Material Models	20
2.1.5. Linear Dynamics	21
2.1.5.1. Residual Response Enhancements	21
2.1.5.2. Support for Multiple Units Systems in DDAM Analysis	21
2.1.5.3. Remote Modal Files Usage for Spectrum Analyses	21
2.1.5.4. Plain Cylindrical Journal Bearing and Squeeze Film Damper Modeling	21
2.1.5.5. Modal Assurance Criterion Calculation Enhancements	22
2.1.5.6. COMBIN14 Enhancements	22
2.1.5.7. Multiple Load Vectors Generation and Usage in Mode-Superposition Analysis	22
2.1.6. Transient Dynamics	22
2.1.6.1. Initial Velocity as a Sum of Rotational and Translational Velocities	22
2.1.6.2. Initial Condition Supports Initial Acceleration	22
2.2. Multiphysics	22
2.2.1. Acoustics	23
2.2.2. Thermal	23
2.2.3. Diffusion	24
2.2.4. Coupled-Field	24
2.2.5. System Coupling	25
2.2.6. Low Frequency Electromagnetics	25
2.3. Solvers	25
2.3.1. Sparse Solver Enhancements	25
2.3.2. PCG Solver Enhancements	25
2.3.3. Distributed ANSYS Enhancements	25
2.3.4. GPU Acceleration Enhancements	25
2.4. Commands	26
2.4.1. New Commands	26
2.4.2. Modified Commands	26
2.4.3. Undocumented Commands	29
2.5. Elements	31
2.5.1. New Elements	31
2.5.2. Modified Elements	31
2.5.3. Undocumented Elements	33
2.6. Other	33
2.6.1. Results File Compression	33
2.7. Documentation	33
2.7.1. <i>Technology Demonstration Guide</i>	33
2.7.1.1. Forced Response Analysis of a Mistuned Bladed Disk	33
2.7.1.2. C*-integral Evaluation for 3-D Surface Flaws	33
2.7.1.3. Wire Crimping Modeled with General Contact	34
2.7.2. <i>Feature Archive</i>	34
2.7.3. Documentation Updates for Programmers	34
2.7.3.1. Routines and Functions Updated	34

2.8. Known Incompatibilities	34
2.8.1. Elements with New Real Constants	34
2.8.2. PIPE288/PIPE289 Output	35
2.8.3. COMBI214 in Nonlinear Transient Analyses	35
2.8.4. Jobname.ESAV File Required for SPRS and DDAM Spectrum Analyses	35
2.8.5. Tangential Contact Stiffness for Bonded and Rough Contact	35
2.8.6. Debonding with Friction	35
2.8.7. Gurson Plasticity Model with Chaboche Kinematic Hardening	35
2.8.8. Release Number Removed from File Names	35
2.8.9. Changes in Format of ASCII View Factor File	36
2.8.10. XFEM Analysis Database Format Change	36
2.8.11. Windows 10	36
3. Autodyn	37
3.1. New Features and Enhancements	37
4. Aqwa	39
4.1. Aqwa Solver Modules	39
4.1.1. Global Sloped Seabed	39
4.1.2. Nonlinear Roll Damping	39
4.1.3. Seabed Friction on Dynamic Cables	39
4.1.4. Structure-Structure Cables	39
4.2. Hydrodynamic Analysis Systems	39
4.2.1. Additional Nodal Information	39
4.2.2. Extended Stability Analysis Capabilities	40
4.2.3. Spreading Function and Carpet Spectrum	40
5. Beamcheck	41
6. Fatjack	43
7. ANSYS Composite PrepPost (ACP)	45
7.1. New Features in ANSYS Composite PrepPost (ACP) 17.0	45
7.1.1. Variable Material Data	45
7.1.2. Enhanced Composite Workflows	45
7.1.3. Interactive Lay-up Definition in Excel	45
7.1.4. Script-Based Definition of User-Defined Plots	45
7.1.5. Enhancement of Expression Parameters	46
7.1.6. New Sandwich Failure Criteria	46
7.2. New Features in ANSYS Composite PrepPost (ACP) 16.2	46
7.2.1. CAD Selection Rule	46
7.2.2. Surface Section Cuts and BECAS Interface	46
7.2.3. Control Drop-Off Elements on Top and Bottom Surface	46
7.2.4. Reordering Extrusion Guides in Solid Models	46
7.2.5. User-Defined Plots	46
7.2.6. Draping and Angle Plots	47
7.2.7. Composite Failure Criteria	47
7.3. New Features in ANSYS Composite PrepPost (ACP) 16.1	47
7.3.1. Performance Improvements	47
7.3.2. Solid Model Sensor	47
7.3.3. GUI Improvements	47
7.3.4. Additional Flat Wrap Geometry Formats	48
7.4. Supported Platforms for ANSYS Composite PrepPost (ACP) 17.0	48
7.5. Known Incompatibilities	48
II. ANSYS Fluids Products	49
1. Fluent	51
1.1. New Features in ANSYS Fluent 17.0	51

1.2. Supported Platforms for ANSYS Fluent 17.0	59
1.3. New Limitations in ANSYS Fluent 17.0	59
1.4. Resolved Issues and Limitations	60
1.5. Updates Affecting Code Behavior	62
2. CFX	65
2.1. Supported Platforms	65
2.2. New Features and Enhancements	65
2.3. Resolved Issues and Limitations	67
2.4. Updates Affecting Code Behavior	67
3. TurboGrid	69
3.1. Supported Platforms	69
3.2. New Features and Enhancements	69
4. BladeModeler	71
4.1. Supported Platforms	71
4.2. BladeGen	71
4.2.1. New Features and Enhancements	71
4.3. BladeEditor	71
4.3.1. New Features and Enhancements	71
5. CFD-Post	73
5.1. Supported Platforms	73
5.2. New Features and Enhancements	73
5.3. Resolved Issues and Limitations	73
5.4. Updates Affecting Code Behavior	74
6. Polyflow	75
6.1. New Features	75
6.2. Supported Platforms	76
6.3. New Limitations in ANSYS Polyflow 17.0	76
6.4. Past Versions of ANSYS Polyflow Release Notes	76
7. Forte	77
7.1. New Features and Enhancements	77
7.2. Known Limitations	78
7.3. Supported Platforms	78
III. ANSYS Electronics Products	79
1. Icepak	81
1.1. Introduction	81
1.2. New and Modified Features in ANSYS Icepak 17.0	81
1.3. Resolved Issues and Limitations in ANSYS Icepak 17.0	82
IV. ANSYS Geometry & Mesh Prep Products	89
1. DesignModeler	91
2. SpaceClaim	93
3. CAD	95
4. Meshing	97
4.1. Incompatibilities and Changes in Product Behavior from Previous Releases	97
4.2. Nonlinear Mechanical Enhancements	97
4.3. Shell Meshing Enhancements	97
4.4. Mesh Control Enhancements	98
4.5. Mesh Editing Enhancements	98
4.6. Meshing Application Advisory	98
5. IC Engine Release Notes	99
6. ICEM CFD	101
6.1. Highlights of ANSYS ICEM CFD 17.0	101
6.1.1. Initializing blocking	101

6.1.2. Splitting free blocks	101
6.1.3. Editing blocks	102
6.1.4. User interface	102
6.1.5. Output interfaces	102
6.2. Documentation	102
6.2.1. Tutorials	103
7. Fluent Meshing	105
7.1. Changes in Product Behavior from Previous Releases	105
7.2. New Features	105
V. ANSYS Simulation Products	109
1. Workbench	111
1.1. ANSYS Workbench	111
1.1.1. ANSYS Workbench-Remote Solve Manager Enhancements	111
1.1.2. ANSYS Workbench-EKM Enhancements	111
1.2. External Connection	112
1.3. Engineering Data Workspace	112
1.4. External Data	113
1.5. External Model	113
1.6. Enhancement to Mechanical Model Cells	113
1.7. FE Modeler	113
1.8. System Coupling	114
1.9. TurboSystem Release Notes	114
1.9.1. Supported Platforms	114
1.9.2. New Features and Enhancements	114
1.10. CFD-Post in Workbench	115
2. ANSYS ACT	117
3. Remote Solve Manager (RSM)	119
3.1. New Features and Enhancements	119
3.2. Resolved Issues and Limitations	119
4. EKM	121
4.1. New Features and Enhancements	121
4.2. Issues Resolved in this Release	124
4.3. Issues and Limitations	124
5. DesignXplorer	125
5.1. Enhancements	125
5.2. Resolved Defects and Limitations	126
VI. ANSYS AIM	127
1. Advisories	129
2. Enhancements in AIM 17.0	131
3. Enhancements in AIM 16.2	133
4. Enhancements in AIM 16.1	135
5. Limitations	137

Global Release Notes

The release notes are specific to ANSYS, Inc. Release 17.0 and arranged by application/product, with the exception of:

- [Advisories \(p. x\)](#)
- [Installation \(p. x\)](#)
- [Licensing \(p. x\)](#)
- [Online Video Access \(p. xi\)](#)
- [ANSYS Customer Portal \(p. xi\)](#)
- [Platform Support \(p. xi\)](#)

Note that installation- and licensing-specific information is detailed in some application and product sections.

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 \(p. xi\)](#) for the following:

- [ANSYS 16.2](#)
- [ANSYS 16.1](#)
- [ANSYS 16.0](#)
- [ANSYS 15.0](#)

See [ANSYS Customer Portal](#)> Downloads> Previous Releases> ANSYS Documentation and Input Files to download zip files containing the Product and Release Documentation.

The Release Documentation files include the following:

- ANSYS Platform Support Strategy & Plans
- ANSYS, Inc. Installation and Licensing Tutorials
- ANSYS, Inc. Known Issues and Limitations
- ANSYS, Inc. Licensing Guide
- ANSYS, Inc. Quick Start Installation Guide
- ANSYS, Inc. Quick Start Licensing Guide
- ANSYS, Inc. Release Notes
- Linux Installation Guide
- SpaceClaim Release Notes (as applicable)
- Windows Installation Guide

1. Advisories

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 \(p. xi\)](#) 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 17.0. Review these items carefully.

- All installation screens have been enhanced to allow for size adjustment. This feature provides support for both 4K monitors and high resolution tablets.
- The multiple CAD configuration screens have been combined into an "accordion style" single screen. This enhancement simplifies CAD configuration by allowing users to perform all steps within a single view.
- The installation wizard has been enhanced to be easier to use, more intuitive and to provide a consistent feel through out the installation process.
- If the product installation is able to query your license server, the CAD options on the product selection menu are pre-selected based upon the information from your license server. If the installation program was unable to query your license server, no CAD options are pre-selected as in previous releases.
- ANSYS Forte is now included in the product installation and can be selected on the product selection menu for installation.
- ANSYS Viewer can be selected from the product selection menu for installation. The ANSYS Viewer is a stand-alone 3D viewer for Fluent, CFX, and SpaceClaim graphic files.

3. Licensing

The following enhancements have been made to ANSYS, Inc. Licensing for Release 17.0:

- The ANSYS License Manager has been upgraded to FlexNet version 11.13.1.1.
- The product order file has been updated. If you typically change the default order in which products are checked out, be certain to re-order the licenses using either the new automated script or ANSLIC_ADMIN.
- For product order files that have been customized by utilizing the ANSLIC_ADMIN **Set Site Preferences>Specify Product Order** option, you can use the **-updatesiteprefs** command to automatically update your modified product order file to include any product changes that have occurred in the most recently installed release of ANSYS License Manager.
- ANSYS License Management Center now includes an option to display the FlexNet licensing activity (Imstat). This View FlexNet License Status option displays license server information, vendor daemon status and license usage information.

4. Documentation

Documentation is now installed per-product; only the documentation associated with the products you install will be included by default. You can choose to install all documentation by running a documentation-only install from the installation launcher.

Help Viewer Enhancements

The Help Viewer has been updated with enhanced capabilities:

- When using the Help Viewer, you can now enter a term in the Search box in the Toolbar and press Enter to initiate a search.
- You can now search on MAPDL commands that begin with special characters (*, /, ~) without enclosing the term in quotes.

Updated Product Documentation

Visit the ANSYS Customer Portal [Online Documentation](#) page to view or search the latest updates to the Release 17.0 documentation.

5. Online Video Access

To review an extensive library of How-To Videos that detail how to use ANSYS product features, go to the ANSYS How-To Videos YouTube page at [YouTube](#).

6. 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>.

7. Platform Support

Information about past, present and future platform support is viewable via the ANSYS, Inc. website (Support> [Platform Support](#)).

Included on the web page is:

- ANSYS Platform Support – Summary of Strategy and Plans
- Browser Support
- CAD Support
- Graphics Cards Tested
- Remote Display Support
- Interconnects Support
- Platform Support by Application

7.1. Screen Resolution Recommendations

Minimum supported screen resolution (except handheld mobile)

- 1024 x 768 (4:3 aspect ratio)
- 1366 x 768 (16:9 aspect ratio)
- 1280 x 800 (16:10 aspect ratio)

Recommended resolution

- 1920x1080 (16:9)
- 1920x1200 (16:10) or higher

8. New Product Levels

For Release 17.0, product levels have been consolidated and updated to include many more capabilities per product level. Except for the LS-DYNA Export capability that has been removed from the Explicit STR implementation in Mechanical Enterprise, no features have been removed from individual products. No features have been removed from individual applications such as ANSYS Mechanical APDL or ANSYS Workbench Mechanical. Your current license will continue to function as usual until you upgrade to the new release.

The following table outlines the capabilities of the new product levels at Release 17.0:

Product Name	Capabilities
DesignSpace	Same as previous DesignSpace product
ANSYS Mechanical Pro	New product at Release 17.0 featuring: <ul style="list-style-type: none"> • Static Stress Analysis • Steady-State and Transient Thermal Analysis • Modal Analysis • Eigenvalue Buckling Analysis • Fatigue Analysis (ANSYS Fatigue Module)

Product Name	Capabilities
ANSYS Mechanical Premium	<p>New product at Release 17.0, featuring everything in ANSYS Mechanical Pro and:</p> <ul style="list-style-type: none"> • Static Stress Analysis with Nonlinear Materials • Harmonic Analysis • Linear Transient Stress Analysis • Spectrum Analysis • Random Vibration Analysis • Rotordynamics Analysis • Rigid Body Dynamics (ANSYS Rigid Body Dynamics)
ANSYS Mechanical Enterprise	<p>ANSYS Mechanical renamed as ANSYS Mechanical Enterprise. Nothing removed from previous ANSYS Mechanical product, featuring everything from ANSYS Mechanical Premium and:</p> <ul style="list-style-type: none"> • Advanced Nonlinear Materials • Nonlinear Transient Stress Analysis • Fracture Analysis • Acoustics • Electric and Electrostatic Analysis • Coupled-Field Analysis • Substructuring • Shell and Solid Composites Pre- and Postprocessing (ANSYS Composite PrepPost) • Hydrodynamics (ANSYS Aqwa Suite with Coupled Cable Dynamics) • Explicit Dynamics (ANSYS Explicit STR except for the LS-DYNA Export capability) • Advanced Geometry Creation and Manipulation (ANSYS SpaceClaim Direct Modeler) • Design Optimization and Probabilistic Design (ANSYS DesignXplorer) • Workflow Automation (ANSYS Customization Suite)

Product Name	Capabilities
ANSYS Mechanical Enterprise PrepPost	<p>ANSYS Mechanical PrepPost renamed as ANSYS Mechanical Enterprise PrepPost. Nothing removed from previous ANSYS Mechanical PrepPost product, and includes all PrepPost features of ANSYS Mechanical Enterprise including:</p> <ul style="list-style-type: none"> • Fatigue Analysis (ANSYS Fatigue Module) • Rigid Body Dynamics (ANSYS Rigid Body Dynamics) • Explicit Dynamics (ANSYS Explicit STR except for the LS-DYNA Export capability) • Shell and Solid Composites Pre- and Postprocessing (ANSYS Composite PrepPost) • Hydrodynamics (ANSYS Aqwa Suite with Coupled Cable Dynamics) • Advanced Geometry Creation and Manipulation (ANSYS SpaceClaim Direct Modeler) • Design Optimization and Probabilistic Design (ANSYS DesignXplorer) • Workflow Automation (ANSYS Customization Suite)
ANSYS Mechanical Enterprise Solver	<p>ANSYS Mechanical Solver renamed as ANSYS Mechanical Enterprise Solver. Nothing removed from previous ANSYS Mechanical Solver product, and includes all Solver features of ANSYS Mechanical Enterprise including:</p> <ul style="list-style-type: none"> • Rigid Body Dynamics (ANSYS Rigid Body Dynamics) • Explicit Dynamics (ANSYS Explicit STR except for the LS-DYNA Export capability) • Hydrodynamics (ANSYS Aqwa Suite with Coupled Cable Dynamics)
ANSYS Autodyn	Same as previous ANSYS Autodyn product
ANSYS LS-DYNA	Same as previous ANSYS LS-DYNA product

New products enable all functionalities and multiple applications. However, they are still a single product or a single task, and therefore only allow access to each application sequentially, but not simultaneously. For example, if you have a single ANSYS Mechanical Enterprise license available, you can perform a Mechanical APDL solution, but you would not be able to use SpaceClaim Direct Modeler until the solution is complete.

The HPC capabilities for each of the solvers remain as they were in previous releases. The number of cores allowed to be used by the solver without any HPC licenses is 2 (DMP or SMP) for the Mechanical APDL solver, 1 for the Rigid Body Dynamics solver, 1 for the Explicit solver, and 1 for the Aqwa solver.

If you are running a previous release of ANSYS products with the Release 17.0 licensing manager, you will notice that some additional features have been made available to your earlier product. Although these new features are available and can be used, the previous releases will not contain the full set of features of the new product levels.

Part I: ANSYS Structural Products

Release notes are available for the following ANSYS Structural products:

[Mechanical Application \(p. 3\)](#)

[Mechanical APDL \(p. 15\)](#)

[Autodyn \(p. 37\)](#)

[Aqwa \(p. 39\)](#)

[Beamcheck \(p. 41\)](#)

[Fatjack \(p. 43\)](#)

[ACP \(p. 45\)](#)

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
- 1.2. General Enhancements
- 1.3. Performance Enhancements
- 1.4. Graphics Enhancements
- 1.5. Geometry Enhancements
- 1.6. Model Assembly and External Model Enhancements
- 1.7. Contact and Connection Enhancements
- 1.8. Mesh Enhancements
- 1.9. Fracture Enhancements
- 1.10. Composites Enhancements
- 1.11. Analysis Enhancements
- 1.12. Linear Dynamics Enhancements
- 1.13. Loads/Supports/Conditions Enhancements
- 1.14. Solution Enhancements
- 1.15. Rigid Body Solver Enhancements
- 1.16. Explicit Dynamics Enhancements
- 1.17. Results Enhancements
- 1.18. ANSYS Product Improvement Program

1.1. Incompatibilities and Changes in Product Behavior from Previous Releases

Release 17.0 includes several new features and enhancements that result in product behaviors that differ from previous releases. These behavior changes are presented below.

- **Node Selection** and **Element Selection**. The **Select Type** drop-down list is no longer available in the **Graphics Toolbar**. You now use a **Node** and **Element** filter button on the Graphics Toolbar to directly select nodes and elements in the **Geometry** window.
- **Model Properties in Workbench**. The Workbench project page properties "Process Nodal Components" and "Nodal Component Key" have been renamed to "Create Geometry Face Components" and "Component Key," respectively. The functionality is much the same: your geometry is synthesized from node-based components present in the upstream FE mesh files and imported into Mechanical as Named Selections. By default, the Create Geometry Face Components property is turned off. However, the application automatically imports the node- or element-based Named Selections (FE based) into Mechanical.
- **Weak Springs**. The default setting for the **Weak Springs** property (Analysis Settings>Solver Controls) has been changed from **Program Controlled** to **Off**. This change will help to prevent masking problems in the setup of models that could affect convergence and it will also assist when diagnosing other issues. For Analysis Systems created prior to release 17.0, the Program Controlled setting remains as the default setting.

- **Distributed Solve Process Settings.** Mechanical now defaults to distributed mode when running solutions. The setting **Distributed Solution** is selected by default.
- **Fracture Analysis.** The child object of the **Fracture** folder, "Crack", has been renamed "Semi-Elliptical Crack" and the **Fracture** property within the Analysis Settings object has been moved from the **Solver Controls** category to the **Fracture Controls** category.
- **Body:** The "**Behavior**" property, which is used to set 2D analysis behavior on geometry, has been renamed to "**2D Behavior**".
- **Contact Result Trackers.** Changes for Contact Result Trackers include the following:
 - The default setting for the **Enhanced Tracking** property is now set to **Yes**.
 - As a result of new Contact Tracker outputs being added, the **Normal Stiffness** output type has been renamed **Max Normal Stiffness**. Legacy databases will still label this output as "Normal Stiffness."
- **Exporting FE Connections.** The ability to export finite element connection information (constraint equations, beams, and weak springs) via the **Solution Information** object is **no longer supported**.
- **Resetting Graphical View following Geometry Refresh.** You can now change the setting for whether a geometry refresh will reset the graphical view in Mechanical. The default setting is **No**.
- **Radiation Probe.** The "**Net Radiation**" property of the **Radiation** probe has been renamed "**Outgoing Net Radiation**." This was done to reflect that the application may produce a negative value for the Net Radiation, indicating that heat is *entering* the scoped entity.
- **Nonlinear Adaptive Region.** The "**Skewness**" option of the "**Criterion**" property has been renamed to "**Mesh**."

Note

When using the **Nonlinear Adaptive Region** condition during 2D analyses on the Linux platform, Mechanical does not support the **Mesh** option of the **Criterion** property.

- **Nonlinear Adaptivity Controls.** The "**Concave Patch Angle**" and "**Convex Patch Angle**" properties have been replaced by the "**Boundary Angle**" property. The default value for this new property is 15° for 3D analyses and 10° for 2D analyses.
- **Reaction Probes.** In the past, scoping Reaction Probes to a Remote Displacement that was applied to a rigid body in Modal, MSUP Harmonic (standalone or linked), MSUP Transient, Response Spectrum, or Random Vibration analyses could have yielded incorrect values for certain problems due to over-constraint. For this reason, Reaction Probes no longer support Remote Displacement as a Boundary Condition property selection under these conditions.

1.2. General Enhancements

The following general enhancements have been made at Release 17.0:

- **ANSYS Customization Toolkit (ACT) Console.** To enable you to interactively use the ACT Mechanical APIs, the **Standard Toolbar** in Mechanical now includes an option that opens the **ACT Console**.

- **Searching the Tree. Find in Tree** (available from the Edit Menu or using the **F3** shortcut key) enables you to perform a string-based search on the entire tree to select an object or objects.
- **Worksheet View for Named Selections.** You can now view and sort all of your Named Selections in a table using the Worksheet.
- **Hotkey Additions.** Hotkey additions have been added for the following options:
 - **Extend Selection** menu, including Extend to Adjacent, Extend to Limits, Extend to Instances, and Extend to Connection.
 - **Solve.** The **F5** key now initiates the solution process.
- **Group Tree Objects.**
 - The grouping feature is now more widely available across tree object types.
 - Hotkey combinations have been added to further ease of use of the grouping feature. For example, you can use **Ctrl+G** to group and **Shift+Ctrl+G** to ungroup objects.
 - You can now Hide (and Show) bodies grouped under the **Geometry** object.
 - You can now Suppress/Unsuppress grouped objects.
- **Filtering the Tree.** The **Tree Filter Toolbar** has been separated into two separate toolbars and you can now hide the filtering options in order to better manage screen space.
- **Specifying Named Selections using Worksheet Criteria.** The following are new **Worksheet** criterion options:
 - **Thickness:** enables you to specify the thickness of a surface or shell body when creating a Named Selection.
 - **Element Connections:** enables you to generate node-based Named Selections based on how many elements a node is connected to.
 - **Body Name:** enables you to generate a Named Selection based on body name. When selecting by body name in the tree, the operators include: **Equals**, **Not Equals**, and **Contains** (for finding partial matches).
- **Toolbars Menu.** The selections of the **Toolbars (View Menu)** option are now available by right-clicking anywhere in the toolbars area of the User Interface enabling you to display or hide the selections.

1.3. Performance Enhancements

The following performance enhancements have been made at Release 17.0:

- **Mapping Speed Improvements.** The performance of Triangulation, Distance Based Average, as well as Kriging Algorithms have improved significantly. Mapping speeds are as much as 15 times faster than in previous releases.
- **Calculating Element Orientations.** Element Orientation calculations are now multi-threaded and use a faster search algorithm. Calculations are as much as 14x faster on most models.
- **Display Improvements.** Significant display improvements have been made for imported and spatially varying loads that are scoped to bodies. Display times are as much as 50x faster on large models.

- **Viewing Finite Element Connections.** Memory usage and the time required to display Constraint Equation connections has been improved. This is especially true for large models.
- **Calculating Response PSD Result.** Response PSD result calculations are now multi-threaded on Windows Operating Systems. The result evaluation time is as much as 100x faster on large models.
- **File Management Enhancements for PSD Analysis.** Mechanical has a new file management method for **Random Vibration** analyses that improves solution times by not copying the Modal Solution file for "In Process" solutions. For larger models, solution processing time can be as much as two times faster than previous releases.
- **Result Evaluation Improvements.** For very large models that contain numerous bodies, element-based results, such as stress and strain results, now evaluate as much as 5x faster.
- **Beam Body Models.** Memory management has been significantly improved when working with models that include a large number of line bodies and line body edges.

1.4. Graphics Enhancements

The following graphical enhancements have been made at Release 17.0:

- **Cross-Section Display of Beams.** You can now view user-defined beam cross sections displayed on the mesh and on results.
- **Specifying Options.** The option **Shell Expansion Edge Angle** is now available in the **Graphics** category of the **Options** preference settings. When applying shell thicknesses, this option enables you to change the default setting of the angle used to determine whether adjacent element normals are averaged.
- **Capping Section Planes.** You can now "cap" the faces clipped from a model as a result of using section planes on Geometry views. In addition, you can change the color of the capped surface to match the body color of the geometry.
- **Display Close Vertices.** The new **Close Vertices** feature enables you to graphically display tightly clustered vertices on your model. This feature includes a scaling menu.
- **Graphics Preference.** The following **Graphics** preferences are now available.
 - **Font Magnification.** The **Font Magnification Factor** option now controls the font size presented on figures within the **Print Preview** and **Report Preview** (which includes user-defined **Figure** objects), as well as the images captured from the **Image to File** feature.
 - **Graphics Resolution.** The **Graphics Resolution** option now modifies the resolution for the **Image to Clipboard** (and hotkey **Ctrl+C**) contextual menu option.
- **Directional Vector Visibility.** Mechanical has added two new graphical display options on the Graphics Options Toolbar that enable you to display **Ply Normal Direction** and **Element Reference Direction** for imported ply structures.
- **Element Orientation.** The Element Orientation feature now provides an associated display toolbar enabling you to modify how the orientations are displayed in the Geometry window. For example, you can increase and decrease the size and density of the orientation vectors, display the vectors for a specific axis, or display vectors aligned with each element.
- **Exporting Data.** Mechanical now supports the Standard Tessellation Language (STL) file format for exporting geometry and current mesh (that may include result-based deformation scaling) information. Using an ap-

propriate STL supported application, you can render and view this information as it would appear in Mechanical.

- **Color Coding of Parts.** You can now assign colors to materials in the Engineering Data Workspace and have these colors display in Mechanical.
- **Edge Graphics Options.** A new **Edge Coloring** option, **By Body Connection**, shows connectivity between bodies at the mesh level.

1.5. Geometry Enhancements

The following geometry enhancements have been made at Release 17.0:

- **Geometry Export.** You can now export your entire geometry or selected parts of the geometry from Mechanical as a binary Part Manager Database (.pmdb) file.
- **Geometry from Deformation Results.** You can now transfer the deformed geometry from a Structural, Modal, Eigenvalue Buckling, or Explicit Dynamics analysis and use it as the initial geometry in a downstream Mechanical model or analysis system. You can specify a Time or Mode, and a Scale Factor for generating the deformed geometry. In addition, Model Assembly and Mesh Conversion settings are available to modify or assemble the new geometry.
- **Mesh Cross-Sections for Beams:** User-defined cross sections defined and assigned to **line bodies** in DesignModeler or ANSYS SpaceClaim Direct Modeler are now imported into the Mechanical application as mesh cross-sections. You can choose to send these cross sections to the MAPDL Solver as a **Pre-Integrated** (default) cross-section or as a **Mesh** section using the new property **Cross Section (For Solver)**.
- **Element Orientation.** The **Element Orientation** feature now supports scoping for individual elements in addition to body scoping. Element scoping supports either direct element selection or you can define **element-based** Named Selections. And, you can now define Element Orientation using a predefined Coordinate System, in addition to the Surface and Edge Guide method.
- **Named Selections.** For face- and edge-based Named Selections, new properties are available that display the **Surface Area** (for faces) and **Length** (edges).

1.6. Model Assembly and External Model Enhancements

The following enhancements for [Model Assembly and External Model](#) have been made at Release 17.0:

External Model

The following enhancements are specific to the **External Model** component.

- The **External Model** system now enables you to import the following items into Mechanical:
 - **Materials.** Materials defined in Mechanical APDL common database (.cdb), ABAQUS Input, or NASTRAN Bulk Data files can now be imported into Mechanical by linking the **External Model** Component system to the Engineering Data workspace.
 - **Line Bodies.** Beam element-based bodies can be imported from mesh files using the **Process Line Bodies** Mesh Conversion option.
 - **Node and Element Components.** You can now choose to import node- and/or element-based components from .cdb files to Mechanical as Named Selections. You can also specify components

keys in order to filter the components contained in your mesh file. The following new properties are available for the **External Model** component system to support these new capabilities:

- **Process Nodal Components** (active by default)
- **Nodal Component Key**
- **Process Element Components** (active by default)
- **Element Component Key**
- **Coordinate Systems.** Coordinate Systems from mesh-based files can now be imported using the **Process Coordinate Systems** option.
- A new **Mesh Conversion Options** property has been added, **Vertex Insertion Angle**, when importing **mesh-based geometries** using the **External Model** system. This property defines the threshold angle for splitting an edge.

Model Assembly

The Model Assembly feature now allows the following objects, defined in upstream Mechanical systems, to be imported into the downstream system:

- Beam Connections
- Command object
- Joints
- Point Mass
- Remote Points
- Springs

1.7. Contact and Connection Enhancements

The following contact and connection enhancements have been made at Release 17.0:

- **Beam Behavior for Remote Points:** The option **Beam** is now available for the **Behavior** property of the **Remote Points** object. This option uses linear massless beam elements to connect your model or to apply remote loads. This option requires you to define a **Material** and a **Radius** for the beam cross section. This new behavior can be useful to prevent over-constraint issues that may occur with CE's, to model shells, or to account for material effects.
- **Contact Result Tracker.** For this feature:
 - You can now add a Contact Result Tracker while a solution is in progress.
 - New contact output types have been added to fully support all possible solver outputs, such as the number of contacts that include too much Penetration and Stabilization Energy.
- **Quickly Create Objects Based on Contact Regions.** Mechanical enables you to easily create mesh connections and certain result objects using the new RMB **Create** option that is available on **Contact Region** objects.

- **Connection Group Contact Detection. Cylindrical Faces** is a new contact detection property that enables you to include or exclude cylindrical faces.

1.8. Mesh Enhancements

Refer to the 17.0 Release Notes of the [Meshing](#) application for the new features and enhancements associated with Meshing in the Mechanical application.

1.9. Fracture Enhancements

The following fracture enhancements have been made at Release 17.0:

- **Arbitrary Cracks.** You can now define both planar and non-planar cracks of any arbitrary shape. This new arbitrary crack:
 - Uses a surface body to define the crack surface.
 - Is meshed with tetrahedron elements.
 - Is supported by Static Structural and Transient Structural analyses.
 - Supports computation of fracture parameters based on Stress Intensity Factors (SIFS), J-Integral, Virtual Crack Closure Technique (VCCT), Material Force, T-Stress, and C*-Integral.
- **Fracture Meshing.** A tetrahedron element-based crack mesh is now supported for Semi-Elliptical cracks.
- **Fracture Results.** You can now compute Material Force, T-Stress, and C*-Integral fracture parameters for all crack definitions (semi-elliptical, arbitrary, or pre-meshed) in either Static Structural or Transient Structural analyses.

1.10. Composites Enhancements

The following composite enhancements have been made at Release 17.0:

- **Additional Result Types for Composite Analysis.** Mechanical now supports ply-based **User Defined** results and **Elemental Coordinate System** results. These results are supported through [traditional Details view scoping](#) as well as through the [automated context menu \(right-click\) option](#) when you select one or more Ply objects.
- **Specifying Ply-based Named Selections using Worksheet Criteria.** You can now define ply-based Named Selection using the Criterion option of the Worksheet.
- **Ply Graphics Options.** In addition to Ply fiber direction, transverse direction, and Element Normal direction, Mechanical now enables you to view Ply normal direction as well as Element reference direction using new toolbar options.
- **Shell Composite Workflow.** The workflow for composite shells, when generated from ACP, is now the same as the workflow for composite solids. In addition, you can assemble shell and solids with or without composites in the same Mechanical Model.

1.11. Analysis Enhancements

The following analysis enhancements have been made at Release 17.0:

- **Substructure Analysis.** Mechanical now enables you to specify flexible bodies in your Rigid Dynamics analyses using the **Condensed Part** feature to define substructures. This feature enables you to treat a set of bodies as a single superelement consisting of matrices and load vectors with far fewer degrees of freedom, suitable for the Rigid Dynamics solver.
- **Deformed Geometry Analysis.** You can now quickly perform a follow-on analysis using the deformed structure of a solved Structural, Modal, Eigenvalue Buckling, or Explicit Dynamics analysis. You can specify a Time or Mode, and a Scale Factor for generating the deformed geometry.
- **Trace Analysis.** You can now import the trace layout of a circuit board for use in structural- and thermal-based analyses. Supported file formats include BRD/MCM/SIP, TCB files that were created using Electronic Design Automation (EDA) software (such as Cadence Allegro), Ansoft Neutral Files (ANF) Version 2 in ASCII format, and ODB++ files.
- **Nonlinear Adaptive Region.** A newly added option, **Mesh**, for the **Criterion** property, supports 2D analyses and replaces the **Skewness** option for 3D analyses.

1.12. Linear Dynamics Enhancements

The following enhancements have been made at Release 17.0 for Linear Dynamic features and analyses:

- **User Defined Results. Random Vibration** and **Response Spectrum** analyses now support **User Defined Results**. Random Vibration analyses provide two new result properties: **Scale Factor** and **Probability**. The **Scale Factor** property measures the standard deviation, or sigma, of a zero-mean Gaussian distribution for random vibration results, while the **Probability** property measures the reliability coverage of the random vibration results.
- **Harmonic Analysis** (Full, MSUP, and Linked) now supports:
 - **Deformation** contour results for **Total Velocity** and **Directional Velocity** as well as **Total Acceleration** and **Directional Acceleration**.
 - **Amplitude** contour results (when the **Amplitude** property is set to **Yes**).
- **Response PSD Probe:**
 - Mechanical now calculates an **Expected Frequency** value for the Response PSD Probe. This value is the calculation of the centroid of the area under the response PSD curve with respect to frequency.
 - The Response PSD Probe now enables you to define a specific frequency range as well as calculate a root-mean-square (RMS) value for that frequency range. New associated properties include:
 - Selected Frequency Range
 - RMS Percentage
 - Expected Frequency
- **Reactions: Forces and Moments.**
 - For Random Vibration and Response Spectrum analyses, you can now scope **Force Reaction** and **Moment Reaction** probes to fixed supports and displacements. In addition, the **Force Reaction** probe can be scoped to a grounded spring.
 - The **Bolt Pretension** probe is now supported for Random Vibration and Response Spectrum analyses.

- **Response PSD Tool.** For [Random Vibration analyses](#), the **Response PSD Tool** is a new results tool that enables you to control the sampling points of **Response PSD** probes.
- **Cyclic Symmetry.** [Full Harmonic Response Analysis](#) (including pre-stress effects) now supports the definition of Cyclic Regions in the model.
- **Rigid Bodies.** [Random Vibration](#) and [Response Spectrum](#) analyses now support rigid body definition (i.e., set the **Stiffness Behavior** property for a body set to **Rigid**).
- **Residual Vectors.** You can now calculate or include residual vectors for a stand-alone **MSUP Harmonic Response Analysis** when the **Solution Method** property is set to **Mode Superposition**.
- **Displacement as Base Excitation.** You can now define the Displacement boundary condition as a base excitation, as a relative or an absolute result, during a Mode-Superposition Transient analysis or a Mode Superposition Harmonic Response analysis.
- **Complex Frequency Dependent Loading.** The [Acceleration](#), [Remote Force](#), **Moment**, and [Displacement](#) boundary conditions can now be defined using tabular data to be fully frequency dependent. And in addition to **Magnitude**, the **Phase Angle** can now be frequency dependent.
- **PSD Base Excitation.** Mechanical now supports uniform base excitation in Random Vibration analyses. In addition, functionality for the property's options have changed.
 - The **All Supports** option previously excited only fixed and remote displacements. The option name has changed to **All Fixed and Remote Displacements**. The functionality remains the same.
 - The new **All Supports** option will excite *all* boundary conditions that can be excited individually.
- **Customizable Frequency Ranges.** For a [Harmonic Analysis](#) (Full, MSUP, and Linked) you can now specify custom frequency ranges using the new property **User Defined Frequencies**.
- **Commands in MSUP Harmonic response analyses (standalone and Linked).** You can now choose a solution phase for the content of a **Commands** object to execute.
- **Spring Material Damping.** You can now introduce damping effects into a spring using a Material that includes a constant damping coefficient. Support is based on the analysis type. The application applies damping as viscous for a damped Modal and Full Harmonic Response systems and as structural damping for MSUP systems.
- **Design Assessment.** Solution combination using Design Assessment now supports the selection of Random Vibration and Response Spectrum analysis types.

1.13. Loads/Supports/Conditions Enhancements

The following loads/supports/conditions enhancements have been made at Release 17.0:

- **Heat Flow.** This boundary condition now supports the ability to span multiple bodies and to be scoped to shared topology.
- **Force.** This boundary condition now supports the ability to span multiple bodies.
- **Pressure.** During Static Structural and Transient Structural analyses, you can now apply **Normal To** pressures directly to geometry without creating surface effect elements.

1.14. Solution Enhancements

The following solution enhancements have been made at Release 17.0:

- **Element Violations.** The new property **Identify Element Violations** enables you to identify and view elements that have failed to meet certain solver criteria by scoping the failed elements to a Named Selection. In addition, the application now generates an error message if an element fails to meet the solver criteria.
- **Solution Messages.** When solving in the **In Process** mode using the MAPDL solver, selected warning and error messages are now displayed in the Messages window during the solution. In previous releases, these messages were only reported at the end of the solution process.
- **Solution Object Statistics.** For analysis systems using MAPDL solver, the **Solution** object has new properties that provide solution processing statistics, including the time that the solver takes to complete the solution, the total memory used to solve the analysis, as well as the generated result file size.

1.15. Rigid Body Solver Enhancements

The following Rigid Body Solver enhancements have been made at Release 17.0:

- Beginning in release 17.0, the Rigid Body Solver considers moments of inertia in the selected coordinates system, rather than in the global coordinate system.
- The new **Program Controlled** option has been added to the analysis settings. The option automatically chooses the time integration and the automatic time stepping control based on the current model. Refer to [Time Integration Type - Transient Analysis of Multiple Rigid Bodies Only in the ANSYS Mechanical User's Guide](#) for more information.

1.16. Explicit Dynamics Enhancements

The Explicit Dynamics Analysis System is a Workbench integrated provision of the Autodyn FE (Lagrange) and multiple-material Euler solvers, and Euler-Lagrange Coupling (providing FSI). The ANSYS Autodyn product includes all of the Explicit Dynamics system capabilities, but only those aspects relating to FE (Lagrange) are available with ANSYS Mechanical Enterprise.

The following Explicit Dynamics Solver enhancements have been made at Release 17.0:

- The import of [deformed geometry](#) from an upstream system and the export of deformed geometry into a downstream system in Workbench is now possible for an Explicit Dynamics system.
- [Dimensionally Reduced](#) rigid body behavior is supported, where rigid body meshing simplifies the representation of a rigid model by reducing it to the contact regions and centroid of the rigid body.
- The existing maximum limit of 200 boundary conditions (loads and supports) that can be defined in a model has been removed. An unlimited number of boundary condition definitions can be used.
- The existing maximum limit of 200 initial conditions that can be defined in a model has been removed. An unlimited number of initial condition definitions can be used.
- The [nodal shell thickness](#) is now taken into account in the contact analysis and allows a more accurate shell to shell contact.

- The usability of the Zerilli-Armstrong material strength model has been significantly improved by the implementation of an implicit strain rate correction that will reduce high frequency oscillations that are sometimes observed in the yield surface under high strain rates.
- An Explicit Dynamics solve will utilize 2 cores by default when an ANSYS HPC license is available. If no HPC license is available the Explicit Dynamics solve will run in serial utilizing 1 core.
- The Explicit Dynamics analysis system now supports additional [varieties of MPI software](#): Platform MPI, Intel MPI, and Microsoft MPI (MSCluster).

A new [Explicit Dynamics Analysis Guide](#) is included in the Mechanical Application documentation set. This new guide consolidates the information specific to the Explicit Dynamics solver that was previously scattered throughout the [Mechanical User's Guide](#). In addition, this document gives guidance on using the Explicit Dynamics system with a Structural system to improve the outcome of an analysis.

1.17. Results Enhancements

The following results enhancements have been made at Release 17.0:

- **Reactions: Forces and Moments.** For Static Structural and Full Transient Structural analyses, you can now scope Force Reaction probes and Moment Reaction probes to Geometry.
- **Line Body Pipe Results.** You can now view pipe stress and strain results for Line Bodies (via the **Calculate Beam Section Results** property).
- **User Defined Results for the Mechanical APDL Solver.** Maximum hoop stress (PIPE_HOOP_STRESS) is now available in the Worksheet **Result Summary** enabling you to create a **User Defined Result**. For this User Defined Result, pipe element stresses need to be written to the result file. Therefore, you need to set the **Calculate Beam Stress Results** property to **Yes** when you define this result.
- **Results Legend Options.** **High Fidelity** is a new context menu option is available for the Results Legend. You use this option when [color display issues occur](#) when you specify a very small interval between adjacent bands.
- **User Defined Results.** Using the **Worksheet**, you can now create a **User Defined Result** for element centroids on the X, Y, and Z axes.
- **Contact Result Tracker:**
 - The **Contact Result Tracker** object now provides a context menu option to individually evaluate each Contact Result Tracker object.
 - You can now add a **Contact Result Tracker** during solution processing operations.
- **Automated Result Scoping for Composites.** Mechanical now supports **User Defined Results** and Elemental **Coordinate Systems** results when creating result objects using the RMB insertion option available on ply objects.
- **Result Set Listing.** A new **Tabular Data** context menu option, **Create Equivalent Strain Results**, is available when you are displaying multiple result sets following the solution process (via the **Solution** object).

1.18. ANSYS Product Improvement Program

ANSYS Product Improvement Program. The new ANSYS Product Improvement Program has been implemented for Release17.0. This voluntary program enables ANSYS to anonymously collect and analyze product usage data reported by ANSYS Mechanical. If you want to participate, you can opt in by picking "Yes" in the **Help Menu>ANSYS Product Improvement Program** dialog box.

Chapter 2: Mechanical APDL Release Notes

Release 17.0 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 \(p. 15\)](#)
- [Multiphysics \(p. 22\)](#)
- [Solvers \(p. 25\)](#)
- [Commands \(p. 26\)](#)
- [Elements \(p. 31\)](#)
- [Other \(p. 33\)](#)
- [Documentation \(p. 33\)](#)

Also see [Known Incompatibilities \(p. 34\)](#) and the [ANSYS Customer Portal \(p. xi\)](#) for important information about this release.

2.1. Structural

Release 17.0 includes the new features and enhancements for the following structural analysis disciplines:

- [2.1.1. Contact](#)
- [2.1.2. Elements and Nonlinear Technology](#)
- [2.1.3. Material and Fracture Modeling](#)
- [2.1.4. Geomechanics](#)
- [2.1.5. Linear Dynamics](#)
- [2.1.6. Transient Dynamics](#)

2.1.1. Contact

Release 17.0 includes the following enhancements for structural analyses involving contact:

- [2.1.1.1. Coupled Pore Fluid-Mechanical Contact Interactions](#)
- [2.1.1.2. CONTA177 Line-to-Surface Element Supports 3-D Beam-to-Beam Contact](#)
- [2.1.1.3. General Contact Enhancements](#)
- [2.1.1.4. Enhanced Initial Interface Adjustment with Mesh Morphing](#)
- [2.1.1.5. Calculation of Initial Normal Contact Stiffness for Anisotropic Materials](#)
- [2.1.1.6. Debonding Under the Presence of Friction](#)
- [2.1.1.7. Thermal Contact Modeling for Thermal Shells](#)

2.1.1.1. Coupled Pore Fluid-Mechanical Contact Interactions

The multiphysics capabilities of the surface-to-surface contact elements ([CONTA171](#), [CONTA172](#), [CONTA173](#), [CONTA174](#)) and the node-to-surface contact element ([CONTA175](#)) now support coupled pore

fluid-mechanical-thermal interactions between surfaces or a surface and its surroundings. The elements can be used with the coupled pore-pressure mechanical solid elements ([CPT212](#), [CPT213](#), [CPT215](#), [CPT216](#), and [CPT217](#)).

This capability is useful in geomechanical and biomechanical applications where pore fluid pressure can flow between contacting surfaces, into small gap regions for near-field contact, and from non-contacting surfaces to the environment. For more information, see [Modeling Pore Fluid Flow at the Contact Interface in the *Mechanical APDL Contact Technology Guide*](#).

2.1.1.2. *CONTA177 Line-to-Surface Element Supports 3-D Beam-to-Beam Contact*

Prior to this release, 3-D beam-to-beam contact and 3-D edge-to-edge contact could be modeled only with the 3-D line-to-line contact element, [CONTA176](#). The 3-D line-to-surface contact element [CONTA177](#) now supports 3-D beam-to-beam (or edge-to-edge) contact in addition to 3-D beam-to-surface (or 3-D edge-to-surface) contact.

In a pair-based contact definition that uses [CONTA177](#), the associated 3-D target elements ([TARGE170](#)) can be surface segments (TRIA, QUAD, TRI6, QUA8) or line segments (LINE, PARA). Both 3-D beam-to-surface contact and 3-D beam-to-beam contact can therefore be modeled with one unified contact element type. [CONTA177](#) supports three different scenarios:

- Contact between one beam (or edge) and a solid or shell surface
- External contact between two beams that lie next to each other and are roughly parallel
- External contact between two beams that cross

All three scenarios can be included together or separately in a model.

2.1.1.3. *General Contact Enhancements*

The following enhancements are available for the general contact capability:

[2.1.1.3.1. 3-D Beam-to-Beam and Beam-to-Surface Contact in a General Contact Definition](#)

[2.1.1.3.2. Rigid Target Primitives Supported in a General Contact Definition](#)

[2.1.1.3.3. Improved Preprocessing and Solution Processing for General Contact](#)

2.1.1.3.1. 3-D Beam-to-Beam and Beam-to-Surface Contact in a General Contact Definition

Beam-to-beam contact and beam-to-surface contact are now supported in a general contact definition. The beam-to-surface contact is a special case of the edge-to-surface general contact which was made available in the previous release. The beam-to-beam contact is possible due to new enhancements to the [CONTA177](#) 3-D line contact element, so that now one element type can handle both 3-D beam-to-beam contact and 3-D beam-to-surface contact within a general contact definition.

In addition to creating [CONTA177](#) elements on feature edges of 3-D deformable solids and perimeter edges of shells, the [GCGEN](#) command now also creates 3-D line contact elements on 3-D beams. The general contact feature, therefore, now offers automated contact definition and creation for all contact types, including: 2-D and 3-D surface-to-surface contact; 3-D beam-to-beam contact; 3-D edge-to-edge contact; 3-D beam-to-surface contact; and 3-D edge-to-surface contact. The general contact procedure greatly simplifies the contact specification process and requires minimal user input compared to the pair-based contact definition procedure.

2.1.1.3.2. Rigid Target Primitives Supported in a General Contact Definition

Rigid target primitives and other target segment types requiring radius values (2-D circle; 3-D line, parabola, sphere, cylinder, and cone) are now supported in a general contact definition. The **SECTYPE** and **SECDATA** commands now include a radius subtype (**SECTYPE**,CONTACT,RADIUS) for specifying radius values for target segment types in a general contact definition. All target segment types are therefore now supported. For more information, see [Creating Rigid Target Surfaces in the Mechanical APDL Contact Technology Guide](#).

2.1.1.3.3. Improved Preprocessing and Solution Processing for General Contact

A number of enhancements have been made in the areas of preprocessing and solution processing for general contact. These include:

- Extended use of node components for identifying general contact surfaces on the **GCDEF**, **SECTYPE**, and **KEYOPT** commands.
- Node component name extensions (_TOP, _BOT, _EDGE, _FACE) to limit the scope of surfaces considered when inputting node components on the **GCDEF**, **SECTYPE**, and **KEYOPT** commands.
- A new TABLESOL option on the **GCDEF** command that lists a table of actual general contact interactions considered during solution.
- The ability to specify a range of general contact surfaces when listing contact interactions via the **GCDEF** command.

For more information, see the **GCDEF** command description, and [General Contact in the Mechanical APDL Contact Technology Guide](#).

2.1.1.4. Enhanced Initial Interface Adjustment with Mesh Morphing

During an initial contact adjustment in which contact nodes are physically moved to the target surface to close a gap or reduce penetration, the solid elements underneath the contact elements can now be morphed (**CNCHECK**,MORPH) to improve mesh quality at the contact interface. This enhancement enables initial surface adjustments for cases where large adjustments are necessary (for example, where **CNCHECK**,ADJUST would fail due to severe mesh distortions resulting from moving the contact nodes). For more information, see [Adjusting Large Gaps and Penetrations](#).

2.1.1.5. Calculation of Initial Normal Contact Stiffness for Anisotropic Materials

The initial normal contact stiffness for contact elements that overlay solid elements with anisotropic elastic material properties is now calculated using the elastic moduli along the contact normal direction. This enhancement helps to improve nonlinear convergence.

2.1.1.6. Debonding Under the Presence of Friction

You can now include friction between contact surfaces undergoing debonding, even before debonding is complete. If friction is defined between the contact surfaces, the tangential stress is calculated as the maximum between the tangential stress as governed by the debonding model and the tangential stress as governed by the friction law.

2.1.1.7. Thermal Contact Modeling for Thermal Shells

You can now use the 3-D surface-to-surface contact elements ([CONTA173](#) and [CONTA174](#)) to model thermal contact at the top and bottom surfaces of thermal shell elements ([SHELL131](#) and [SHELL132](#)). The degrees of freedom of thermal shells can be TEMP, TBOT, or TTOP, and the contact can occur at either the top or the bottom surface. A new option on the contact elements (KEYOPT(13)) enables you to control the temperature degrees of freedom (TEMP, TBOT, TTOP) activated on the contact and target surfaces when heat transfer is taking place between thermal shell elements or between thermal shell and thermal solid elements. For more information, see [Modeling Heat Transfer Among Thermal Shells](#) in the [Contact Technology Guide](#).

2.1.2. Elements and Nonlinear Technology

Release 17.0 includes the following enhancements to elements and nonlinear technology used in structural analyses:

[2.1.2.1. Mesh Nonlinear Adaptivity](#)

[2.1.2.2. Structural Infinite Solid Element](#)

[2.1.2.3. 3-D Shell Stress Output](#)

[2.1.2.4. 3-D Thermal Surface Effect](#)

2.1.2.1. Mesh Nonlinear Adaptivity

Remeshing in a [mesh nonlinear adaptivity](#) analysis improves the mesh quality during solution to achieve convergence for difficult nonlinear large-deformation problems.

The [general remeshing method](#) now supports 2-D models. 2-D general remeshing applies to the [mesh-quality-based](#) criteria and is used with the [PLANE182](#) (2-D four-node structural solid) element. The regions to remesh in a 2-D model are initialized by seed elements having a maximum corner angle greater than or equal to a specified threshold.

3-D mesh nonlinear adaptivity now offers [non-manifold](#) detection information output. Support for models involving contact with friction has also been enhanced.

For more information, see [Mesh Nonlinear Adaptivity in the Mechanical APDL Advanced Analysis Guide](#).

2.1.2.2. Structural Infinite Solid Element

The [INFIN257](#) structural infinite solid element now supports static, transient, and harmonic analyses. [SOLID285](#) is now available as a base element. In a static analysis, base elements can now have anisotropic material properties. For more information, see the documentation for [INFIN257](#), and [INFIN257 - Structural Infinite Solid in the Mechanical APDL Theory Reference](#).

2.1.2.3. 3-D Shell Stress Output

A new option is available for the [SHELL181](#), [SHELL208](#), [SHELL209](#), [SHELL281](#), and [ELBOW290](#) elements to obtain shell stress output (KEYOPT(10)). The option outputs normal stress component Sz, independently recovered during the element solution output from the applied pressure load.

2.1.2.4. 3-D Thermal Surface Effect

A new 3-D thermal effect element, [SURF155](#), is now available for 3-D thermal analyses. The element can be used for various load and surface-effect applications. It can be overlaid onto an edge of any 3-D thermal solid element. Various loads and surface effects can exist simultaneously.

2.1.3. Material and Fracture Modeling

Release 17.0 includes the following enhancements to material modeling and fracture analysis technology used in structural analyses:

- 2.1.3.1. Gurson Plasticity with Chaboche Kinematic Hardening
- 2.1.3.2. Density as a Function of Temperature
- 2.1.3.3. Mixed u-P Formulation for User-Defined Materials
- 2.1.3.4. Total Strain Energy Density Output
- 2.1.3.5. Fracture Parameters
- 2.1.3.6. Singularity-Based XFEM Analysis
- 2.1.3.7. Fatigue Crack Growth

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](#).

2.1.3.1. Gurson Plasticity with Chaboche Kinematic Hardening

The hydrostatic portion of the Gurson-Chaboche model's yield surface has been modified. Nonphysical behavior issues related to certain parameter values have been addressed, convergence has been improved, and the constitutive equations in the new model are more consistent with each other. The updated Gurson plasticity model can be stiffer for certain parameter values than in prior releases. For more information, see [Gurson in the Mechanical APDL Material Reference](#) and [Gurson's Model in the Mechanical APDL Theory Reference](#).

2.1.3.2. Density as a Function of Temperature

The ability to specify mass density (**TB,DENS**) as a function of temperature is now available. Density is also a supported material data type for defining field-variable values (**TBFIELD**). For more information, see [User-Defined Field Variables in the Mechanical APDL Material Reference](#).

2.1.3.3. Mixed u-P Formulation for User-Defined Materials

The `UserMat` subroutine for creating your own material model has been enhanced to support [mixed-uP formulation](#), enabling hyperelastic materials considered to be incompressible or nearly incompressible. For more information, see [Subroutine UserMat \(Creating Your Own Material Model\) in the Mechanical APDL Programmer's Reference](#).

2.1.3.4. Total Strain Energy Density Output

A new SEND option (**ENTO**) supports total strain energy density output (**ETABLE**, **PLNSOL**, **PRNSOL**, **PLESOL**, **PRESOL**, and **ESOL**).

2.1.3.5. Fracture Parameters

The UMM method for calculating fracture parameters [J-integral](#) and [stress-intensity factors](#) (SIFs) now supports crack-surface pressure. Also, the J-integral now supports body force. For more information, see [Evaluation of Fracture Mechanics Parameters in the Mechanical APDL Fracture Analysis Guide](#).

2.1.3.6. Singularity-Based XFEM Analysis

The *eXtended Finite Element Method* (XFEM) for modeling cracks and other discontinuities can now account for crack-tip singularities as well as the jumps in displacements across the crack surfaces. With the new [singularity-based](#) XFEM approach, cracks are allowed to terminate either inside a finite element or at

the edge of the element. The new capability is especially useful for 2-D [stationary crack analysis](#) (supported by the [PLANE182](#) element). For more information, see [XFEM-Based Crack Analysis and Crack-Growth Simulation in the *Mechanical APDL Fracture Analysis Guide*](#).

2.1.3.7. Fatigue Crack Growth

Singularity-based XFEM-based crack-growth analysis can also be used to simulate 2-D [fatigue crack growth](#) in engineering structures. The method is based on Paris' Law and offers a convenient engineering approach for simulating cracks and fatigue crack propagation without resorting to actually modeling the cracks or remeshing the crack-tip regions as the crack propagates. The fatigue crack-growth analysis supports linear isotropic materials only, and ignores deflection/rotation effects, crack-tip plasticity effects, and crack-tip closure or compression effects. For more information, see [Fatigue Crack Growth in the *Mechanical APDL Fracture Analysis Guide*](#).

2.1.4. Geomechanics

This release introduces new geomechanics capabilities consisting of the following enhancements:

[2.1.4.1. Soil Analysis](#)

[2.1.4.2. Porous Elasticity](#)

[2.1.4.3. Geomechanical Plasticity Material Models](#)

2.1.4.1. Soil Analysis

A new soil analysis type is available ([**ANTYPE**,SOIL](#)). You can now perform a soil analysis including geostatic stress equilibrium or consolidation. The new analysis type is valid for structural and fluid-pore-pressure degrees of freedom. For more information, see [Porous Media in the *Mechanical APDL Material Reference*](#), [Structural-Pore-Fluid-Diffusion Analysis in the *Mechanical APDL Coupled-Field Analysis Guide*](#), and the [**SSOPT**](#) command.

2.1.4.2. Porous Elasticity

Use the new porous elasticity material model to represent porous materials such as soils or polymer foams displaying nonlinear elastic behavior caused by the effect of voids on the bulk modulus of the material during hydrostatic compression. For more information, see [Porous Elasticity in the *Mechanical APDL Material Reference*](#).

2.1.4.3. Geomechanical Plasticity Material Models

New material plasticity models are available for geomechanical simulations involving soil, aggregate materials, and concrete:

- Use the modified [Cam-clay](#) model (based on the critical state concept) for soil simulation. Cam-clay can also be used with [porous elasticity](#) to model the effect of voids on the elastic behavior of the material.
- Use the [Mohr-Coulomb](#) material model to represent aggregate materials, which begin to plastically deform when shear stress exceeds the internal friction resistance between the material particles.
- Use the [jointed rock](#) material model to represent geologic and aggregate materials with inhomogeneous behavior that causes weakness along planes within the material. Such planes include joints, stratification planes, and geometric stress concentrations or regions of strain localization.
- Use the [Drucker-Prager concrete](#) material model, a combination of two Drucker-Prager yield surfaces, to represent the large differences in tensile and compressive behavior of concrete. The model can also be

combined with the anisotropic [Mohr-Coulomb](#) model to define joints (representing failure along planes of weakness) in the concrete material.

For more information, see [Geomechanics in the Mechanical APDL Material Reference](#).

2.1.5. Linear Dynamics

Release 17.0 includes the following enhancements for structural analyses involving linear dynamics:

- 2.1.5.1. Residual Response Enhancements
- 2.1.5.2. Support for Multiple Units Systems in DDAM Analysis
- 2.1.5.3. Remote Modal Files Usage for Spectrum Analyses
- 2.1.5.4. Plain Cylindrical Journal Bearing and Squeeze Film Damper Modeling
- 2.1.5.5. Modal Assurance Criterion Calculation Enhancements
- 2.1.5.6. COMBIN14 Enhancements
- 2.1.5.7. Multiple Load Vectors Generation and Usage in Mode-Superposition Analysis

2.1.5.1. Residual Response Enhancements

To improve the accuracy of mode-superposition transient or harmonic analysis results, the residual response method is now available (*KeyResp* = ON on the **RESVEC** command). It is similar to the residual vector method (*KeyVect* = ON on **RESVEC**) except that it uses the residual static responses calculated directly from the residual flexibility.

The new method is particularly useful when the residual vector method does not apply (for example, when the residual vector has zero frequency, or when the equations are not symmetric and the unsymmetric eigensolver [**MODOPT**,UNSYM] is used). For more information, see [Using the Residual Vector or the Residual Response Method to Improve Accuracy in the Mechanical APDL Structural Analysis Guide](#).

2.1.5.2. Support for Multiple Units Systems in DDAM Analysis

DDAM spectrum analysis can now be performed in units systems other than BIN. You can specify a units system complying with the mass and length units of your model (**/UNITS**). For more information, see the **DDASPEC**, **VDDAM**, and **ADDAM** commands and [Step 3: Obtain the Spectrum Solution in the Mechanical APDL Structural Analysis Guide](#).

2.1.5.3. Remote Modal Files Usage for Spectrum Analyses

When running a spectrum analysis (SPRS, MPRS, PSD, or DDAM), remote read-only modal files usage is now supported. The new capability is especially useful when running several spectrum analyses based on the same modal analysis. For more information, see the **MODDIR** command.

2.1.5.4. Plain Cylindrical Journal Bearing and Squeeze Film Damper Modeling

The **COMBI214** element has a new option integrating the Reynolds equation for thin fluid film lubrication, assuming a finite length. In a static analysis, the stiffness and damping characteristics of a plain cylindrical journal bearing or squeeze film damper can be calculated using a small perturbation near a user-defined equilibrium position. In a large-deflection nonlinear transient analysis, the bearing forces are calculated based on the bearing definition and the instantaneous displacements and velocities. For more information and examples, see [Using the COMBI214 Element in the Mechanical APDL Rotordynamic Analysis Guide](#).

2.1.5.5. Modal Assurance Criterion Calculation Enhancements

Two enhancements are available for the modal assurance criterion calculation tool (RSTMAC). It now supports the reading of acceleration data from the .UNV file format. For more information, see [Universal Format File Records in the Mechanical APDL Basic Analysis Guide](#). A new relative tolerance (*TolerR* on **RSTMAC**) has been added for nodes matching. It is based on a fraction of the minimum element size in the model. For more information, see [Comparing Nodal Solutions From Two Models or From One Model and Experimental Data \(RSTMAC\) in the Mechanical APDL Basic Analysis Guide](#).

2.1.5.6. COMBIN14 Enhancements

In a full transient analysis, real constants K and CV1 can be defined as tabular parameters using time as the primary variable (*Var1* = TIME on the ***DIM** command). In a static analysis, real constant K can be defined as a tabular parameter using time as the primary variable (*Var1* = TIME on the ***DIM** command). For more information, see the **COMBIN14** element.

2.1.5.7. Multiple Load Vectors Generation and Usage in Mode-Superposition Analysis

The maximum number of load vectors generated during a modal analysis with the element result superposition key activated (*MSUPkey* = YES on the **MXPAND** command) defaults to 1000, but that number can now be increased (*Lab* = NUMLV on the **/CONFIG** command). The limit on the number of load vectors scaled during a mode-superposition analysis (**LVSCALE**) has been removed.

2.1.6. Transient Dynamics

Release 17.0 includes the following enhancements for structural analyses involving transient dynamics:

2.1.6.1. Initial Velocity as a Sum of Rotational and Translational Velocities

2.1.6.2. Initial Condition Supports Initial Acceleration

2.1.6.1. Initial Velocity as a Sum of Rotational and Translational Velocities

In a full-method transient dynamic analysis, the new **ICROTATE** command enables you to specify initial velocity at a node or group of nodes as a sum of rotational velocity about an axis and translational velocities.

2.1.6.2. Initial Condition Supports Initial Acceleration

In transient analyses, you can now specify acceleration as an initial condition (**IC**). For more information, see [Establish Initial Conditions in the Mechanical APDL Structural Analysis Guide](#).

2.2. Multiphysics

Release 17.0 includes the following enhancements for analyses involving multiphysics environments:

2.2.1. Acoustics

2.2.2. Thermal

2.2.3. Diffusion

2.2.4. Coupled-Field

2.2.5. System Coupling

2.2.6. Low Frequency Electromagnetics

2.2.1. Acoustics

Floquet Periodic Boundary Condition A new Floquet periodic boundary condition is available for the infinite periodic acoustic structure. For more information, see [Floquet Periodic Boundary Condition \(FPBC\) in the Mechanical APDL Acoustic Analysis Guide](#), and [Acoustic Boundary Conditions in the Mechanical APDL Theory Reference](#).

Body Loads A new body force load label, SPRE, is available on the **BF** command for defining static pressure. (The CHRGD label for defining static pressure has been removed.)

Perfectly Matched Layers (PML) Enhancement The following enhancements are available for PML:

- A new PML element coordinate system (**PSYS**) is available in case the material is defined in the element coordinate system (**ESYS**). The **ESYS** command no longer defines PML element coordinate systems.
- The PML can now attenuate both the propagating and evanescent wave (**PMLOPT**).

Acoustic Duct Port A new capability is available to launch a specified analytic acoustic mode into a guided duct. For more information, see [Specified Mode Excitation in an Acoustic Duct in the Mechanical APDL Acoustic Analysis Guide](#), and [Analytic Port Modes in a Duct in the Mechanical APDL Theory Reference](#).

Acoustic Analysis Efficiency The following enhancements improve the efficiency of acoustic analyses:

- Efficient one-way mechanical structure-acoustic coupling has been implemented with automatic selection of upstream structural working frequencies or time steps during the acoustic solution.
- The morphing process has been improved and made more efficient for [acoustic linear perturbation](#) solutions. The bisection process has been implemented with mesh morphing occurring during the nonlinear static solution (**AUTOTS,ON**). Setting *PRELP* = YES on the **ANTYPE** command keeps the acoustic mesh unchanged during the prestress solution to improve efficiency.
- Optimizing coupled acoustic element attributes via the **ECPCHG** command compresses the size of the *Jobname.emat* file up to 80 percent.
- In harmonic analyses, the **FLUID130** absorbing element with the symmetric formulation can offer up to a 2x solution-time improvement.
- The acoustic fluid can now be incompressible with an equivalent infinite sound speed (*KEYOPT*(6) = 1 on **FLUID30**, **FLUID220**, and **FLUID221**).

Postprocessing The following enhancements are available for postprocessing an acoustic analysis:

- A radiated far sound field from a vibrating plate can be calculated for one-way coupling without modeling the acoustic domain (**PLFAR** or **PRFAR**).
- Acoustic energy on selected elements can now be calculated; the energy includes acoustic potential and kinetic energy.
- The acoustic quantities calculated via the **PRAS** command can now be obtained via the ***GET,,ACUS** command.

2.2.2. Thermal

2-D and 3-D Radiation Enclosures Allowed in the Same Model You can now include 2-D and 3-D radiation enclosures in the same model and solve them via the [radiosity solver method](#). The new cap-

ability is especially useful for complex models in which some parts are meshed with 2-D elements (planar or axisymmetric) and some with 3-D elements.

3-D View Factor Smoothing Enhancement A memory-efficient, distributed-parallel algorithm is now provided for 3-D view-factor-smoothing calculations, enabling 3-D view-factor-smoothing for models with a large number of radiation surfaces.

User-Defined Thermal Material Enhancements The following improvements are available for the [user-defined thermal material subroutine, UserMatTh](#):

- The user-defined thermal material is now available for the coupled thermal-fluid pipe element, [FLUID116](#), with a temperature-only degree of freedom (`KEYOPT(1) = 1`).
- State variables for layered thermal elements [SOLID278](#) and [SOLID279](#) can now be stored for all layers.

2.2.3. Diffusion

Transport Effect in Diffusion Analysis Diffusion elements [PLANE238](#), [SOLID239](#), and [SOLID240](#) now support the [BF,VELO](#) body load to input the velocity of transport. You can use the new capability to model advection or particle transport due to an external force.

2.2.4. Coupled-Field

The following enhancements are available for analyses involving the coupled-field elements [PLANE223](#), [SOLID226](#), and [SOLID227](#).

New Coupled-Field Analyses The coupled-field elements now support an electric-diffusion field combination, available for these new coupled-field analyses:

- [Electric-Diffusion](#) (`KEYOPT(1) = 100100`)
- [Thermal-Electric-Diffusion](#) (`KEYOPT(1) = 100110`)
- [Structural-Electric-Diffusion](#) (`KEYOPT(1) = 100101`)
- [Structural-Thermal-Electric-Diffusion](#) (`KEYOPT(1) = 100111`)

Electromigration The new [electric-diffusion](#) capability of the coupled-field elements supports electromigration. Use this capability to model the transport of charged particles in an electric field.

Hydrostatic Stress Migration The [structural-diffusion](#) capability of the coupled-field elements now supports hydrostatic stress migration. Use this capability to model the transport of particles due to a hydrostatic stress gradient.

Thermomigration The [thermal-diffusion](#) capability of the coupled-field elements now supports temperature-dependent saturated concentration ([MP,CSAT](#)) and thermomigration (Soret effect). Use this capability to model the transport of particles due to a temperature gradient.

Migration Model The new [migration model \(TB,MIGR\)](#) used with the coupled-field elements enables you to input material data for electromigration, hydrostatic stress migration, and thermomigration. [TB,MIGR](#) supports the atomic (`TBOPT = 0`) and vacancy (`TBOPT = 1`) migration models. You can use the migration model with a coupled-diffusion analysis to simulate the phenomenon of electromigration of atoms or vacancies in metallic interconnects.

2.2.5. System Coupling

Negative Heat Transfer Coefficient for System Coupling When using [system coupling](#) for one-way or two-way thermal analysis, if near-wall temperatures (TBULK) are transferred from the CFD code to Mechanical, a negative heat transfer coefficient (HCOE) can be transferred (**SCOPT**).

2.2.6. Low Frequency Electromagnetics

The magnetic vector potential (MVP) formulation for 3-D models (used by the SOLID97 element) has been undocumented. Use the 3-D Magnetic Scalar Potential (MSP) or the edge-based formulations instead.

2.3. Solvers

Release 17.0 includes the following new enhancements that improve solution procedures and features:

[2.3.1. Sparse Solver Enhancements](#)

[2.3.2. PCG Solver Enhancements](#)

[2.3.3. Distributed ANSYS Enhancements](#)

[2.3.4. GPU Acceleration Enhancements](#)

2.3.1. Sparse Solver Enhancements

The performance of the shared memory sparse solver (**EQSLV**,SPARSE) has been enhanced. For analyses that include unsymmetric matrices, the solver performance can be up to 60 percent faster than the previous release regardless of the processor hardware used.

The scalability of the sparse solver has been significantly improved when used in Distributed ANSYS. For solutions that use eight or more cores, the solver performance can be up to 50 percent faster than in the previous release.

2.3.2. PCG Solver Enhancements

The PCG solver now supports unsymmetric matrices when solving single-field thermal analyses. Both static and full transient analyses are supported in this situation. Also, the PCG solver now supports thermal transient analyses involving the Quasi method (**THOPT**,QUASI).

2.3.3. Distributed ANSYS Enhancements

The following enhancements are available for Distributed ANSYS:

- The Block Lanczos eigensolver now runs in a fully distributed fashion in Distributed ANSYS, leading to significantly faster performance and scalability.
- Support for spectrum analysis (**ANTYPE**,SPECTR) in a distributed solution has been added.
- The overall scalability of the program has been significantly improved for some models, particularly at higher core counts (> 128 cores).
- Intel MPI is now the default MPI software for Distributed ANSYS.

2.3.4. GPU Acceleration Enhancements

The following enhancements are available for the [GPU Accelerator capability](#).

- The [NVIDIA GPU](#) driver requirements have been updated. Refer to your **ANSYS, Inc. Installation Guide** for platform-specific driver versions.
- Support has been added for the NVIDIA Quadro K5200 and NVIDIA Quadro M6000 GPU cards.
- The [Intel Xeon Phi](#) driver requirements have been updated. Refer to your **ANSYS, Inc. Installation Guide** for platform-specific driver versions.

2.4. Commands

This section describes changes to commands at Release 17.0.

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](#)

[2.4.2. Modified Commands](#)

[2.4.3. Undocumented Commands](#)

2.4.1. New Commands

The following new commands are available:

- **APORT** -- Specifies input data for plane wave and acoustic duct ports.
- **ICROTATE** -- Specifies initial velocity at nodes as a sum of rotation about an axis and translation.
- **ECPCHG** -- Optimizes degree-of-freedom usage in a coupled acoustic model.
- **/FCOMP** -- Specifies file compression level.
- **LANBOPTION** -- Specifies Block Lanczos eigensolver options.
- ***MERGE** -- Merges two dense matrices or vectors into one.
- **MODDIR** -- Activates the remote read-only modal files usage.
- **PSYS** -- Sets the PML element coordinate system attribute pointer.
- ***REMOVE** -- Suppresses rows or columns of a dense matrix or a vector.
- **SCOPT** -- Specifies System Coupling options.
- **SSOPT** -- Defines a solution option for soil analysis (**ANTYPE**,SOIL).

2.4.2. Modified Commands

The following commands have been enhanced or otherwise modified:

- **ANSOL** -- Specifies averaged nodal data to be stored from the results file in the solution coordinate system. The command can now account for the void volume ratio and pore pressure for [coupled pore-pressure mechanical solid \(CPT\) elements](#).
- **ANTYPE** -- Specifies the analysis type and restart status. A new soil analysis type (SOIL) has been added. A new *PRELP* argument is available for acoustic-structural linear perturbation analyses.

- **BF** -- Defines a nodal body force load. New load labels available for acoustic analysis include FPBC (Floquet periodic boundary condition) and PORT (interior port). The static pressure load label for acoustics has been changed to SPRE. A new transport velocity load label (VELO) is available for diffusion analysis.
- **CGROW** -- Defines crack-growth information. The command now supports [fatigue crack growth](#) analysis.
- **CNCHECK** -- Provides and/or adjusts the initial status of contact pairs. A new MORPH option activates mesh morphing during an initial adjustment of contact nodes toward the target surface.
- ***COMP** -- Compresses a matrix using a specified algorithm. The command can now be used to compress sparse matrices based on a threshold value that is relative to the maximum term in the matrix.
- **D** -- Defines degree-of-freedom constraints at nodes. The command now supports pore fluid labels for pressure (PRES) and displacements (UX, UY, UZ).
- **EINFIN** -- Generates structural infinite elements from selected nodes. The command has new options to support the enhanced capabilities of the [INFIN257](#) structural infinite solid element, including arguments for calculating the pole positions and specifying the material properties of the structural elements.
- **EMTGEN** -- Generates a set of [TRANS126](#) elements. The *FKN* argument now accepts a negative value, which is interpreted as the modulus of elasticity and used in determining the contact stiffness.
- **ESOL** -- Specifies element data to be stored from the results file. A new SEND option (ENTO) enables total strain energy density output. The command can now account for the void volume ratio and pore pressure for [coupled pore-pressure mechanical solid \(CPT\) elements](#).
- **ETABLE** -- Fills a table of element values for further processing. A new SEND option (ENTO) supports total strain energy density output. The command can now account for the void volume ratio and pore pressure for [coupled pore-pressure mechanical solid \(CPT\) elements](#).
- **F** -- Specifies force loads at nodes. Fluid flow (FLOW) is now positive when flow is out of the nodes, and negative when flow is into the nodes.
- ***GET** -- Retrieves a value and stores it as a scalar parameter or part of an array parameter. The command now retrieves values (via *Entity* = CINT) from [XFEM](#) stationary-crack or crack-growth analyses, and [fatigue crack growth](#) analyses. Additional acoustic analysis quantities (*Entity* = ACUS) are also available. The command can now retrieve the void volume ratio and pore pressure for [coupled pore-pressure mechanical solid \(CPT\) elements](#) (via *Entity* = NODE).
- **GCDEF** -- Defines interface interactions between general contact surfaces. Enhancements to this command include: additional input options for specifying section numbers of general contact surfaces; compression of the **GCDEF** table output; the ability to specify section ID ranges for listing general contact definitions; and a new option to list the actual contact interactions used during solution.
- **GCGEN** -- Creates contact elements for general contact. The command now creates [CONTA177](#) 3-D line contact elements on 3-D beam elements in support of beam-to-beam and beam-to-surface contact in a general contact definition.
- **HFANG** -- Defines or displays spatial angles of a spherical radiation surface for sound radiation parameter calculations. The default values for the ending angles (*PHI2* and *THETA2*) have changed.
- **IC** -- Specifies initial conditions at nodes. The command can now specify initial accelerations.
- **ICLIST** -- Lists the initial conditions. The command now supports initial accelerations.

- ***INIT** -- Initializes a vector or dense matrix. New complex conjugate (CONJ) and filter (FILTER) options are available.
- **KEYOPT** -- Sets element key options. New input options on the *ITYPE* field make it easier to define key option settings for contact elements used in a general contact definition.
- **LDREAD** -- Reads results from the results file and applies them as loads. This command can now be used to apply temperature output from layered thermal solid elements ([SOLID278](#), [SOLID279](#)) as body loads on layered structural solid elements.
- **MODOPT** -- Specifies modal analysis options. The new *FREQMOD* argument specifies a working frequency for use with the Floquet periodic boundary condition that is applied to acoustic elements in a modal analysis.
- **MSOLVE** -- Starts multiple solutions for an acoustic analysis. New options on this command support a plane wave angle sweep when the Floquet periodic boundary condition is present.
- **NLADAPTIVE** -- Defines the criteria under which a mesh is refined or modified during a nonlinear solution. The command now supports 2-D remeshing.
- **NLMESH** -- Controls remeshing in nonlinear adaptivity. The command now supports 2-D remeshing.
- **NUMMRG** -- Merges coincident or equivalently defined items. The command now supports section types.
- **PLESOL** -- Displays the solution results as discontinuous element contours. A new SEND option (ENTO) supports total strain energy density output. Acoustic potential energy (MENE) and kinetic energy (KENE) are now available for acoustic analyses. The command can now account for the void volume ratio and pore pressure for [coupled pore-pressure mechanical solid \(CPT\) elements](#)
- **PLFAR** -- Plots pressure far fields and far field parameters. The new PLAT label allows you to plot acoustic parameters radiated by a vibrating structure.
- **PLNSOL** -- Displays results as continuous contours. A new SEND option (ENTO) supports total strain energy density output. The command can now account for the void volume ratio and pore pressure for [coupled pore-pressure mechanical solid \(CPT\) elements](#)
- **PMLOPT** -- Defines perfectly matched layers (PMLs) for acoustic, structural, and piezoelectric analyses. The PML element coordinate system input on this command is now specified by the new **PSYS** command instead of the **ESYS** command.
- **PRAS** -- Calculates specified acoustic quantities. The command supports additional acoustic quantities and offers new options for performing the calculations.
- **PRCINT** -- Lists the fracture-parameter (**CINT**) results data. The command now supports [fatigue crack growth](#) analysis.
- **PRESOL** -- Prints the solution results for elements. A new SEND option (ENTO) supports total strain energy density output. Acoustic potential energy (MENE) and kinetic energy (KENE) are now available for acoustic analyses. The command can now account for the void volume ratio and pore pressure for [coupled pore-pressure mechanical solid \(CPT\) elements](#)
- **PRFAR** -- Prints pressure far fields and far field parameters. The new PLAT label allows you to print acoustic parameters radiated by a vibrating structure.

- **PRNSOL** -- Prints nodal solution results. A new SEND option (ENTO) supports total strain energy density output. The command can now account for the void volume ratio and pore pressure for [coupled pore-pressure mechanical solid \(CPT\) elements](#)
- **SECDATA** -- Describes the geometry of a section. The command now supports an equivalent beam radius for use in a general contact definition.
- **SECFUNCTION** -- Specifies shell section thickness as a tabular function. The *KCN* argument (for specifying the local coordinate system reference number) has been replaced by *PATTERN* (for specifying an interpretation pattern for array parameters). The function of the former *KCN* argument is now handled exclusively via the *CSYSID* argument on the ***DIM** command.
- **SECTYPE** -- Associates section type information with a section ID number. For the CONTACT section type, the new RADIUS subtype enables specification of an equivalent beam radius or rigid target radii for use in a general contact definition. In addition, new input options for the *SECID* field make it easier to identify contact elements that are used in a general contact definition.
- **SF** -- Specifies surface loads on nodes. The equivalent source surface flag (MXWF) can now be applied on the surface of structural elements to represent a vibrating panel in an acoustic analysis.
- ***SMAT** -- Creates a sparse matrix. When using the copy operation to create a new sparse matrix from an existing one, you can now extract a submatrix by specifying rows and columns to copy from the original sparse matrix.
- **TB** -- Defines a material data table. The following material data types have been added: **DENS** (mass density), **JROCK** (jointed rock), **MC** (Mohr-Coulomb), **MIGR** (migration model), **PELAS** (porous elasticity), **SOIL** (soil materials). Also, the following material data tables have been enhanced: **CONC**, **SDAMP** (material-damping coefficient), and **CGCR** (crack-growth fracture criterion) now supports [fatigue crack growth](#) analysis. The friction specification (**FRIC**) has been enhanced; when orthotropic friction is a function of sliding distance or sliding velocity, the total magnitude of sliding distance or sliding velocity can now be used to interpolate the friction coefficients (*TBOPT* = EORTHO).
- **TBFIELD** -- Defines field-variable values for material data tables. Mass density (**DENS**) is now available as a supported material data type.
- **XFDATA** -- Defines a crack in the model by specifying nodal level set values. The command now supports the [singularity-based eXtended Finite Element Method](#) (XFEM) approach.
- **XFENRICH** -- Defines parameters associated with crack propagation using XFEM. As with **XFDATA**, this command now supports the singularity-based XFEM analysis approach.

2.4.3. Undocumented Commands

The following commands have been undocumented:

Command	Reason
ADAPT	Adaptive meshing based on the ADAPT macro has been removed.
SEGEN	The automatic superelement generation procedure has been removed.
MADAPT	Adaptive meshing based on the MADAPT macro has been removed.

Command	Reason
FMAGBC	Used with PLANE53 and SOLID97, which have been undocumented.
FMAGSUM	
FOR2D	
HMAGSOLV	
LMATRIX	
TORQ2D	
TORQC2D	
TORQSUM	
NOORDER	Frontal-based ordering has been removed.
WAVES	
WERASE	
WFRONT	
WMID	
WMORE	
WSTART	
WSORT	
PDANL	Probabilistic Design (PDS) has been removed.
PDCDF	
PDCFLO	
PDCLR	
PDCMAT	
PDCORR	
PDDMCS	
PDDOEL	
PDEXE	
PDHIST	
PDINQR	
PDLHS	
PDMETH	
PDPINV	
PDPLT	
PDPROB	
PDRESU	
PDROPT	
/PDS	
PDSAVE	
PDSCAT	
PDSENS	

Command	Reason
PDSHIS	
PDUSER	
PDVAR	
PDWRITE	
RSFIT	
RSPLIT	
RSPRNT	
RSSIMS	

In addition, the following GET functions are obsolete due to [preprocessing enhancements for general contact](#) (p. 17) and have been undocumented: ELMTOSEC, ELMTOTYP, NDTOSEC, NDTOTYP, CMTOSEC, and CMTOTYP.

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 17.0.

Some elements are not available from within the GUI. For a list of those elements, see [GUI-Inaccessible Elements](#).

2.5.1. New Elements

2.5.2. Modified Elements

2.5.3. Undocumented Elements

2.5.1. New Elements

The following new element has been added:

- [SURF155](#) -- A 3-D thermal effect element for 3-D thermal analyses. The element can be used for various load and surface-effect applications. It can be overlaid onto an edge of any 3-D thermal solid element. Various loads and surface effects can exist simultaneously.

2.5.2. Modified Elements

The following elements have been enhanced:

- [COMBIN14](#) -- This spring-damper element now supports tabular real constants as a function of time.
- [COMBI214](#) -- This 2-D spring-damper bearing has new element technology (KEYOPT(1) > 0) which allows it to act as a cylindrical journal bearing or squeeze film damper.
- [CONTA171](#), [CONTA172](#), [CONTA173](#), [CONTA174](#), [CONTA175](#) -- These surface-to-surface and node-to-surface contact elements now support coupled pore fluid-mechanical-thermal interactions and can be used with the coupled pore-pressure mechanical solid (**CPT**) elements. For more information, see [Modeling Pore Fluid Flow at the Contact Interface](#) in the [Contact Technology Guide](#).
- [CONTA173](#), [CONTA174](#) -- These 3-D surface-to-surface contact elements can now model thermal contact at the top and bottom surfaces of 3-D thermal shell elements ([SHELL131](#) and [SHELL132](#)). A new option (KEY-

OPT(13)) controls which temperature degrees of freedom (TEMP,TBOT,TTOP) are activated at the contact and target surfaces.

- [CONTA177](#) -- This 3-D line-to-surface contact element can now model beam-to-beam and edge-to-edge contact for nearly parallel and crossing beams, similar to the [CONTA176](#) element.
- [CPT212](#), [CPT213](#), [CPT215](#), [CPT216](#), [CPT217](#) -- These coupled pore-pressure mechanical solid elements can now output the void volume ratio and pore pressure (PMSV:VRAT,PPRE).
- [FLUID30](#), [FLUID220](#), [FLUID221](#) -- These 3-D acoustic elements now support incompressible fluid with an equivalent infinite sound speed (KEYOPT(6) = 1). In addition, the Floquet periodic boundary condition (**BF**,,FPBC) can be applied in harmonic and modal analyses.
- [HSFLD241](#), [HSFLD242](#) -- These hydrostatic fluid elements now allow for a change in fluid density, keeping the mass constant for incompressible fluids under thermal expansion (KEYOPT(6) = 2).
- [CIRCU124](#), For this general circuit element, KEYOPT(1)=5, 6, and 7 have been undocumented. They were used with the PLANE53 and SOLID97 elements, which have also been undocumented.
- [INFIN110](#) -- This 2-D infinite solid element has a new real constant:

THK -- Thickness (plane) or fraction of the 360° basis (axisymmetric)

Prior to this release, the element required no real constants. See [Elements with New Real Constants](#) (p. 34).
- [INFIN111](#) -- KEYOPT(1) = 1 has been undocumented for this 3-D infinite solid element (because low-frequency electromagnetics has also been undocumented).
- [INFIN257](#) -- This structural infinite solid element now supports static, transient, and harmonic analyses. [SOLID285](#) is now available as a base element. In a static analysis, base elements can now have anisotropic material properties.
- [PLANE223](#), [SOLID226](#), [SOLID227](#) -- These coupled-field elements have several enhancements related to coupled-diffusion analysis, including a new coupled electric-diffusion capability, enhanced structural-diffusion and thermo-diffusion capabilities, and support for the new migration model (**TB**,MIGR). See [Coupled-Field](#) (p. 24).
- [PLANE238](#), [SOLID239](#), [SOLID240](#) -- These diffusion solid elements now account for transport effects from a velocity field, with the transport velocity input as a body load (**BF**,,VELO). A new option (KEYOPT(2)) controls how transport effects are implemented.
- [SHELL181](#), [SHELL208](#), [SHELL209](#), [SHELL281](#), [ELBOW290](#) -- These shell or shell-component elements have a new option for obtaining shell stress output (KEYOPT(10)). The option outputs normal stress component Sz, independently recovered during the element solution output from the applied pressure load.
- [SURF156](#) -- This 3-D structural surface line-load-effect element has three new real constants:

EFSY - Foundation stiffness in the element y direction
EFSZ - Foundation stiffness in the element z direction
ADDMAS - Added mass per unit length

Prior to this release, the element required no real constants. See [Elements with New Real Constants](#) (p. 34).

2.5.3. Undocumented Elements

The following elements have been undocumented:

Element	Reason
INFIN9	Legacy element. Use INFIN110 instead.
SHELL28	Legacy element. Use SHELL181 (KEYOPT(1) = 1) with an orthotropic material instead.
SHELL41	Legacy element. Use SHELL181 (KEYOPT(1) = 1 or KEYOPT(3) = 2) instead.
PLANE53	Legacy element. Use PLANE233 instead.
SOLID97	3-D MVP formulation has been undocumented.
CIRCU125	Obsolete element.

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

2.6. Other

This section contains information about enhancements for this release not discussed elsewhere in this document.

2.6.1. Results File Compression

The results file can now be compressed while it is being written, if desired. Issue the [/FCOMP](#) command to request file compression.

2.7. Documentation

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

2.7.1. Technology Demonstration Guide

The following example problems have been added to the [Technology Demonstration Guide](#):

[2.7.1.1. Forced Response Analysis of a Mistuned Bladed Disk](#)

[2.7.1.2. C*-integral Evaluation for 3-D Surface Flaws](#)

[2.7.1.3. Wire Crimping Modeled with General Contact](#)

2.7.1.1. Forced Response Analysis of a Mistuned Bladed Disk

A free-vibration and forced-response analyses of a tuned and mistuned NASA Rotor 67 fan using [cyclic modeling methods](#) and [linear perturbation](#) solution approaches. The problem includes a modal analysis, a prestressed modal analysis using linear perturbation, and a prestressed mode-superposition harmonic analysis using linear perturbation.

2.7.1.2. C*-integral Evaluation for 3-D Surface Flaws

Evaluates the [C*-integral](#) for cracks in structural components. C*-integral characterizes the crack tip conditions in homogenous materials undergoing a secondary (steady-state) creeping deformation.

Analyses of a simple semicircular surface flaw in a rectangular block and a warped flaw along a tubular joint are discussed.

2.7.1.3. Wire Crimping Modeled with General Contact

Contact modeling via the [general contact](#) method. A multi-strand wire is joined to an electrical terminal (connector) through a mechanical-deformation process known as crimping. General contact is used to automatically create all contact surfaces required in this crimp joint model, with minimal user input. The general contact procedure is especially useful when a large number of contacting surfaces are involved in the model and the geometry makes it difficult to determine contact pairs.

2.7.2. Feature Archive

Legacy features, commands, elements, and theory information continue to move to the [Feature Archive](#). While ANSYS, Inc. intends to support legacy capabilities for the immediate future, some may be undocumented in future releases. Consider moving to their recommended replacements.

2.7.3. Documentation Updates for Programmers

This section contains information about product updates of interest to programmers:

2.7.3.1. 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.

2.8. Known Incompatibilities

The following incompatibilities with prior releases are known to exist at Release 17.0:

- [2.8.1. Elements with New Real Constants](#)
- [2.8.2. PIPE288/PIPE289 Output](#)
- [2.8.3. COMBI214 in Nonlinear Transient Analyses](#)
- [2.8.4. Jobname.ESAV File Required for SPRS and DDAM Spectrum Analyses](#)
- [2.8.5. Tangential Contact Stiffness for Bonded and Rough Contact](#)
- [2.8.6. Debonding with Friction](#)
- [2.8.7. Gurson Plasticity Model with Chaboche Kinematic Hardening](#)
- [2.8.8. Release Number Removed from File Names](#)
- [2.8.9. Changes in Format of ASCII View Factor File](#)
- [2.8.10. XFEM Analysis Database Format Change](#)
- [2.8.11. Windows 10](#)

2.8.1. Elements with New Real Constants

The following elements have been enhanced with new real constants where none had existed before:

- [INFIN110](#)
- [SURF156](#)

Prior to this release, these elements could be used without regard to the real constant set number. (Because the elements did not require real constants, the current real constant set was simply ignored.) Now, real constants must be defined (**R**) for these elements and a real constant set ID must be specified (**REAL**).

2.8.2. PIPE288/PIPE289 Output

The **PIPE288** and **PIPE289** elements no longer output the maximum hoop stress (MAX HOOP STRESS). If this value is needed, issue a ***GET,,SECR,ELEMID,S,Y,MAX** command.

2.8.3. COMBI214 in Nonlinear Transient Analyses

When using element **COMBI214** in a nonlinear transient analysis, the orientation node is no longer requested and nodes can be coincident. The stretches calculation is now based on the displacements differences just as in the linear case.

2.8.4. Jobname.ESAV File Required for SPRS and DDAM Spectrum Analyses

When performing a single-point response-spectrum (SPRS) analysis or a dynamic-design-analysis-method (DDAM) spectrum analysis, the element saved-data file (Jobname.ESAV) must now be available. Prior to this release, the .ESAV file was not required for these analysis types.

2.8.5. Tangential Contact Stiffness for Bonded and Rough Contact

The algorithm for updating tangential contact stiffness at each iteration has been changed for bonded and rough contact. If you request contact-stiffness updating (KEYOPT(10) = 2 on the contact element or the **CNCHECK,AUTO** command), tangential contact stiffness may be higher than in previous releases.

2.8.6. Debonding with Friction

When friction is defined between contact surfaces undergoing debonding, the tangential stress is now calculated as the maximum between the tangential stress as governed by the debonding model, and the tangential stress as governed by the friction law. In prior releases, any friction defined at the debonding interface was ignored until the debonding was complete.

2.8.7. Gurson Plasticity Model with Chaboche Kinematic Hardening

The hydrostatic portion of the Gurson-Chaboche model's yield surface has been modified. In prior releases, Gurson-Chaboche was based on the work of Arndt et al. [18]. Now, the model more closely adheres to the M hlich and Brocks [22] model, and the constitutive equations are more consistent with each other. For more information, see [Gurson in the Mechanical APDL Material Reference](#) and [Gurson's Model in the Mechanical APDL Theory Reference](#).

2.8.8. Release Number Removed from File Names

The following file names previously contained the release number as a part of the file name, but now do not: start.ans, stop.ans, config.ans, menulist.ans, and start.dsp.

2.8.9. Changes in Format of ASCII View Factor File

The ASCII view factor file (**VFOPT**) has been changed to accommodate larger model sizes. Integer formats have changed from I4 to I8 for quantities such as the element number, enclosure number, and so on. The floating point data format remains unchanged.

2.8.10. XFEM Analysis Database Format Change

For XFEM analyses using the [phantom-node method](#), the database structure used in prior release differs from the structure used in this release, making databases incompatible between releases for this type of analysis.

2.8.11. Windows 10

Mechanical APDL has not yet been certified for Windows 10. Certain graphical user interface (GUI) elements may not work properly when running the program on the new operating system.

Chapter 3: Autodyn Release Notes

The ANSYS Autodyn product comprises all of the following explicit solvers: FE (Lagrange), Euler, FCT, ALE, and SPH, and various means to couple them together. All are integrated into the Autodyn Component system, while the FE (Lagrange) and Euler—including Euler-Lagrange coupling—are also integrated into the Explicit Dynamics Analysis system (see [Explicit Dynamics Enhancements \(p. 12\)](#)). Both systems are part of the Autodyn product.

3.1. New Features and Enhancements

The following new features and enhancements are available in release 17.0. Refer to the product specific documentation for full details.

- The 2D multi-material Euler solver has been optimized and now runs faster. For large size meshes (> 1 million elements) a typical speedup of 50% or more has been achieved.
- [Dimensionally Reduced](#) rigid body behavior is supported for the Explicit Dynamics Analysis system in Workbench, where rigid body meshing simplifies the representation of a rigid model by reducing it to the contact regions and centroid of the rigid body. Dimensionally Reduced rigid bodies can be transferred into the Autodyn component system.
- The existing maximum limit of 200 boundary conditions (loads and supports) that can be defined in a model has been removed. An unlimited number of boundary condition definitions can be used.
- The existing maximum limit of 200 initial conditions that can be defined in a model has been removed. An unlimited number of initial condition definitions can be used.
- The [nodal shell thickness](#) is now taken into account in the contact analysis and allows a more accurate shell to shell contact.
- The usability of the Zerilli-Armstrong material strength model has been significantly improved by the implementation of an implicit strain rate correction that will reduce high frequency oscillations that are sometimes observed in the yield surface under high strain rates.

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 17.0. Refer to the product specific documentation for full details of the new features.

[4.1. Aqwa Solver Modules](#)

[4.2. Hydrodynamic Analysis Systems](#)

4.1. Aqwa Solver Modules

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

4.1.1. Global Sloped Seabed

A [globally sloped seabed](#) can now be defined in Aqwa analyses. For more information on how to include this in an Aqwa Workbench analysis, see [Configuring the Geometry](#).

4.1.2. Nonlinear Roll Damping

[Nonlinear roll damping](#) has been extended to Aqwa-Naut and Aqwa-Fer. Nonlinear bilge keel roll damping can be included using an ITTC recommended procedure. A coefficient can be input to define quadratic roll damping about a specified axis.

For information on how to include these effects in Aqwa Workbench, see [Define Parts Behavior](#).

4.1.3. Seabed Friction on Dynamic Cables

[Seabed inline and lateral friction coefficients](#) on dynamic cables can be included in Time Response analyses.

4.1.4. Structure-Structure Cables

Cables between structures can now be configured so that they do not pass through the seabed.

4.2. Hydrodynamic Analysis Systems

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

4.2.1. Additional Nodal Information

In addition to the motions of the structure's center of gravity (CoG), the Aqwa Workbench editor now allows you to output:

- In a Hydrodynamic Diffraction analysis: nodal motion response amplitude operators (RAOs) in the fixed reference axes (FRA) or relative to the wave surface

- In a Hydrodynamic Time Response analysis: absolute nodal motions in the FRA or relative to the wave surface, or the relative motions between nodes on different structures
- In a Hydrodynamic Frequency Statistical analysis: significant nodal motions in the FRA, in the local structure axes (LSA), or relative to the wave surface

4.2.2. Extended Stability Analysis Capabilities

System natural frequencies and mode shapes can now be calculated and reported, including animation. Two Stability Analysis systems may now be linked to allow the second to use the results of the first as a starting condition.

4.2.3. Spreading Function and Carpet Spectrum

The new [Wave Spreading](#) option allows you to define a multidirectional wave spectrum. Additionally, you may set this option to Manual Definition in order to specify a carpet spectrum.

Chapter 5: Beamcheck Release Notes

There are no new features or enhancements in ANSYS Beamcheck in Release 17.0.

Chapter 6: Fatjack Release Notes

There are no new features or enhancements in ANSYS Fatjack in Release 17.0.

Chapter 7: ANSYS Composite PrepPost (ACP)

The following enhancements are available in release 17.0. Refer to the product specific documentation for full details of the new features. Notes on features incorporated into the 16.1 and 16.2 releases are also included here for users upgrading between major releases.

- [7.1. New Features in ANSYS Composite PrepPost \(ACP\) 17.0](#)
- [7.2. New Features in ANSYS Composite PrepPost \(ACP\) 16.2](#)
- [7.3. New Features in ANSYS Composite PrepPost \(ACP\) 16.1](#)
- [7.4. Supported Platforms for ANSYS Composite PrepPost \(ACP\) 17.0](#)
- [7.5. Known Incompatibilities](#)

7.1. New Features in ANSYS Composite PrepPost (ACP) 17.0

The following features were added to ANSYS Composite PrepPost (ACP) for the 17.0 release:

- [7.1.1. Variable Material Data](#)
- [7.1.2. Enhanced Composite Workflows](#)
- [7.1.3. Interactive Lay-up Definition in Excel](#)
- [7.1.4. Script-Based Definition of User-Defined Plots](#)
- [7.1.5. Enhancement of Expression Parameters](#)
- [7.1.6. New Sandwich Failure Criteria](#)

7.1.1. Variable Material Data

The temperature-, shear-, and degradation factor-dependent material data functionality has been extended. In release 17.0, the composite material elasticity and strength properties can depend on [any scalar user-defined field](#). This allows you to account for not only basic effects like temperature or ply-shearing due to draping on the composite behavior of the simulation, but any other factor such as void content, degree of curing, or humidity.

7.1.2. Enhanced Composite Workflows

Composite workflows for shell and solid element analyses have been streamlined. Multiple ACP-Pre systems and Mechanical Models can now be assembled in one analysis system. In this way, sub-structures can be meshed and connected independently. Composite shells and composite solids follow the same workflow logic and composite shell models can now be combined with composite solid models. For more information, see [Workbench Workflow Examples](#).

7.1.3. Interactive Lay-up Definition in Excel

A new Excel Link interface allows for efficient, live, spreadsheet-based definition and data exchange on composite lay-ups in ACP. For more information, see [Excel Spreadsheet Interaction](#).

7.1.4. Script-Based Definition of User-Defined Plots

Definition of user-defined plots can now be defined entirely in a Python script contained in the plot properties. For more information, see [User-Defined Plot](#).

7.1.5. Enhancement of Expression Parameters

Parameters can now be published based on [multi-line script expressions](#). Previously, the expression output parameters were limited to a single-line Python expression.

7.1.6. New Sandwich Failure Criteria

The library of failure criteria was extended by added a shear crimping failure criteria for sandwich structures. For more information, see [Shear Crimping Failure](#).

7.2. New Features in ANSYS Composite PrepPost (ACP) 16.2

The following features were added to ANSYS Composite PrepPost (ACP) for the 16.2 release:

- [7.2.1. CAD Selection Rule](#)
- [7.2.2. Surface Section Cuts and BECAS Interface](#)
- [7.2.3. Control Drop-Off Elements on Top and Bottom Surface](#)
- [7.2.4. Reordering Extrusion Guides in Solid Models](#)
- [7.2.5. User-Defined Plots](#)
- [7.2.6. Draping and Angle Plots](#)
- [7.2.7. Composite Failure Criteria](#)

7.2.1. CAD Selection Rule

The new [CAD Selection Rule](#) allows you to directly define ply geometries on loaded CAD surfaces and solids. Capture tolerances make this a very powerful feature, allowing projection of ply definitions from a flat surface to a curved part. This feature also allows you to easily scale an arbitrary ply geometry.

7.2.2. Surface Section Cuts and BECAS Interface

A 2D mesh surface of the lay-up at a section cut can now be created in an ACP shell model. The section cut feature has been extended so that the section cut surface and its [lay-up information can be exported](#) to Mechanical APDL or to the cross section analysis tool [BECAS](#). This feature allows you to derive beam models from an ACP shell model.

7.2.3. Control Drop-Off Elements on Top and Bottom Surface

New options in ACP solid modeling allow the [disabling of drop-off elements](#) on the top and/or bottom surface for selected element sets.

7.2.4. Reordering Extrusion Guides in Solid Models

[Extrusion guides](#) defined for an ACP solid model can now be reordered easily, this allowing finer control of solid model extrusion.

7.2.5. User-Defined Plots

Plots can now be created for arbitrary [user-defined results](#) based on Python expressions. This allows for visualization of scalar user-specific results.

7.2.6. Draping and Angle Plots

Capabilities to visualize draping results and ply angles have been enhanced. The [Angle Plot](#) can now display contours for:

- Design angle
- Draped fiber angles
- Draped shear angles
- Draped transverse shear angles

The [Draping Mesh Plot](#) is now a regular plot object within Lay-up Plots and can allow you to visualize the draping mesh or the flatwrap.

7.2.7. Composite Failure Criteria

The [Tsai-Hill Criterion](#) for woven fabrics has been amended to distinguish between UD and woven fabrics.

7.3. New Features in ANSYS Composite PrepPost (ACP) 16.1

The following features were added to ANSYS Composite PrepPost (ACP) for the 16.1 release:

[7.3.1. Performance Improvements](#)

[7.3.2. Solid Model Sensor](#)

[7.3.3. GUI Improvements](#)

[7.3.4. Additional Flat Wrap Geometry Formats](#)

7.3.1. Performance Improvements

The following enhancements to ACP performance have been made:

- The time required to open and save/close large ACP models has been significantly reduced.
- The caching of geometry operations (CAD geometry-based thickness definition) has lead to significant performance improvements when working with imported CAD geometries.
- Solid model update has been further parallelized which, together with the cached geometry operations, results in faster solid model generation.

7.3.2. Solid Model Sensor

The [sensor](#) feature has been extended to calculate the mass properties of solid models. This leads to accurate calculation of mass and center of gravity of solid models after the application of geometry options (for example, [Cut-off Geometry](#)).

7.3.3. GUI Improvements

[New icons](#) indicate the update status of locked and inactive objects. These icons convey more information about the status of an object than previous versions.

7.3.4. Additional Flat Wrap Geometry Formats

Flat wrap geometries can now be exported in the .IGES and .STP file formats, in addition to the .DXF format. For more information, see [Production Ply](#).

7.4. Supported Platforms for ANSYS Composite PrepPost (ACP) 17.0

Platform/OS levels that are supported in the current release are posted on the [ANSYS website](#).

7.5. Known Incompatibilities

There are no known incompatibilities with previous releases for Release 17.0.

Part II: ANSYS Fluids Products

Release notes are available for the following ANSYS Fluids products:

[Fluent \(p. 51\)](#)

[CFX \(p. 65\)](#)

[TurboGrid \(p. 69\)](#)

[ANSYS BladeModeler \(p. 71\)](#)

[CFD-Post \(p. 73\)](#)

[Polyflow \(p. 75\)](#)

[Forte \(p. 77\)](#)

Chapter 1: Fluent Release Notes

The following sections contain release information for ANSYS Fluent 17.0.

- [1.1. New Features in ANSYS Fluent 17.0](#)
- [1.2. Supported Platforms for ANSYS Fluent 17.0](#)
- [1.3. New Limitations in ANSYS Fluent 17.0](#)
- [1.4. Resolved Issues and Limitations](#)
- [1.5. Updates Affecting Code Behavior](#)

1.1. New Features in ANSYS Fluent 17.0

New features available in ANSYS Fluent 17.0 are listed below. Where appropriate, references to the relevant section in the User's Guide are provided.

User Interface

- Functionality previously accessible from the drop-down menus is now accessed from ribbon tabs that correspond to the steps in a typical simulation process, from setup to solving and postprocessing. Especially common commands and settings are also available directly in the ribbon. For additional information on the ribbon, see [The Ribbon in the *Fluent User's Guide*](#).
- Improvements to the user experience:
 - Elements of the interface can be undocked and re-docked, as well as made full screen.
 - The console now visually indicates whether text is an error (red), user entered (blue), or Fluent generated (black).
 - There is an auto-scroll check-box in the lower-right of the console that allows you to toggle whether or not Fluent automatically scrolls to the bottom of the console when new content is printed.
 - The tops of multi-selection lists now display the number of items selected and the total number of items.
 - The axis triad now provides additional options for manipulating the orientation of 3D objects in the graphics window. For more information on this functionality, see [Selecting a View in the *Fluent User's Guide*](#).

For additional information about the user interface, see [Graphical User Interface \(GUI\) in the *Fluent User's Guide*](#).

Documentation

- The tutorials for the adjoint solver module and the battery model module have been moved from the Fluent Advanced Add-On Modules manual to the Fluent Tutorial Guide on the ANSYS Customer Portal, in order to allow them to be updated and made available between releases.

Solver-Numerics

- Specified mass flow rate, mass flux, and average mass flux are now defined according to the selected **Reference Frame** option in boundary conditions. In previous releases these quantities were relative to the adjacent cell zone motion and independent of the selected reference frame.
- It is now possible to specify a time-varying wall rotation or wall translation speed using a transient profile ([Standard Transient Profiles in the Fluent User's Guide](#)) or UDF.
- The Warped-Face Gradient Correction is a new computational enhancement designed to improve the gradient accuracy for all gradient methods: cell based, node based, and least squares based gradients, on meshes that contain poor quality cells. For additional information about warped-face gradient correction, see [Warped-Face Gradient Correction in the Fluent User's Guide](#).
- For the coupled solver, the conservative coarsening option for the algebraic multigrid solver is now enabled by default for improved convergence, especially for cases with native polyhedral meshes (as opposed to converted polyhedra) and/or highly stretched cells. For more details about this option, see [Coarsening Parameters in the Fluent User's Guide](#). In rare cases where this degrades convergence, the **Conservative Coarsening** option may be disabled in the **Multigrid** tab of the **Advanced Solution Controls** dialog box.

Solver-Meshing

- It is now possible to delete flow control zones when solving a contact detection problem. There are also two new text commands for creating and deleting flow control zones under `/define/dynamic-mesh/controls/contact-parameters/flow-control-parameters`.
- The mesh check is now enhanced to evaluate the quality of mapped mesh interfaces. There is also a new text command, `define/mesh-interface/improve-quality`, that runs the same quality check and allows you to increase the mapped interface tolerance to improve the quality of any mapped interfaces that Fluent determines could be improved.
- The six degrees of freedom (6 DOF) solver is now available for steady-state problems.
- You can now specify rigid body motion or mesh deformation on top of rigid body motion using a user-defined function. You can also specify if the deformation or motion is in the relative or absolute reference frame. For additional information, see [Rigid Body Motion](#) and [User-Defined Motion in the Fluent User's Guide](#).
- A rigid-body orientation calculator has been added to the **Dynamic Mesh Zones** dialog box. This calculator converts rotation matrices and Euler angles to the axis and angle format that is required in the **Dynamic Mesh Zones** dialog box. Note that when converting inputs, the calculator also computes those values into the other available input types as well as the required output format. For additional information about this orientation calculator, see [Rigid Body Motion in the Fluent User's Guide](#).
- You can now use overset meshing to decompose complex geometrical problems into a system of geometrically simpler overlapping meshes. For additional information on overset meshing, see [Overset Meshes in the Fluent User's Guide](#).

Models

- Turbulence
 - A new ω -based Reynolds stress model called the stress-BSL model is available. It uses a linear model for the pressure-strain term like the stress-omega model, but differs with regard to the scale equation: it solves the scale equation from the baseline (BSL) $k\text{-}\omega$ model, and thus removes the

free-stream sensitivity observed with the stress-omega model. ([Stress-Omega and Stress-BSL Models](#))

- The Shielded Detached Eddy Simulation (SDES) and Stress-Blended Eddy Simulation (SBES) models have been improved and are now supported as full features. These are hybrid RANS-LES models that have an improved shielding function compared to DDES / IDDES, in order to provide strong shielding of the RANS boundary layer and a fast “transition” between RANS and LES in separating shear layers. ([Shielded Detached Eddy Simulation \(SDES\)](#) and [Stress-Blended Eddy Simulation \(SBES\)](#))
- Combustion and Species Transport
 - CHEMKIN mechanisms with any reaction type supported in CHEMKIN can now be imported in ANSYS Fluent. ([Importing a Volumetric Kinetic Mechanism in CHEMKIN Format](#))
 - In non-premixed, premixed, and partially premixed combustion, the laminar diffusion coefficient with the unity Lewis number assumption is now represented by the ratio of the mixture's laminar thermal conductivity and its specific heat. This approach is similar to the previous formulation of the laminar coefficient as a ratio of the mixture's molecular viscosity and the Schmidt number. However, it yields more accurate results in calculations of the heat transfer near the wall, where laminar diffusion is generally a large fraction of the total diffusion (laminar and turbulent).
 - For non-premixed and partially premixed combustion models, you can now use user-defined functions to specify the variation of scalar dissipation, mean mixture fraction grid, and mean progress variable grid for flamelet generation. ([DEFINE_FLAMELET_PARAMETERS](#))
 - In species transport simulations, you can now model electrochemical reactions on electrode-electrolyte interfaces. ([Electrochemical Reactions](#))
 - For the Partially-Premixed Combustion model with diffusion flamelet generated manifolds, you can now define the normalized progress variable either based on equilibrium calculations or using the solution of the flamelet with lowest strain rate. ([Flamelet Generated Manifold](#))
 - The **Direct Integration** option is now available for the **Relaxation to Chemical Equilibrium** model. If selected, the equilibrium calculations will always be performed at all cells and ISAT tabulation will not be used.
 - For coal combustion simulations with the Species Transport model, you can now define up to three coal streams using the **Coal Calculator** dialog box.
 - For transient simulations, a new chemistry acceleration method, Dynamic Cell Clustering (DCC), is now available with the ANSYS CHEMKIN-CFD Solver. The method is enabled by default. ([Dynamic Cell Clustering with ANSYS CHEMKIN-CFD Solver](#))
 - For the **Species Transport** model, the **Species Model** dialog box has been reorganized. Chemistry solvers are now available in a drop-down list and are grouped with the **Integration Parameters...** button.
- Pollution Formation
 - The Method of Moments model has been extended to include the soot aggregation modeling. ([Soot Aggregation](#))
- Discrete Phase Model
 - For DEM particles, new collision laws are available: Hertzian ([The Hertzian Collision Law](#)), Hertzian-dashpot ([The Hertzian-Dashpot Collision Law](#)), and rolling friction ([Rolling Friction Collision Law for DEM](#)).

- For discrete phase simulations, you can now account for particle rotation. The additional physical phenomenon can be specified per injection ([Particle Rotation](#)). A new lift force arising from particle rotation in fluids, Magnus lift, can be considered for injections with rotating particles. ([Magnus Lift Force](#))
- For vaporizing droplet and multicomponent materials, you can now enable averaging of physical properties between the droplet surface and bulk gas mixture conditions. In addition, for the convection / diffusion-controlled Vaporization model, you can now use the Variable Lewis Number formulation option. The property averaging and Variable Lewis Number formulation will allow you to improve the accuracy of the DPM simulations for a wide range of combustion applications. ([Heat Transfer to the Droplet](#) and [Physical Property Averaging](#))
- A new model, the Rough Wall model, accounts for the influence of the surface roughness on the particle motion. The model is able to better predict particle behavior especially in confined geometries such as cyclones and pipe systems. ([Rough Wall Model](#))
- A wall-film boundary condition can now be modeled using the Kuhnke model. The model considers four impingement regimes (rebound, spread, splash, and dry splash) and takes into account wall conditions (dry or wet) and the influence of surface roughness and film thickness. ([The Kuhnke Model](#))
- Tracking of child droplets within the same time step is now supported as a full feature. ([Including Droplet Breakup](#))
- Volume of Fluid
 - The default formulation for Stokes wave theory is now based on the work by John D. Fenton and is more accurate than the previous default for finite amplitude waves at high wave steepness. For additional information, see [Stokes Wave Theories in the Fluent Theory Guide](#).
 - You can now specify a multi-directional numerical beach, which is useful for suppression of numerical reflections arising from having multiple pressure outlet boundaries. For additional information, see [Numerical Beach Treatment for Open Channels in the Fluent User's Guide](#).
 - There are several improvements to open channel flow.
 - Open channel flow is now compatible with velocity inlets.
 - You can specify segregated velocity inputs for moving obstacles, water, and air.
 - You can specify a buffer layer between water and air.

For additional information regarding open channel flow, see [Modeling Open Channel Flows](#) and [Modeling Open Channel Wave Boundary Conditions in the Fluent User's Guide](#).
- Eulerian Multiphase Model
 - You can now modify the minimum volume fraction in the **Solution Limits** dialog box. For additional information about this feature, see [Setting Solution Limits in the Fluent User's Guide](#).
- Electric Potential Model
 - A new model called Electric Potential is now available for simulating an electric potential field in ANSYS Fluent. ([Modeling Electric Potential Field](#))
- Eulerian Wall Film Model

- Variable wall film density is now supported for mass, momentum, energy, and scalar formulations.

Cell Zones and Boundary Conditions

- For simulations that have boundary conditions that are coupled to GT-POWER (through the **1D Simulation Library** dialog box), Fluent has been upgraded to be compatible with version 7.5 of GT-SUITE.
- It is now possible to define one or more settings at single or multiple boundaries / cell zones of a given type at once using the `define/boundary-conditions/set/<type>` text commands. These commands facilitate quick and efficient setup of cell / boundary conditions for cases with large numbers of zones.

Whereas `define/boundary-conditions/<type>` commands force you to make all specified boundaries / cell zones have exactly the same settings, the new commands allow you to define one setting at a time to be the same across all specified boundaries / cell zones.

- You can now specify the reference frame for backflow temperature, pressure, and flow directions for pressure outlets—you can choose either **Absolute** or **Relative to Adjacent Cell Zone**.

Mesh Morpher/Optimizer

- When using an unstructured control point distribution to define your mesh morphing, it is now possible to specify that an approximate number of control points are created on the mesh nodes of a particular boundary zone or surface, with a distribution that is based on the distribution of the cell faces in that zone. For details, see the instructions related to the **Define Control Points** dialog box in [Setting Up the Mesh Morpher/Optimizer in the Fluent User's Guide](#).

Parallel Processing

- Parallel scalability has been improved for the following cases:
 - cases that have both solid zones and the shell conduction model enabled
 - simulations run on a high number of processors (these improvements have been demonstrated on cases with 830 million cells run on 129,000 processors)
 - moving and deforming meshes
 - volume of fluid (VOF) multiphase simulations with moving and deforming meshes
 - simulations involving species PDF tabulation (for such cases, diagnostic messages will no longer affect performance)
- Automatic mesh partitioning has been improved to allow faster partitioning for cases in which the cells have undergone adaption.
- The METIS partition library has been updated to the latest version, which generally improves the partition quality and may positively affect the convergence. This also supports partitioning large cases: cases with up to two billion cells have been partitioned successfully.
- ANSYS Fluent HDF I/O has been improved, especially with regard to file writing. In addition, it now automatically switches the default HDF I/O mode to NODE0 if the filepath is accessible to compute-node 0. ([Reading and Writing Files Using Hierarchical Data Format \(HDF\)](#))

- The algebraic multigrid solver now automatically reorders the linear system for improved convergence. In rare cases where this degrades convergence, the reordering may be disabled using the Scheme command `(rpsetvar 'reorder/amg 0)`.
- A new model-weighted partitioning method has been added, which creates partitions that seek to balance the overall number of cells along with the time needed for the models and attributes used as weights (DPM, VOF, solid zones, and so on). For DPM- or DEM-intensive cases, this method has shown significant performance improvements. ([Partitioning](#))

Data Import and Export

- When exporting data to CFD-Post, you can now select specific surfaces for results export without exporting the full 3D data field. For additional information, see [Exporting to ANSYS CFD-Post](#) and [ANSYS CFD-Post-Compatible Files in the Fluent User's Guide](#).

Graphics, Postprocessing, and Reporting

- It is now possible to import and export graphics in HSF format. HSF files are highly compressible and can be viewed and interacted with using the “HOOPS Viewer” application available for iOS and Android devices. The “HOOPS Viewer” application can be downloaded to iOS and Android equipped devices from the “App Store” and “Google Play” respectively.
- It is now possible to visualize the quality of any non-conformal interfaces that you have created, by selecting **Mesh** and **Interface Overlap Fraction** for **Contours of** in the **Contours** dialog box.
- A ruler is now available for estimating the scale of the displayed geometry/model in the graphics window. If in 3D, the view must be set to orthographic.
- To improve the quality of contour plots for cases with native polyhedral meshes (as opposed to converted polyhedra), a higher order method is used for interpolating cell values to nodes. To return to previous behavior, use the TUI command `display/set/nodewt-based-interp?`.
- An object called a **Report Definition** has been added to Fluent’s options for capturing values during solver computation. Report definitions, along with **Report Files** and **Report Plots** expand upon the functionality of monitors and reports provided in previous releases. Now you can save multiple report definitions (similar to monitors) to a single file (using **Report Files**) and plot multiple report definitions to the same graphics window (using **Report Plots**). For additional information on this feature, see [Report Definitions in the Fluent User's Guide](#).
- Report definitions are available for use in custom field functions (see [Custom Field Functions in the Fluent User's Guide](#) for more information).

Workbench

- The Fluent transcript file is now available for viewing under the **Solution** tab. You can review the transcript files of multiple design points and multiple component or analysis systems. ([Monitoring Fluent Solutions in Workbench](#))
- A new Fluent reporting capability enables you to quickly review the case settings, convergence history, and simulation results and export these data in an HTML format. ([Generating Fluent Project Reports](#))
- You can now use the **Fluent Solution Images** view to review contour and vector plots generated from the Fluent solution. This feature is available for multiple design points and multiple component or analysis systems. ([Monitoring Fluent Solutions in Workbench](#))

- The **Workbench Tools** toolbar in Fluent has been enhanced with new options to provide a convenient way to perform the basic Workbench commands, such as **Reload**, **Clear Generated Data**, **Reset**, and others, directly from within Fluent.
- Generation of the interpolation file (*.ip) is now enabled by default. Fluent-based systems will, by default, use a previously generated interpolation file (*.ip) (if one exists) as a better initial guess for new simulations when the input mesh file for the **Setup** cell has changed. ([Fluent in the Workbench User's Guide](#))
- A new option, **Reuse Fluent Session for Design Points**, is now available in the **Options** dialog in Workbench. When this option is selected, then during foreground parametric updates, the same Fluent session will be used for successive design point runs. ([Fluent in the Workbench User's Guide](#))

Add-Ons

- Adjoint Module
 - The Design Tool has been enhanced in order to allow you to:
 - rotate, translate, or scale a zone
 - bound the deformation of a zone by an imported surface
 - specify that the design conditions of a zone are strictly enforced for every node of the zone
 - automatically check if any nodes have multiple constraining or deformation conditions that conflict with each other
 - export the modified geometry of 3D cases as an .stl file

For details, see [Defining Conditions for the Deformation](#) and [Shape Modification in the ANSYS Fluent Advanced Add-On Modules](#).

 - The physics of a primary flow that is a compressible ideal gas can now be modeled in the adjoint solver. ([Using the Adjoint Solution Methods Dialog Box in the ANSYS Fluent Advanced Add-On Modules](#))
 - Improvements have been made to the numerics, which should lead to faster convergence for many problems. To receive the full benefit of the improvements, you may need to adjust the adjoint solution control settings compared to those used for previous releases. In particular, for cases in which the adjoint energy equation is enabled it should be possible to use a higher value for the **Energy Equation Scaling** in the **Adjoint Solution Controls** dialog box.
- Battery Module
 - The computationally effective Reduced Order Method for solving MSMD potential equations is now available. ([Reduced Order Solution Method \(ROM\)](#))
 - You can now model and simulate battery thermal runaway processes using ANSYS Fluent semi-empirical One-Equation and Four-Equation thermal abuse models. ([Thermal Abuse Model](#))
- Fuel Cell Modules
 - A new fuel cell model, called the PEM Fuel Cell model, is available for modeling energy conversion processes in fuel cells with micro-porous layers. The advantages of this new PEM Fuel Cell model compared to the

previous PEMFC sub-model of the Fuel Cell and Electrolysis model include the following capabilities and features:

- modeling anode and cathode micro-porous layers that have distinct properties
- solving the capillary pressure equation in porous media (liquid saturation obtained from the capillary pressure can be discontinuous at the material interfaces)
- modeling transport for all three of water's phases (gas, liquid, and dissolved)
- modeling dissolved water transport through the catalyst-membrane-catalyst assembly (the Osmotic drag treatment through the membrane is included in the transport equation)
- solving liquid saturation in gas channels to provide meaningful values of liquid volume fraction for the purpose of modeling the increase of the pressure drop in the presence of liquid water

The modeling capabilities of the PEM Fuel Cell model are described in [PEMFC Model Theory](#) and [Using the PEMFC Model](#).

The PEMFC sub-model in the Fuel Cell and Electrolysis add-on module is being retained in this release. However, it will be removed in future releases.

- For Fuel Cell and Electrolysis and SOFC Fuel Cell modules, a separate license is no longer required.

- Continuous Fiber Module

For the continuous fiber model, the enhancements include:

- the ability to define fiber material properties using UDFs

UDF access is available for viscosity, density, specific heat capacity, thermal conductivity and vapor-liquid equilibrium. ([User-Defined Functions \(UDFs\) for the Continuous Fiber Model](#))

- a new type of fiber injections that enables you to use an ASCII file for entering injection fiber data in a tabular format ([Fiber Injection Types](#))
- additional postprocessing capabilities for displaying injection fibers ([Display of Fiber Locations and Grid Points](#) and [Fiber Display](#))
- the ability to specify that the fiber viscosity depends on the fiber velocity gradient ([Melt Spinning](#))
- the ability to perform fiber simulations in parallel, where the CFD flow is solved on parallel compute nodes and the fibers are computed on the host node ([Running the Fiber Module in Parallel](#))

- Macroscopic Particle Module

The macroscopic particle model (MPM) that has previously been made available as a UDF is now included with Fluent as an add-on module. This module allows modeling the transport of large particles in a fluid flow, particle tracking, and particle-particle and particle-wall interactions. The model provides a special treatment for flow blockage and momentum exchange and a calculation of drag and torque on particles. ([Using the Macroscopic Particle Model in the ANSYS Fluent Advanced Add-On Modules](#))

General

- Support for the Fluent License Manager has been extended to version 17.0, though it will not be supported in version 18.0 and later releases. The purpose of this extension is to provide time for those still using it to migrate and upgrade to the ANSYS, Inc. License Manager.

Fluent as a Server

- It is now possible to return incremental monitor data by specifying filters in the command. See [Fluent as a Server Commands \(fluent .\)](#) and [Instant Commands in the Fluent as a Server User's Guide](#).

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 17.0 Beta Features Manual*, which is available on the [ANSYS Customer Portal](#).

1.2. Supported Platforms for ANSYS Fluent 17.0

Information about past, present, and future operating system and platform support is viewable via the [ANSYS website](#).

1.3. New Limitations in ANSYS Fluent 17.0

The following is a list of new or recently discovered limitations known to exist in ANSYS Fluent 17.0. Where possible, suggested workarounds are provided.

- Meshes
 - At non-conformal interfaces, the **Matching** option is no longer allowed with the **Mapped** option. When opening a case set up in a previous release with both options enabled, you will be prompted to recreate the interface without the **Matching** option.
- Solver-Numerics
 - You should not vary the timestep size during a calculation run when using second-order discretization. Doing so creates an error that reduces with a reduction of the timestep jump.
- Cell Zones and Boundary Conditions
 - Fluid zones designated as 3D fan zones cannot have non-conformal interfaces.
- Parallel Processing
 - Using the Intel MPI option on the Windows 10 platform causes the **mpiexec.exe** process to remain active even after Fluent is closed, since the Intel MPI version is not supported on Windows 10. It is therefore recommend that you use the default MPI (that is, Platform MPI) on Windows 10 instead.
 - If you are using the “PARALLEL INDEPENDENT” or “PARALLEL COLLECTIVE” mode in Windows, it is possible that during HDF5 I/O you may receive a segmentation violation, due to Platform MPI version 9.1.3.1. As a workaround, you can use Intel MPI.
- Graphics, Reporting, and Postprocessing

- The TUI command `solve/set/report-definitions/convert-monitors-to-report-defns` is not compatible with the use of monitors in the Convergence Manager.
- The mouse-annotate feature is no longer available. Annotations can still be created using the Annotate dialog box (see [Annotate Dialog Box in the *Fluent User's Guide*](#) for additional information).
- User-Defined
 - Any scripts or journals that attempt to add menu items to Fluent pull-down menus (which have been replaced with the Fluent ribbon) will no longer work. You must create separate user-defined menus to house all user-defined menu items. For additional information about user-defined menus, see [Adding Menus to the Right of the Ribbon in the *Fluent Customization Manual*](#).
- User-Defined Functions (UDFs)
 - The **Visual Studio Express 2015 for Windows** installer on Windows 10 installs libraries in non-standard locations, resulting in UDF compilation failures on this platform. It is recommended that you instead use the **Visual Studio Express 2015 for Desktop** installer, or manually set the library path based on your local installation (for example, `LIB="C:\Program Files (x86)\Microsoft Visual Studio 14.0\VC\lib\onecore\amd64";%LIB%`).
- Other
 - Fluent does not support non-ASCII characters in the names of files, zones, and boundaries.
- For a complete list of known limitations, including those that exist from previous releases, refer to [Known Limitations in ANSYS Fluent 17.0 in the *Fluent Getting Started Guide*](#).

1.4. Resolved Issues and Limitations

This section lists issues and limitations that have existed in previous releases, but that have been resolved and removed in ANSYS Fluent 17.0.

- Models
 - For simulations that are coupled to GT-POWER (through the **1D Simulation Library** dialog box), the species order from GT-SUITE is no longer permanently locked in the saved Fluent case file. This means that the Fluent species order can later be changed after the Fluent simulation has successfully coupled with GT-SUITE using a particular mapping.
 - DEM particles can now simulate particle rotation.
 - Periodic boundary conditions can now be used with the DEM collision model.
 - The following models and features are now supported with encrypted CHEMKIN gas kinetics mechanisms:
 - the Reactor Network model
 - parallel diesel unsteady flamelet model
 - parallel flamelet generation
 - All reaction types supported in CHEMKIN can now be used in ANSYS Fluent. The previous limitations still apply when using any of the following options:

- density-based solver
- non-stiff chemistry solver
- finite-rate / eddy-dissipation turbulence-chemistry interaction
- When modeling shell conduction on a wall, you can now specify the thickness and heat generation rate of a layer using an input parameter.
- The $k-\epsilon$ model and the Reynolds stress model have been improved to ensure that viscous heating contributions are taken into account by scalable wall functions.
- The DPM Domain option of the hybrid parallel DPM tracking method is now available with non-conformal interfaces.
- When setting up the phase interaction for a multiphase flow, the **population-balance** option is now available for the mass transfer mechanism.
- Boundary Conditions
 - When a thin wall is defined on an interface wall zone created at a mapped interface, the thermal coupling will no longer ignore the thermal resistance due to the specified thickness.
- Mesh
 - ANSYS Fluent can now import CGNS polyhedral mesh and data.
 - A fix has been introduced to ensure that when a mesh is replaced, the data is interpolated to the new mesh.
- Solver
 - A correction has been made for the calculation of the gradient using the Green-Gauss Node Based method, so that it is no longer necessary to use the (`rpsetvar 'recon/node-loop-old? #f`) text command to ensure correct calculations for the following:
 - coupled and/or two-sided walls in multiphase simulations
 - coupled and/or two-sided walls in single phase simulations in which the legacy boundary treatment is enabled (through the `solve/set/nb-gradient-boundary-option?` text command)
 - A fix has been introduced for significantly mismatched solid-solid mapped interfaces in the parallel version of Fluent, to ensure that you do not observe different results (including jumps in the residuals) if you change the number of processes.
- Postprocessing
 - A fix has been introduced so that when you are postprocessing field variables that are only available for walls (such as wall film height, wall strain, and heat flux) for parallel cases, the correct values appear at all partition junctions on the wall if you display node values.
- Parallel
 - When reading or writing case and data files in the Hierarchical Data Format (HDF), the following are now supported:

- stretched non-conformal mesh interfaces
- the ANSYS Icepak network model
- On Windows, it is now possible to read meshes into a parallel ANSYS Fluent session when you use more than 20 million cells per core.

1.5. Updates Affecting Code Behavior

This section contains a comprehensive list of the code changes implemented in ANSYS Fluent 17.0 that may affect the output generated by the previous release.

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

Solver-Numerics

- Specified mass flow rate, mass flux, and average mass flux are now relative to the selected **Reference Frame** option in boundary conditions. In previous releases these quantities were relative to the adjacent cell zone motion and independent of the selected reference frame. This may result in differences in flow quantities (such as velocity, pressure, and temperature) between release 16.2 and release 17.0.
- For the coupled solver, the conservative coarsening option for the algebraic multigrid solver is now enabled by default for improved convergence, especially for cases with native polyhedral meshes (as opposed to converted polyhedra) and/or highly stretched cells. For more details about this option, see [Coarsening Parameters in the Fluent User's Guide](#). In rare cases where this degrades convergence, the **Conservative Coarsening** option may be disabled in the **Multigrid** tab of the **Advanced Solution Controls** dialog box.
- For transient cases that use the PRESTO! pressure discretization scheme, corrections have been made in the face pressure interpolation for velocity inlet boundaries: previously the face pressure was extrapolated from within the cell without consideration of the PRESTO! scheme, and now the contribution of the PRESTO! scheme is enforced. This may result in changes compared to previous releases in the velocity distribution near the inlets, if the inlet faces are very skewed with respect to the flow direction or if the mesh is very coarse in proximity to the inlet. To revert to pre-release 17.0 behavior, use the following Scheme command: `(rpsetvar 'pressure/extrapolate-inlet/presto-on-vinlet? #f)`.

Solver-Meshing

- The six degrees of freedom solver is now second order by default. In some cases this could lead to a change in mesh motion and therefore a difference in solution from previous releases. To revert to pre-release 17.0 behavior, set `define/dynamic-mesh/control/six-dof-parameters/second-order?` to `no`.
- Improvements have been made in the quality-based smoothing algorithm, in order to achieve better parallel scalability. These improvements may result in a slightly different mesh and (as a consequence) solution compared to previous releases for dynamic mesh cases that have smoothing and/or remeshing enabled.

Turbulence

- An improvement has been made in the stress-omega Reynolds stress model, and has also been applied to the new stress-BSL model: the ratio of the specific dissipation rate to the turbulence kinetic energy is now limited, in order to avoid singularities that might lead to spikes in the ω residual history. This

may lead to small solution changes compared to prior releases. To revert to the previous behavior (where the ratio is not limited), set the rpvar ``coeff-drms-omega-or-bsl-mu-dok` to zero (the default value is 0.001).

- The $k-\varepsilon$ model and the Reynolds stress model have been improved to ensure that viscous heating contributions are taken into account by scalable wall functions. This may result in solution changes compared to prior releases.
- For DES simulations, the SST model is now the default RANS model rather than the Spalart-Allmaras model. While this changes solution results for newly set up case files, existing case files will be unchanged.
- Corrections have been made that affect the near-wall treatment of turbulence in corner cells (that is, cells that have more than one wall face). These corrections ensure the proper accounting of dissipation in the turbulence kinetic energy equation, and eliminate the solution dependency on the grid ordering. This may result in solution changes compared to prior releases.

Heat Transfer

- The solution will be more accurate for cases in which a thin wall is defined on an interface wall zone created at a mapped interface, because the thermal coupling will no longer ignore the thermal resistance due to the specified thickness.

Discrete Phase Model

- The defaults for the following settings have been made sensitive to the case setup:
 - The default DPM under-relaxation factor for transient simulations with unsteady particle tracking is set to 0.9. For all other types of analysis, the default remains 0.5.
 - The default value for **Max. Number of Steps** is now 50,000 for steady particle tracking. For unsteady tracking, the default remains 500.
 - In transient flow simulations with unsteady particle tracking, DPM source terms are no longer set to zero at the beginning of each time step. You can change this behavior using the text user interface command `define/models/dpm/interaction/reset-sources-at-timestep?`.

For existing case files, the current state of the first two settings is respected. For the third setting, the new default is enforced when a case is read that has been written by an earlier release of ANSYS Fluent.

- The Lagrangian wall film splashing algorithms have been revised for 2D simulations, in order to ensure consistency with the 3D implementation. These changes may result in differences in the splashed particle patterns for 2D simulations compared to previous releases.
- The spatial staggering for atomizer and solid cone injections has been improved. It will now produce consistent results when comparing serial and parallel solutions, independent of the number and shapes of partitions.

Eulerian Multiphase Models

- In multiphase porous media (both Mixture and Eulerian multiphase), it was reported that without the use of the reservoir model (beta for releases 16.0 and 17.0), it returned different solutions from release 15.0. This was caused by a fix to an inconsistency in the release 15.0 formulation of Darcy's law regarding the phase superficial/physical velocity when developing the generalized relative permeability model in the reservoir

model in release 16.0. Before the reservoir model becomes full feature, a fix has been added to release 17.0 that makes the linear relative permeability model the default, so that solutions in release 17.0 will match those of release 15.0 without the use of a beta feature (relative permeability model).

Parallel Processing

- The algebraic multigrid solver now automatically reorders the linear system for improved convergence. In rare cases where this degrades convergence, the reordering may be disabled using the Scheme command `(rpsetvar 'reorder/amg 0)`.
- The METIS partition library has been updated to the latest version, which generally improves the partition quality and may positively affect the convergence. This also supports partitioning large cases: cases with up to two billion cells have been partitioned successfully.

User Interface

- For a list of modified and/or new text commands for this release, see [Text Command List and Settings Changes in the *Fluent Migration Manual*](#).

Chapter 2: CFX Release Notes

The following sections contain release information for Release 17.0 of ANSYS CFX.

- [2.1. Supported Platforms](#)
- [2.2. New Features and Enhancements](#)
- [2.3. Resolved Issues and Limitations](#)
- [2.4. Updates Affecting Code Behavior](#)

2.1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the [ANSYS website](#).

2.2. New Features and Enhancements

This section lists features and enhancements that are new in Release 17.0 of ANSYS CFX. Topics include:

Turbulence

- A new Intermittency transition model is available. It is a further development based on the Gamma Theta transition model, and has advantages as described in [One Equation Intermittency Transition Model in the CFX-Solver Theory Guide](#).
- The Stress-Blended Eddy Simulation (SBES) model has been added. This is a hybrid RANS-LES model that provides strong shielding of the RANS boundary layer and fast “transition” between RANS and LES in separating shear layers. For details, see [The Stress-Blended Eddy Simulation \(SBES\) Model in the CFX-Solver Modeling Guide](#).

Transient

You can run a transient case (for example, a transient blade row case) that involves conjugate heat transfer. For details, see [Timescale Control for Solid Domains \(Transient Cases\) in the CFX-Pre User's Guide](#).

Transient Blade Row

- For Time Transformation cases, the CFX-Solver Manager can display monitor points with a corrected (physical) time on any Plot Monitor; they no longer need to be displayed on a special “Time Corrected Monitor”. In addition, you can export data from any Plot Monitor without any restrictions on exporting monitor points with different corrected time values.
- You can set up cases that simulate multiple inlet/outlet disturbances on a single blade row using the Fourier Transformation method. For details, see [Multiple Disturbances in the CFX-Solver Modeling Guide](#).
- You can set up cases that involve a long-period disturbance, such as an impeller or rotor attached to a full 360° domain, or a fan in a crosswind, using the Fourier Transformation Transient Rotor Stator (FT-TRS) interface. For details, see [Case 6: Transient Rotor Stator Cases with Asymmetric Flow](#).
- You can set up multiple stage cases using the Time Transformation method. For details, see [Case 5: Transient Rotor Stator Multi-Stage Cases in the CFX-Solver Modeling Guide](#).

- You can stop and then restart a Time Transformation or Fourier Transformation case with an increased number of time steps per period. For details, see [Stopping and then Restarting Simulations with an Increased Number of Time Steps Per Period in the CFX-Solver Modeling Guide](#).

System Coupling

You can use CFX with System Coupling. For details, see [Coupling CFX to an External Solver: System Coupling Simulations in the CFX-Solver Modeling Guide](#).

Initialization

New initialization options are available. For details, see the descriptions for the **Initialization Option** Solution cell property in [Properties View in the CFX Introduction](#). This property synchronizes interactively with the **Initialization Option** setting in CFX-Solver Manager's **Define Run** dialog box on the **Initial Values** tab. For details on the **Initial Values** tab, see [Initial Values Tab in the CFX-Solver Manager User's Guide](#).

Solution Caching for Design Point Studies

There are two new ways to control the retention of solution data:

- **For Old Design Point Solutions**

- Solution cell property "Keep Latest Solution Data Only" ([Properties View in the CFX Introduction](#))
- Shortcut menu command "Clear Old Solution Data" ([Context Menu Commands in the CFX Introduction](#))
- Workbench preference "Keep Latest Solution Data Only" ([CFX in the Workbench User's Guide](#))

Note: When using System Coupling, you should set the Solution cell property "Keep Latest Solution Data Only" to "False" (which is not the default) to retain old solution data.

- **For Cached Design Point Solutions**

- Solution cell property "Cache Solution Data" ([Properties View in the CFX Introduction](#))
- Shortcut menu command "Clear Cached Solution Data" ([Context Menu Commands in the CFX Introduction](#))
- Workbench preference "Cache Solution Data" ([CFX in the Workbench User's Guide](#))

Large Problems

You can partition, solve, and interpolate large problems. For details, see [Executable Selection in the CFX-Solver Manager User's Guide](#).

User Fortran

- Before Release 17.0, Fortran routine `USER_GET_MESHDATA` was not supported for solid domains. Starting with Release 17.0, `USER_GET_MESHDATA` is supported for solid domains.
- You must have the Intel Fortran 15.0.2 compiler to build User Fortran for ANSYS CFX (Windows and Linux).

Variables

The following variables can now be selected for output into the results: `Spinodal Pressure`, `Spinodal Temperature`.

Enhancements to Stage (Mixing-Plane) model

- The Constant Total Pressure option for **Downstream Velocity Constraint** has been made the default option instead of Stage Average Velocity.
- Improvement to energy closure

These two changes improve the conservation of Total Pressure and Rothalpy across a mixing-plane. To revert back to previous numerics, use the following expert parameter option: `stage energy closure option = 1`.

2.3. Resolved Issues and Limitations

This section lists issues and limitations that have existed in previous releases, but that have been resolved and removed in Release 17.0 of ANSYS CFX.

- A call to `USER_GET_MESHDATA` will return a different value for the near-wall distance. Before Release 17.0, a call to Fortran routine `USER_GET_MESHDATA` to get the near-wall distance would return the near-wall distance multiplied by a constant. The value of the constant depended on the choice of turbulence model. Starting with Release 17.0, a call to `USER_GET_MESHDATA` to get the near wall distance returns the (unmodified) near-wall distance. Note that the near-wall distance is evaluated as the distance from an element's centroid to a boundary face, measured normal to the boundary face.

2.4. Updates Affecting Code Behavior

This section contains a list of changes that may cause the solution results from ANSYS CFX to differ between Release 17.0 and Release 16.2.

- The CFX-Solver implementation of the CEL function `massFlow` inaccurately computed the total mass flow, and the phasic mass flow, at a boundary with phase-specific boundary conditions (e.g. degassing). The total and phasic mass flows are now more accurately computed at such boundaries.

Chapter 3: TurboGrid Release Notes

The following sections contain release information for Release 17.0 of ANSYS TurboGrid.

[3.1. Supported Platforms](#)

[3.2. New Features and Enhancements](#)

3.1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the [ANSYS website](#).

3.2. New Features and Enhancements

This section lists features and enhancements that are new in Release 17.0 of ANSYS TurboGrid.

- Note that, when **Boundary Layer Refinement Control > Method** is set to `Proportional to Mesh Size`, the number of elements across the boundary layer is calculated as $\text{Base Count} * \text{Global Size Factor} * (\text{Factor Base} + \text{Factor Ratio} * \text{Global Size Factor})$. The default values of Factor Base and Factor Ratio have been changed from 0 and 3 respectively to 3 and 0 respectively.
- You can control the target expansion rate for the streamwise distribution of mesh elements in the inlet and outlet domains. For details, see [Inlet Domain and Outlet Domain Settings in the TurboGrid User's Guide](#).

Chapter 4: ANSYS BladeModeler Release Notes

The following sections contain release information for Release 17.0 of BladeGen and BladeEditor.

- [4.1. Supported Platforms](#)
- [4.2. BladeGen](#)
- [4.3. BladeEditor](#)

4.1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the [ANSYS website](#).

4.2. BladeGen

Note

Corrections to the BladeGen help are listed at [ANSYS BladeGen Documentation Corrections and Additions in the TurboSystem User's Guide](#).

4.2.1. New Features and Enhancements

This section lists features and enhancements that are new in Release 17.0 of ANSYS BladeGen.

- Neutral Data Files can be imported and exported by BladeGen. For details, see [NDF File Import/Export Details and Limitations in the TurboSystem User's Guide](#).

4.3. BladeEditor

4.3.1. New Features and Enhancements

This section lists features and enhancements that are new in Release 17.0 of ANSYS BladeEditor.

- Neutral Data Files can be loaded and written by BladeEditor. For details, see [Loading a Neutral Data File \(*.xml\) in the TurboSystem User's Guide](#) and [Writing a Neutral Data File \(*.xml\) in the TurboSystem User's Guide](#), respectively.

Chapter 5: CFD-Post Release Notes

The following sections contain release information for Release 17.0 of ANSYS CFD-Post.

- 5.1. Supported Platforms
- 5.2. New Features and Enhancements
- 5.3. Resolved Issues and Limitations
- 5.4. Updates Affecting Code Behavior

Note

If CFD-Post crashes while creating graphics on a machine with NVIDIA graphics hardware, try upgrading your graphics driver or set environment variable `VIEWER_FACE_LIST` to 999.

5.1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the [ANSYS website](#).

5.2. New Features and Enhancements

This section lists features and enhancements that are new in Release 17.0 of ANSYS CFD-Post.

- You can specify user surfaces in CFX-Pre Output Control so that solution data is only written from the user surface instead of from every vertex of the mesh, thereby reducing the size of results files. For details, see [User Locations in the CFX-Pre User's Guide](#).
- You can use the **Timestep Animation** option in CFD-Post to create an animation using only timesteps as inputs, allowing you to easily animate transient results. For details, see [Timestep Animation in the CFD-Post User's Guide](#).
- You can export graphics from CFD-Post in .avz file format, which can be displayed in the ANSYS Viewer. For details, see [Save Picture Command in the CFD-Post User's Guide](#).
- You can load Forte results files (.ftind) in CFD-Post.
- When making plots in CFD-Post, you can define a data series from the monitor data or solution residuals of a results file. For details, see [Monitor Variable Selection in the CFD-Post User's Guide](#).
- There is an option that controls how mass flow is computed. For details, see [Files in the CFD-Post User's Guide](#).

5.3. Resolved Issues and Limitations

This section lists issues and limitations that have existed in previous releases, but that have been resolved and removed in Release 17.0 of ANSYS CFD-Post.

- CFD-Post is no longer limited to reading the first two columns of data when creating a Chart Series using data from a file.

5.4. Updates Affecting Code Behavior

This section contains a list of changes that may cause the computed results from ANSYS CFD-Post to differ between Release 17.0 and Release 16.2.

- The algorithm for the `massFlowAveAbs` function has been improved.
- The format/style of CFD-Post reports produced from Workbench are now consistent with general Workbench-produced reports; reports from standalone CFD-Post continue to use the standard CFD-Post format.
- For turbomachinery-related reports ("Turbo reports"), the computed value of power may be different because, in some cases, it is now computed using total enthalpy instead of static enthalpy.

Chapter 6: Polyflow Release Notes

The following sections contain release information for ANSYS Polyflow 17.0.

- [6.1. New Features](#)
- [6.2. Supported Platforms](#)
- [6.3. New Limitations in ANSYS Polyflow 17.0](#)
- [6.4. Past Versions of ANSYS Polyflow Release Notes](#)

6.1. New Features

The new features in ANSYS Polyflow 17.0 are as follows:

- The GPU accelerator is now compatible with the AMF direct solver when it is coupled with the ILU and/or secant methods.
- Contact detection is now available for multiple fluid free surfaces coming into contact. For additional information on fluid-fluid contacts, see [Fluid-Fluid Contact in the Polyflow User's Guide](#).
- Polyflow now supports the use of multiple GPUs to accelerate computation time in parallel.
- Polymer mechanical properties can vary with orientation. Polyflow can now track extension and orientation of the polymer during blow molding and thermoforming simulations. For additional information, see [Evaluation of the Extension Components in the Polyflow User's Guide](#).
- By default, the CFD-Post output will now be created in a specific thread, in order to reduce the serial operations of the solver and increase parallel efficiency.
- There is a new option available for output triggering called **Output at initial state** that outputs the initialized state of a simulation (state prior to beginning calculation). This feature allows you to review the initialized state of an old results file and is useful for both transient and evolution type simulations.
- Evolution functions defined on heat generation from viscous heating now apply to the heat generated from wall friction as well.
- It is now possible to update the orientation of fibers in orthotropic reinforced materials when there are displacements due to deformation of the material. For additional information, see [Orthotropic Materials](#) and [Problem Setup for Reinforced Materials in the Polyflow User's Guide](#).
- When defining a slip condition on a boundary, a new default interpolation option for the force density field has been added. The new option is program controlled, so that Polydata selects the interpolation according to the interpolation of the velocity field and slipping parameters, in order to adapt to the setup and produce better results for slipping simulations.
- Temperature programming is now available for blow molding problems. For additional information on temperature programming, see [Temperature Programming in the Polyflow User's Guide](#).
- A new convergence test has been added for species transport problems. This allows you to soften the convergence criterion for transported species (by increasing its value) in order to reduce solution time,

as transported species fields typically take longer to converge than the main fields. For additional information, see [Convergence and Divergence in the Polyflow User's Guide](#).

- The default finite-element interpolation for 3D flows has changed from the mini-element to a linear representation of the velocity, in order to increase the robustness (especially for nonisothermal cases).

6.2. Supported Platforms

Information about past, present, and future operating system and platform support is viewable via the [ANSYS website](#).

6.3. New Limitations in ANSYS Polyflow 17.0

There are no new limitations to note for ANSYS Polyflow 17.0. For limitations that are present in ANSYS Polyflow 17.0 but that were discovered during previous releases, see [Known Limitations in ANSYS Polyflow 17.0](#) in the [Polyflow User's Guide](#).

6.4. Past Versions of ANSYS Polyflow Release Notes

Previous versions of the ANSYS Polyflow Release Notes are installed as PDFs with the product.

To access these PDFs, point your web browser to

- For Windows:

```
path\ANSYS Inc\v170\polyflow\polyflow17.0.x\help\index.htm
```

- For Linux:

```
path\ansys_inc\v170\polyflow\polyflow17.0.x\help\index.htm
```

where *path* is the directory where you installed ANSYS Polyflow and *x* represents the appropriate number for the release (for example, 0 for polyflow17.0.0).

Chapter 7: Forte Release Notes

The ANSYS Forte 17.0 release (also known as FORTE 40151) is the first release of Forte CFD to be included in the ANSYS Unified Installer and to use the ANSYS License Manager.

[7.1. New Features and Enhancements](#)

[7.2. Known Limitations](#)

[7.3. Supported Platforms](#)

7.1. New Features and Enhancements

This section lists new features and enhancements in Release 17.0 of ANSYS Forte CFD, organized by topic. New features and enhancements listed are those that have been included since the ANSYS Forte 40145 release in June 2015.

Simulation Interface

- Added a new option under Monitor Probes that allows specification of a collection of discrete points (a “point cloud”) on a surface, which may be used for saving heat-transfer information, including accumulated time averages of heat flux, near-wall temperature, and heat-transfer coefficients for use in conjugate heat-transfer simulations. This option may be associated with any wall boundary condition; the point cloud will move with a moving boundary, such as a piston or valve surface.
- Added the ability to specify criteria for Solution Adaptive Mesh refinement to allow dynamic refinement of automatically generated Cartesian meshes, based on instantaneous solution variable values and spatial gradients. A variety of optional user controls are provided.
- Added the ability to access a flame-speed-table library generated using detailed chemistry for an arbitrary fuel blend, simply by specifying the composition of the fuel blend. A new built-in flame-speed-table library provides a comprehensive set of fuel components and blending rules covering engine-relevant conditions for on-the-fly determination of flame speeds used in the flame-propagation model.
- Added an option to explicitly set the number of sectors in periodic symmetry for the Sector Mesh Generator, when generating a body-fitted sector mesh, eliminating any potential error from setting sector degrees for odd-number sectors.
- Added the option to create monitor probes that are based on initialization regions (for volume reporting) or on boundary conditions (for surface reporting).

Job Submission, Monitoring, and Running Options

- Allow monitoring of accumulated time averages of heat-transfer information at point-cloud locations for conjugate-heat-transfer applications.
- Ability to launch, run, and post-process simulations from ANSYS Workbench, using a new ICE-Forte node in the Workbench framework. The WB-ICE (R17) option allows automatic preparation of geometry and initial mesh settings for either automatic mesh (Cartesian Immersed Boundary) cases or for body-fitted sector-mesh cases.

- Intel MPI version 5 is now supported, as well as the previously supported Intel MPI version 4.
- Added the reporting of global equivalence ratio and residual fraction as time-dependent monitor variables to the spatially averaged monitor file, thermo.cvs.

Engineering Models and Computation

- Provided a new option to include dynamic, solution-adaptive refinement of the computational mesh for automatically generated Cartesian meshing, based on local solution variables and their gradients.
- Significant speed-up in table-lookup operations (> 100X speed-up in some cases) for applying stratified initial conditions, using flame-speed tables, and other internal look-up functions.
- Revised the definition of equivalence ratio to consider oxygenated fuel, fuel-rich mixtures and high EGR conditions more accurately. The revised definition is also now consistent with the one used in creating laminar flame-speed lookup tables. This revision is also applied to the equivalence ratio values reported in spatially resolved and spatially averaged outputs.

Licensing

- ANSYS Forte now requires ANSYS license keys and connection to a license server running the ANSYS License Manager; the Reaction Design license keys and License Manager will not enable this ANSYS Forte 17.0 (Forte 40151) version.

7.2. Known Limitations

- In the chemistry-solver options, the **Include Turbulence Kinetics Interaction** option should not be selected when using the **Particle Tracking** option for soot modeling. Numerical error can accumulate with the combination of these models to cause mass imbalance.

7.3. Supported Platforms

Information about present and future operating system and platform support is viewable via the [ANSYS website](#).

Part III: ANSYS Electronics Products

Release notes are available for the following ANSYS Electronics products:

[Icepak \(p. 81\)](#)

Chapter 1: Icepak Release Notes

Release 17.0 of the ANSYS Icepak application offers most of the capabilities from previous releases plus many new features and enhancements.

- [Introduction \(p. 81\)](#)
- [New and Modified Features in ANSYS Icepak 17.0 \(p. 81\)](#)
- [Resolved Issues and Limitations in ANSYS Icepak 17.0 \(p. 82\)](#)

1.1. Introduction

ANSYS Icepak 17.0 is a release of ANSYS Icepak that has new features and resolved issues and limitations.

1.2. New and Modified Features in ANSYS Icepak 17.0

- Graphical User Interface
 - Added capability to disable animation of model rotation when changing views. See Section [Display Options](#) of the User's Guide.
 - Added capability to visualize an object's heat flux, power, or temperature properties in 3D. See Section [Display Powermap Property](#) of the User's Guide.
 - Added capability to filter an object list by object type. See Section [Drop-Down List](#) of the User's Guide.
- Model Import/Export
 - Added capability to independently reposition and resize trace files when importing them. See Section [Importing Trace Files](#) of the User's Guide.
 - Added capability to reposition traces for a block or PCB. See Section [Importing Trace Files](#) of the User's Guide.
 - Added capability to automatically resize an object's thickness (block or PCB) when modifying its trace layer thickness. See Section [Importing Trace Files](#) of the User's Guide.
- Model Building
 - Added capability to scale an object around a point. See Section [Copying an Object Using the Copy object Panel](#) of the User's Guide.
 - Added capability to define the cabinet's autoscale factor. See Section [Interactive Editing](#) of the User's Guide.
 - Added capability to copy shape or bounding box information when using the **Copy from** option. See Section [Copying an Object Using the Object selection Panel](#) of the User's Guide.

- Object Parameters
 - Added capability to specify interface resistance between a face node and objects connected to it. See Section [Star Network Model](#) of the User's Guide.
- Meshing
 - Added capability to display meshing errors in the **Messages** window.
- Solver Setup
 - Added capability to use the Remote Solve Manager in ANSYS Workbench. See Section [Using Icepak With the Remote Solve Manager \(RSM\)](#) of the User's Guide.
 - Added advanced linear solver controls. See Section [Selecting Linear Solver Advanced Controls](#) of the User's Guide.
- Post-processing
 - Added capability to display a post-processing object's mesh, contours, and vectors from the post-processing context menu. See Section [The Post-processing Node Context Menu](#) of the User's Guide.
 - Added capability to display the range of values for a selected variable in the **Message** window. See Section [Defining an Isosurface](#) of the User's Guide.
- Reports
 - Added capability to create a metadata file to be read by ANSYS Engineering Knowledge Manager. See Section [Metadata Files](#) of the User's Guide.
 - Added capability to define a custom report from the model context menu. See Section [The Model Node Context Menus](#) of the User's Guide.

1.3. Resolved Issues and Limitations in ANSYS Icepak 17.0

- Transparency rendering of objects is inconsistent and incorrect. (59699)
- When running a transient optimization, the objective function is computed using values from the initial time-step rather than the final time-step. (60013)
- Solution diverges when non-conformal meshing is enabled. (68912)
- The initial parameter value setup panel does not appear when creating a variable for the **Altitude** parameter. (68944)
- IDF import creates duplicate components if the library definitions contain multiple definitions with the same package names and same part number prefix as that of the component. (88972)
- Icepak does not allow polygonal thick plates with holes. (89702)
- Mesh is missing in a region between the intersection of a non-conformal assembly and polygonal fluid block. (90295)
- Icepak's 2D multi-level meshing displays incorrect behavior on detailed grille. (94866)
- When exporting a block with a geometry other than **Prism**, the power value is not included. (96632)

- When performing a multiple-object edit from the model tree, multiple object panels appear. (96714)
- Local mesh controls are not available when editing multiple **Recirc** openings. (96715)
- Post-processing objects do not display any contours if an object or group of objects is inactive. (96716)
- No option exists to display contours of meshed regions only. (96828)
- When selecting **Radiation** in **Basic Parameters**, the default is **Reference Temperature** rather than **All objects**. (97187)
- Clicking **Reset** in the **Basic solver settings** panel in the **Natural Convection** field, an excess dash symbol is inserted. (97189)
- Polygonal PCBs are rotated incorrectly. (97298)
- Thermal diffusivity values of some materials are incorrectly displayed. (97322)
- Columns are shifted incorrectly in the csv export of a summary report. (97724)
- When solving a specific model using parallel processing, the solver quits and displays an error. (97901)
- Discrepancy in results between R15 and R16 for a specific case. (98312)
- After running the **Cleanup** operation, some files are not removed. (98315)
- Icepak does not create a HTML summary for models with individual side specifications. (98339)
- When the vertical direction of the graphics window is set to Z, the isometric view is slightly inclined. (98390)
- When displaying objects by power or material, CAD objects appear black. (98392)
- Metal fractions are incorrect for models with extremely small grid size, less than 5 microns, due to floating point error. (98402)
- Deactivated layers are not updated in the total package substrate height. (98437)
- When fast trials is enabled, no solution data is available after the trials are complete. (98642)
- When copying an assembly, some mesh parameters are not copied. (98721)
- Fluid blocks with higher priority do not create holes in PCB with ECAD data. (99087)
- After editing a non-conformal assembly mesh and closing the panel, typing any letter on the keyboard enters the character into that last edited field in the non-conformal assembly panel. (99178)
- When importing ECAD, an unnecessary copy of the brd file is created. (100152)
- Searching for a material in the **Libraries** tab produces invalid path error. (100212)
- Icepak creates many coupled walls and takes a long time to solve for some computer configurations. (100669)
- Solutions produce a Fluent error when a temperature surface monitor is defined on min y or max y side of object. (100706)
- A network temperature plot does not display the temperature graph and produces an error if the network has more than two internal nodes. (100708)

- When performing Q3D to Icepak coupling in Workbench, Fluent produces an error while writing temperature files. (100900)
- Saving convergence plots as a PNG after editing display options produces an error. (101023)
- Performing Siwave DC simulation with an Icepak sitemp file produces an error. (101051)
- Monitor point temperature plots display incorrect values after running a transient case. (101137)
- Writing an sitemp file produces an error. (101450)
- When performing Q3D to Icepak coupling in Workbench, the EM Mapping panel produces an error when accessing it. (101599)
- When meshing with **Allow stair-stepped meshing** enabled, an opening with a higher priority than the wall it is on is not meshed. (101868)
- When solving a failed fan simulation, a pressure loss curve specification causes the solve process to fail. (101895)
- Full report results differ from previous release. (101897)
- With **Concurrency** set at 2, attempting to mesh produces a licensing error. (102027)
- When making a group invisible, decorations are still displayed. (102095)
- When importing IDF power information, the **power**, **jc**, and **jb** information is placed on model parts with similar group names rather than the intended part. (102772)
- Volume flow surface is not available for 3D fans. (102835)
- When using local mesh controls, **Low Z Height** does not work for a higher priority fluid block embedded within a lower priority solid block. (103029)
- When renaming multiple objects, the objects are not renamed in the correct sequential order. (103124)
- Polygonal blocks are not mirrored correctly. (103396)
- **Visible grid** and **Action** options available under the **View** menu do not function as reported in the Icepak User's Guide. (104124)
- When coupling HFSS and Icepak in Workbench, volumetric loss data is not transferred to Icepak. (104129)
- When viewing Icepak in 4 window mode, the model corresponding to isometric view is not scaled to fit. (104219)
- IDF import with ECAD file option does not work correctly. (104238)
- The **Quality** tab in the mesh dialog shows incorrect entries after reading in older versions of an Icepak project. (104258)
- Summary report displays incorrect value for a face in a non-conformal assembly. (104315)
- Total power applied on an ECAD PCB is not displayed in the Power and Temperature limits table or in the show object by priority. (104347)

- Joule heating boundary condition: input file for the solver is overwritten while doing parametric trials. (104447)
- Using Full data restart, temperature and mass flow rate values reported on a block, wall, heatsink, and plate are incorrect. (104487)
- If ECAD files are selected as part of cleanup, ECAD data is deleted for inactive objects with ECAD data, including the bool file. (104859)
- Ray tracing radiation model fails during parallel simulations for cases that include thin conducting plates. (105062)
- Monitor files do not contain the zero time-step for state-space simulations. (105598)
- Sentinel TI powermaps are not imported as an assembly of sources. (105612)
- When importing ANF ECAD data, the trace layer thicknesses in Icepak are truncated and no longer match the thickness in the original ANF file. (105613)
- Mesher-HD failed due to slight misalignment of geometry in an assembly. (105629)
- An object face displays incorrect temperatures when solution is run in parallel. (105630)
- Problems are encountered when using selection and snapping tools. (105631)
- ODB++ files from Mentor PADS are not imported correctly. (105731)
- After an error occurs while creating monitor points, selection points remain on the user interface. (105759)
- A fan can't create a hole for a block when xE or yE is smaller than xS or yS. (105775)
- When mirroring a heat sink object, contact resistance and bonded fin plate heights become negative values, causing Fluent to fail. (105824)
- Meshing fails when using Mesher-HD and 3D multi-level meshing. (105982)
- An exit exhaust angle does not work correctly. (106378)
- Solver fails on Windows HPC clusters for models with blower objects. (106380)
- Non-conformal mesh does not allow fluid flow through assembly boundary. (107159)
- Models created from existing models that have surface monitors cannot be opened. (107361)
- Legacy Icepak-Simplorer coupling is not available in R16.1 (107380)
- Remesh is not triggered when loading a solution and running the case again with different number of assemblies meshed separately. (107595)
- When a 2D source is embedded in a block, and the heat flux is processed using an object face, only one side displays the heat flux. (107746)
- When using full data restart for a transient simulation, only data for the first time-step is retrieved. (107837)
- Replacing IDF components with pre-defined libraries does not work. (107841)

- Object faces plotting temperature do not display correctly. (107975)
- "Uninterrupted spaceware event" warning message appears when using a Citrix-based thin client system to run jobs. (108001)
- When editing a 2D object or prism block, switching between start/end and start/length resets all fields. (108073)
- A 3D polygonal shape is created after snapping the max X side of the cabinet with 2D polygonal opening to the max X side of an enclosure block and resizing the opening when prompted. (108268)
- An error occurs when creating a summary report with more than one object selected. (108683)
- Interface creation takes longer than expected. (108692)
- Turbulence quantities are unable to be processed for k-omega turbulence model. (108888)
- **Heat flow** value is displayed incorrectly. (109575)
- Metal fractions are incorrect after rotating and translating a board. (109724)
- Specifying an offset for a staggered heat sink does not work correctly. (109982)
- Some trace definitions from Cadence ECAD are not imported correctly. (110570)
- When plotting mass flow rate, a density value error occurs and mass flow rate is not plotted. (110787)
- Surface monitoring graphs do not display Max Y or Min Y values for heat flux. (111107)
- Heat loss in an overview report does not match the input heat generated. (111269)
- Components missing in Icepak after IDF import. (111361)
- The **Show too hot** option does not work correctly for a package object. (111585)
- Model fails to upgrade from R15 due to the deletion of bool file during file cleanup for inactive objects with ECAD. Related to 104859. (111731)
- Edits to object settings are not saved after performing a save operation. (111912)
- Electric specifications are missing when writing joule heating UDS. (111916)
- The wrong resd files are copies for transient run restarted from interpolated data. (111959)
- When a transient simulation is run restarting from the full data of a completed transient simulation, the contours corresponding to the time duration of the previous solution are unphysical. (111973)
- Mesher-HD failed to generate valid mesh with zero slack assembly. (112065)
- Post-processing results differ when simulations are run in serial and parallel mode. (112675)
- The solution residuals plot displays the incorrect number of iterations completed. (112677)
- For some Cadence ECAD imports, vias information is not received by Icepak. (112716)
- IDX import of components with blank names produces stack trace error. (112785)

- Transient report generates floating point error when **All times** option is enabled. (112788)
- Icepak freezes while running trials with too many trials and parameters. (113186)
- The mesher fails to mesh an empty cabinet. (113228)
- The mesher is unable to account for the network block side that overlaps with the face of a PCB object. (113232)
- The mesher is influenced by inactive objects. (113684)
- The power law specification for a resistance object produces a very high viscous resistance. (113973)
- When importing a network object, if a space exists in the node name, the dimensions of the network faces are incorrect. (114778)
- Time-steps are not plotted correctly when using piecewise constant. (115292)
- An error occurs when running a trial if no **Restart ID** is specified. (115570)
- Ray tracing radiation simulation fails for a specific model. (115618)
- A board is incorrectly resized to fit the cabinet after being rotated. (115721)
- Cadence ECAD import does not include solder pads defined as polygons. (115767)
- When computing the heat flow of an opening inside a fluid block, the value is incorrect. (115859)
- The joule heating variable in Icepak is lost after updating geometry in Design Modeler. (116059)
- When creating multiple variation plots, the initial plot is deleted when loading another. (116289)
- For a grille, specifying a linear loss curve with two data points produces an incorrect pressure drop. (116476)
- For transient simulations, an opening's temperature is displayed in Kelvin instead of SI units. (116496)
- After ECAD import, displaying thousands of vias, used and unused, in the via information panel causes the Icepak user interface to crash. (116504)
- After closing the Tcl console, the check mark next to it in the **View** menu remains. (116840)
- Importing gerber data into Icepak fails when via names contain spaces. (116901)

Part IV: ANSYS Geometry & Mesh Prep Products

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

[DesignModeler \(p. 91\)](#)

[SpaceClaim \(p. 93\)](#)

[CAD Integration \(p. 95\)](#)

[Meshing \(p. 97\)](#)

[IC Engine \(p. 99\)](#)

[ICEM CFD \(p. 101\)](#)

[Fluent Meshing \(p. 105\)](#)

Chapter 1: Geometry Release Notes

This section summarizes the new features in DesignModeler Release 17.0. Topics include:

Capping Faces Capability in Section Planes

To visualize the cross section for solid bodies, Section Planes can now show capping faces, and capping faces by body color, via application-level commands. Capping faces are on by default and set to the body color.

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

Expanded Face Coloring Options

The Face Coloring graphics options now includes a Fluid/Solid Type Legend to identify the body type as being either fluid or solid.

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

Move Tool Enhancement

The Move option of the Body Transformation feature now allows for the reposition or transformation of bodies by PF Points.

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

Project Units Settings Introduced

Control of DesignModeler's units has been improved. Accessible via the path below, the geometry editor can be set to always adopt project units, assign the project unit only at the start of a new file, or assign units independently of the project.

For more information, see Application Options> Units in the [DesignModeler User's Guide](#).

Shared Topology Capabilities from DesignModeler to SpaceClaim

To improve the workflow from DesignModeler to SpaceClaim, DesignModeler geometry containing shared topology is restricted from importation into SpaceClaim. The model will be transferred in the state just prior to the Share Technology feature. Likewise, geometry imported from SpaceClaim to DesignModeler will not have shared topology applied.

For more information, see File Format Support> ANSYS DesignModeler in the [CAD Integration](#) section of the ANSYS Help.

Weld Feature Enhancement

The Weld feature has been enhanced to accommodate more complex connections, with an improved ability to create a continuous weld between bodies.

For more information, see 3D Modeling> Advanced Features and Tools> Weld in the [DesignModeler User's Guide](#).

Chapter 2: SpaceClaim

For detailed information specific to SpaceClaim 2016, see the SpaceClaim 2016 Release Notes on the [ANSYS Customer Portal \(support.ansys.com\)](http://support.ansys.com) at Knowledge Resources> Online Documentation> Geometry.

To view previous release notes, select applicable release under the Previous Releases menu at Knowledge Resources> Online Documentation. Alternatively, see Downloads> Previous Releases> ANSYS Documentation and Input Files to select the applicable Release Documentation file.

Chapter 3: CAD

Note

Support for the Plug-in Creo Parametric's Wildfire 5.0 version is expected to be discontinued after the release of ANSYS 17.0

This section summarizes the new features in CAD Integration Release 17.0.

ANSYS Part Manager File Support

For more information, see File Format Support> ANSYS Part Manager in the [CAD Integration](#) section of the ANSYS Help.

Expanded AutoCAD Support

AutoCAD is now supported as a Reader which does not require that an AutoCAD system to be installed.

For more information, see File Format Support> AutoCAD in the [CAD Integration](#) section of the ANSYS Help.

Expanded Solid Edge Support

Solid Edge is now supported as a Reader which does not require that a Solid Edge system to be installed.

For more information, see File Format Support> Solid Edge in the [CAD Integration](#) section of the ANSYS Help.

Material Processing Support

Material Processing is now supported for SolidWorks (Associative Geometry Interface).

For more information, see File Format Support> SolidWorks in the [CAD Integration](#) section of the ANSYS Help.

Project Schematic Support for Faceted File Types

Selecting the Geometry cell now launches ANSYS SpaceClaim when the following file formats are active:

- AMF: Additive Manufacturing File
- OBJ: Wavefront OBJ format
- PLY: PLY 3D geometry format
- STL: Stereolithography format

Geometry Interfaces Update for New CAD Releases

Geometry interfaces are updated to support new CAD releases including:

- ACIS 2016 (Reader)
- AutoCAD 2016 (Reader)
- Autodesk Inventor 2016 (Plug-in)
- Autodesk Inventor 2016 (Reader / Plug-in support on Windows 10)
- CATIA V5 (Reader/ Plug-in)
- CATIA V6 (Reader)
- Creo Parametric 3.0 (Plug-in)
- JT (Reader)
- Parasolid 27.0 (Reader)
- Rhinoceros V5.020 and v4.0 (Reader)
- SketchUp V2014 and V2013 (Reader)
- Solid Edge ST8 (Reader / Plug-in)
- SolidWorks 2015 (Reader / Plug-in)
- SolidWorks 2016 (Plug-in support on Windows 10))

For detailed version support information, see CAD Integration> Geometry Interface Support in the [CAD Integration](#) section of the ANSYS Help.

Information about past, present and future CAD, operating system and platform support is viewable via the ANSYS, Inc. website (Support> Platform Support).

Chapter 4: 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:

- 4.1. Incompatibilities and Changes in Product Behavior from Previous Releases
- 4.2. Nonlinear Mechanical Enhancements
- 4.3. Shell Meshing Enhancements
- 4.4. Mesh Control Enhancements
- 4.5. Mesh Editing Enhancements
- 4.6. Meshing Application Advisory

Many of the enhancements detailed in the [Mechanical Application Release Notes \(p. 3\)](#) are relevant to the Meshing application. In particular, the following sections describe enhancements that can also affect Meshing:

- [General Enhancements \(p. 4\)](#)
- [Performance Enhancements \(p. 5\)](#)
- [Graphics Enhancements \(p. 6\)](#)
- [Geometry Enhancements \(p. 7\)](#)
- [Contact and Connection Enhancements \(p. 8\)](#)

4.1. Incompatibilities and Changes in Product Behavior from Previous Releases

- The mesh status has been improved for parallel part meshing.

When you generate the mesh, as each part is meshed, the **ANSYS Workbench Mesh Status** window displays the status of the part, and the topology is highlighted in the **Geometry** window.

4.2. Nonlinear Mechanical Enhancements

The following enhancements for nonlinear mechanical systems have been made in Release 17.0:

- The new **Nonlinear Mechanical** physics preference provides global control defaults suitable for nonlinear mechanical analysis.

If you select the **Nonlinear Mechanical** physics preference, the **Shape Checking** control defaults to a new **Nonlinear Mechanical** option, which provides Bézier skewness shape checking.

4.3. Shell Meshing Enhancements

The following shell meshing enhancements have been made in Release 17.0:

- The new **Enable Washers** option in the **Sizing** group controls the pattern of the mesh around holes.

If you enable this option, a layer of equally-spaced quadratic elements will be generated around each cutout. You can also specify the [height of the washers](#), and the [behavior of the washer elements around holes that are close to a boundary](#).

4.4. Mesh Control Enhancements

The following global mesh control enhancements have been made in Release 17.0:

- The new **Export Format** option in the **Defaults** group enables you to export the mesh as a cell-based Fluent mesh.
- The **Use Advanced Size Function** control is now called **Size Function**.

The following size function options have been renamed:

- **Off** is now called **Adaptive**.
- **Fixed** is now called **Uniform**.
- The **Max Size** control is now called **Max Tet Size**.
- The **Defeaturing** group has been removed.

The **Pinch Tolerance** and **Generate Pinch on Refresh** controls have been moved to the **Advanced** group. The **Automatic Mesh Based Defeaturing** and **Defeaturing Tolerance** controls have been moved to the **Sizing** group.

- The **Patch Conforming Options** group has been removed.

The **Triangle Surface Mesher** control has been moved to the **Advanced** group.

- The **Patch Independent Options** group has been removed.

The **Topology Checking** control has been moved to the **Advanced** group.

- The **Shape Checking** and **Element Midside Nodes** controls have been moved from the **Advanced** group to the **Defaults** group.

4.5. Mesh Editing Enhancements

The following mesh editing enhancements have been made in Release 17.0:

- The new **Contact Match** feature enables you to match mesh nodes between solid tet meshed bodies within a specified tolerance.

After generating contact matches, you can use **Node Merge** to merge the coincident nodes and obtain a conformal mesh.

- **Quickly Create Objects Based on Contact Regions**. You can easily create mesh connections and contact matches using the new RMB **Create** option that is available on **Contact Region** objects.

4.6. Meshing Application Advisory

The Gap Tool has been removed from the product.

Chapter 5: IC Engine Release Notes

In this release there is a new **IC Engine (Forte)** analysis system added into Workbench. Using **IC Engine (Forte)** system you can setup a seamless workflow for Forte CFD. It mainly reduces the manual effort in preparing the geometry in ANSYS DesignModeler and the surface mesh in ANSYS Meshing. The transient volume meshes are handled during the simulation in Forte CFD. Additionally the workflow includes the basic setup for simulation with Forte CFD and post-processing in CFD-Post. The workflow is customized for cold flow and combustion simulations. For combustion simulations different approaches are applicable, full-engine simulations and sector simulations using the automated mesh generation approach (AMG). Additionally sector geometries can be mesh with body fitted meshes using the FORTE Sector Mesh Generator (SMG). Forte specific journal can be used to setup the Forte models/sub models for simulation. Simulation results data from the Forte solver has been added to the report which will be generated after the run.

Chapter 6: ICEM CFD Release Notes

This section summarizes the new features in ICEM CFD Release 17.0. Topics include:

- [6.1. Highlights of ANSYS ICEM CFD 17.0](#)
- [6.2. Documentation](#)

6.1. Highlights of ANSYS ICEM CFD 17.0

Release 17.0 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.

ANSYS ICEM CFD 17.0 includes new features and improvements in the following areas:

- [6.1.1. Initializing blocking](#)
- [6.1.2. Splitting free blocks](#)
- [6.1.3. Editing blocks](#)
- [6.1.4. User interface](#)
- [6.1.5. Output interfaces](#)

6.1.1. Initializing blocking

New options are available for initializing blocking:

- An option to initialize **2D Surface Blocking** from **Curves** has been added. A single 2D surface block is created from the selected set of curves. The process expects the curves to be approximately planar and form a single region.
- An option to create **3D Multizone** blocking as a single operation has been added. This automated method uses the sequential operations of 2D Surface Blocking followed by 3D Fill.

The software tries to decompose the volume into a combination of mapped, swept and free blocks. The options **Source Imprint Surfaces** and **Mapped/Swept Decomposition** can be used to guide the software to get more or less decomposition as desired.

6.1.2. Splitting free blocks

New methods have been added for splitting free blocks:

- The new **From Edges** method contains options for splitting a free block by constructing a single face from selected, planar edges or by creating independent sheet faces from loops of selected edges.
- The new **Blend across loops** method contains options for splitting a free block by sweeping a face or projecting a loop of edges. The desired output type (**Swept** or **Free**) determines the selection for creating the split (face or loop of edge, respectively).
- The **From Sheets** option is the legacy option that requires you to create sheet blocks first, and then use those sheets to split the block.

6.1.3. Editing blocks

New options have been added to simplify the process and expand the options for working with mapped, swept and free blocks:

- A **Convert free block face to mapped** option has been added to the **Edit Block** → **Convert Block Type** menu. This can be used to convert side faces on a **Free** block to mapped in preparation for converting the **Free** block to **Swept** (all side faces must be mapped for the conversion to be successful).
- A new method has been added to the **Edit Block** → **Merge Faces** menu. Use the **Block Faces** option to merge two or more faces forming a closed set on 3D free blocks by direct selection.
- **Merge vertices** has been improved in the following ways:
 - A single, hanging vertex on an edge between two free faces may be removed by selecting only that vertex.
 - Tools for swept blocks and free blocks have been added and made more robust.
 - A new **Update associations** option will group curves or adjust adjacent edge associations when vertices are merged.
 - A new **Update edge bunching** option will add the edge node count of the removed edge to the tangential neighbor edge(s).
- Usability and robustness improvements in **Imprint Face**.
- Improvements in displaying **Projected Edge Shape**.
- Improvements in **Blank Blocks** display.
- Improved associate edge or face to reference mesh.

6.1.4. User interface

Available tabs in the user interface are based on user preference rather than licensing. See the **Settings** → **Tools** menu, which has replaced the **Settings** → **Product** menu.

6.1.5. Output interfaces

Additional improvements have been made:

- The output interface for Exodus II format has been updated to allow larger mesh export (> 40 million cells).

6.2. Documentation

All documentation for **ANSYS ICEM CFD Release 17.0** is accessible using the Help menu. Visit the [ANSYS ICEM CFD website](#) for more information.

6.2.1. 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 7: Fluent Meshing Release Notes

The following sections contain release information for ANSYS Fluent Meshing Release 17.0:

[7.1.Changes in Product Behavior from Previous Releases](#)

[7.2.New Features](#)

7.1.Changes in Product Behavior from Previous Releases

- The **Select File** dialog box now dynamically lists files matching a partial file name rather than using the * wildcard character.
- The mesh check will issue a warning if multiple cell zones are maintained across an `interior` boundary. The boundary type in such cases should be set to `internal` instead.
- The functionality for the Boundary Wrapper is available through text commands only.
- The **Auto Mesh** feature now lets you choose to **Merge Cell Zones** or keep cell zones separate when doing object-based volume meshing.

Enabling **Merge Cell Zones** when meshing multiple closed regions will result in the prism cells being merged with tet cell zones for each meshed region.

- When merging volumetric regions, shared face zones will be deleted. However, if there are cell zones associated with the regions, then merging the regions will not delete the shared face zones. In this case, the shared face zones will be deleted when the cell zones are deleted.
- In object-based workflows, merging cell zones requires that they be in the same volumetric region.

To merge cell zones that cannot be in the same volumetric region because they are not contiguous, you will need to first delete the object(s) only, and then use the **Manage Cell Zones** dialog box.

- The options for prism post improvement now accept multiple zones as input.
- The bounded mesh will always be redisplayed after cell modification operations.
- Options previously available in the **Caps** category in the **Controls** dialog box are now available in the **Patch Options** group in the ribbon.

7.2.New Features

The new features available in the meshing mode in Fluent include changes to the object-based workflow. There are also enhancements to many existing features, and improved robustness through defect fixes.

Object Based Meshing Workflow

The following enhancements have been made:

- An option is available for creating labels for unlabeled zones of geometry objects. You can create labels based on the geometry object names or specify a label of your choice.

- Selective volumetric regions can be meshed using the **Auto Fill Volume** option. You can create a tetrahedral, hexcore, or hybrid (tetrahedra and prism or hexcore and prism) mesh in each region.
- An **Update** option is available to recompute Volumetric regions while preserving the region names or types.
- A backup is created for the mesh object surface mesh before volume meshing. The **Restore Faces** option can be used to restore the surface mesh when required.
- An interactive feature to extract the edges of selected face zones or surfaces and creates a edge zone in the selected object, is available as an onscreen tool button.
- The patching tools contain additional options for object/zone granularity and type.
- A new option to create **Polyhedral** cells is now available for **Volume Fill** in the **Auto Mesh** dialog box. This is available for object-based meshing, requires triangular surface mesh, and supports scoped prisms.

Note

When switching a polyhedral mesh to solution mode, a message regarding options for the use of node weights in post processing is presented.

User Experience

The following enhancements have been made:

- The user interface contains a ribbon with options to manage the display and select objects or zones.
- Toolbars can be enabled/disabled and positioned in the interface as required.
- The **Highlight** option enables the highlighting of objects, face zone labels, volumetric regions, or cell zones selected in the tree and the graphics window.
- The **Projection** toolbar enables you to switch between a perspective view of the graphics (default) and an orthographic view.

CAD Import

The following enhancements have been made:

- A new command enables the removal of the path prefix from object/zone names on import.

Miscellaneous

The following enhancements have been made:

- The command for separating face zones by cell neighbors also supports separation of interior face zones.
- The collapse and merge operations for boundary mesh modification can now be undone using the **Undo** option or a hot-key.
- Query functions are now available for obtaining:
 - lists of all unreferenced zones or unreferenced zones based on a filter string
 - lists of regions for an object or for a list of face zones

- the list of face zones for a list of labels specified
- lists of labels for an object based on a filter string, or for a list of face zones

Part V: ANSYS Simulation Products

Release notes are available for the following ANSYS Simulation products:

[Workbench \(p. 111\)](#)

[ANSYS Customization Toolkit \(ACT\) \(p. 117\)](#)

[RSM \(p. 119\)](#)

[EKM \(p. 121\)](#)

[DesignXplorer \(p. 125\)](#)

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
- 1.2. External Connection
- 1.3. Engineering Data Workspace
- 1.4. External Data
- 1.5. External Model
- 1.6. Enhancement to Mechanical Model Cells
- 1.7. FE Modeler
- 1.8. System Coupling
- 1.9. TurboSystem Release Notes
- 1.10. CFD-Post in Workbench

1.1. ANSYS Workbench

Enhancements have been made to the following areas:

- 1.1.1. ANSYS Workbench-Remote Solve Manager Enhancements
- 1.1.2. ANSYS Workbench-EKM Enhancements

1.1.1. ANSYS Workbench-Remote Solve Manager Enhancements

RSM Job Monitor in Workbench

A new **Job Monitor** in Workbench enables you to monitor the progress of project updates and solutions that you have submitted to Remote Solve Manager. The **Job Monitor** can be easily launched from the new **Jobs** menu, or from a button on the status bar. For more information, see [Monitoring and Controlling RSM Jobs in Workbench](#) in the *Workbench User's Guide*.

Icepak Job Submission to RSM

You can now submit Icepak solutions and updates to RSM from Workbench. See [Submitting Icepak Jobs to RSM](#) in the *Workbench User's Guide*.

1.1.2. ANSYS Workbench-EKM Enhancements

Improved EKM Connection Process

When no EKM connections exist, Workbench will launch the connection wizard automatically, resulting in a more streamlined connection process. See [Creating a Connection to an EKM Portal](#) in the *Workbench User's Guide*.

New EKM Connection Manager

A new **Manage Connections** feature provides a centralized system for creating, opening, closing and deleting EKM Portal connections. See [Working with Existing EKM Connections](#) in the *Workbench User's Guide*.

1.2. External Connection

New and Updated Tables

In the Workbench External Connection Add-In guide, the following changes have been made to the appendices:

- The tables in [Appendix A. ANSYS Workbench Component Inputs and Outputs](#) and [Appendix C: Data Transfer Types](#) have been updated.
- Two new appendices, [Appendix B. ANSYS Workbench Internally Defined System Template and Component Names](#) and [Appendix D. Addin Data Types and Data Transfer Formats](#), have been added.

1.3. Engineering Data Workspace

The following enhancements have been made to Engineering Data:

- The following new material models are available for Static Structural and Transient Structural analyses:
 - [Gurson Material Model](#)
 - [Cam-Clay](#)
 - [Drucker-Prager](#)
 - [Jointed Rock](#)
 - [Mohr-Coulomb](#)
 - [Porous Elasticity](#)
- You can now [create customized material models](#) in conjunction with the Material User Programmable Feature (UPF) of the Mechanical ADPL programming interface.
- The **Continuum Damage Mechanics Method** option is now available for the **Damage Evolution Law** material property for [Static Structural & Transient Structural analyses](#).
- The **Table Pane** now provides a context menu option to import comma-separated values (.csv) files to populate tabular data.
- For the Secant Coefficient of Thermal Expansion, the property field named "Reference Temperature" has been changed to "Zero-Thermal-Strain Reference Temperature."
- The ANSYS Workbench preferences (via **Tools>Options**) now provide an Engineering Data option. This option enables you to filter **Toolbox** content based on your selection in the **Properties** pane.

- You can now link Engineering Data to an [External Model](#) Component system. In addition, a consolidation feature is now available that enables you to combine multiple materials that contain the same data and come from the same source into a single material.
- You can now link the [External Model](#) Component system to the Engineering Data workspace in order to import material data from Mechanical APDL common database (.cdb), ABAQUS Input, or NASTRAN Bulk Data files.
- The way in which [Field Variables](#) are defined has changed. System provided field variables (Temperature, Shear Angle, and Degradation Factor) are now incorporated through a Toolbox option.
- You can now create a User-Defined [Field Variables](#).
- You can now assign colors to materials in the Engineering Data Workspace that will be visible in Mechanical.

1.4. External Data

No enhancements have been made to the External Data add-in.

1.5. External Model

For the 17.0 release, the following new features are now available in the External Model system:

- The External Model component's Process Coordinate Systems property enables the External Model system to import any Cartesian and Cylindrical coordinate systems defined in mesh files. This data is transferred to downstream Mechanical systems.
- From the External Model system, you can now choose to import node- and/or element-based components from solid and shell finite element mesh files to Mechanical as Named Selections. You can also specify components keys in order to filter the components contained in your mesh file. The following new properties are available for the External Model component system to support these new capabilities.
 - Process Nodal Components
 - Nodal Component Key
 - Process Element Components
 - Element Component Key

1.6. Enhancement to Mechanical Model Cells

You can use an **External Data** component to import a circuit board's trace layout file (TGZ, Ansoft ANF, or Cadence BRD, MCMP, or SIP) for use by a Mechanical system's Model cell. The **External Data** component also enables you to perform rigid transformations to the trace file.

1.7. FE Modeler

Release 17.0 for FE Modeler has no new features or enhancements.

1.8. System Coupling

The following enhancements were made to System Coupling for Release 17.0:

- CFX can now be a participant in a System Coupling analysis. For details, see [Coupling CFX to an External Solver: System Coupling Simulations in the CFX-Solver Modeling Guide](#).

The following enhancements were made to System Coupling for Release 16.2:

- In Fluent, a System Coupling interface can now be used in a sliding mesh zone or rigid body zone and the displacements can be applied correctly. See [Displacement transferred from System Coupling to a Sliding Mesh Zone](#) for further details.

1.9. TurboSystem Release Notes

TurboSystem is a set of software applications and software features that help you to perform turbomachinery analyses in ANSYS Workbench. For details, see [TurboSystem Introduction in the TurboSystem User's Guide](#).

These release notes cover:

- Turbo Setup
- Vista AFD, Vista CCD, Vista CPD and Vista RTD
- Vista TF

These release notes do not cover:

- ANSYS BladeModeler (see [ANSYS BladeModeler Release Notes](#))
- TurboGrid (see [TurboGrid Release Notes](#))
- CFX-Pre (see [CFX Release Notes](#))
- CFD-Post (see [CFD-Post Release Notes](#))

Note

After reviewing the TurboSystem release notes, you are encouraged to see [Usage Notes](#), which describes some known TurboSystem workflow issues and recommended practices for overcoming these issues.

1.9.1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the [ANSYS website](#).

1.9.2. New Features and Enhancements

This section lists features and enhancements that are new in Release 17.0 of TurboSystem.

- The Turbo Setup system now supports the specification of shrouded/unshrouded blades. For unshrouded blades, the system also now supports the specification of a tip gap.

- The Turbo Setup system now supports performance maps for centrifugal compressors.

1.10. CFD-Post in Workbench

The format/style of CFD-Post reports produced from Workbench are consistent with general Workbench-produced reports; reports from standalone CFD-Post continue to use the standard CFD-Post format.

Chapter 2: ANSYS ACT

The following enhancements have been added to ANSYS ACT 17.0:

Redesigned and Improved ACT Console

The **ACT Console** tool has been updated with a new UI, enhanced functionality, and improved access. It can now be accessed from the Workbench or AIM Extensions menu, the ACT Start Page, and a toolbar icon in Mechanical. The following new and/or enhanced functionality is available:

- Feature-rich command line editor
- Command history window with display-control icons
- "Smart" auto-completion
- Robust bookmarking capabilities, including bookmark import and export
- Extensive collection of keyboard shortcuts

For more information, see [ACT Console](#).

New ACT Start Page

The new **ACT Start Page** provides a common entry point to ACT's customization functionality. Accessible from multiple applications (Workbench, AIM, SpaceClaim, and Electronics Desktop), it provides access to the redesigned Extension Manager, the new Wizards page, the ACT Console, and the extensions Log.

For more information, see [Using the ACT Start Page](#).

Redesigned Extension Manager

The redesigned **Extension Manager**, accessed via the ACT Start Page, enables you to perform extension-related tasks such as defining extension directories, building binary extensions, installing/uninstalling extensions, loading/unloading extensions, and searching installed extensions. It also provides a link for convenient access to the ACT Application Library.

For more information, see [Using the Extension Manager](#).

New Wizards Page

The new **Wizards** page can be accessed via either the ACT Start Page or the Open Wizard toolbar button in DesignModeler or Mechanical. It provides a listing of all the guided process wizards that are loaded for the current context (i.e. the application in which the wizard will be executed). You can view information on or execute wizards from this list.

For more information, see [Launching a Wizard from the Wizards Page](#).

New Task-Level Capabilities for ACT Custom Workflows

ACT custom workflows in Workbench offer the following new task-level capabilities:

- **Task Reuse:**

Once defined and added to the Workbench Toolbox, an ACT-defined taskgroup can be added to the Project Schematic workflow in the same way as an installed (Workbench-defined) system.

Within taskgroup definitions, a new **external** attribute enables you to reference both the ACT custom tasks defined in your extension and "external" tasks (Workbench-defined components).

- **Task-Level RSM Support:**

ACT enables you to leverage RSM capabilities, specifying the remote execution of a task's processes.

Within task definitions, you can use `<rsmJob>` child nodes to specify RSM-related information, including supported platforms, input/output files, process arguments, and the callbacks needed to execute the remote jobs.

For more information, see [Defining a Task](#) and [Defining a Taskgroup](#) in [Capabilities for Custom ACT Workflows in Workbench](#) section of the ACT Developer's Guide.

SpaceClaim and Electronics Desktop Wizards

ACT target-application wizards are now supported for both ANSYS SpaceClaim and ANSYS Electronics Desktop, enabling you to take advantage of ACT's process-compression and automation capabilities in your modeling and your electronics simulations.

For more information, see [Guided Processes](#) in the ACT Developer's Guide. For examples, see [SpaceClaim Wizard](#) or [Electronics Desktop Wizard](#).

Use Project Wizard to Automate AIM

You can use a Workbench Project wizard to automate a step-by-step process in AIM. By executing the Project wizard from the AIM Project tab, you take advantage of ACT's process-compression and automation capabilities to execute a step-by-step through a simulation in AIM.

For more information, see [Guided Processes](#) and [Project Wizard \(AIM Project Tab\)](#) in the [ACT Developer's Guide](#).

New ANSYS ACT XML Reference Guide

A new ANSYS ACT XML Reference Guide provides information on the ACT XML definition. It is available under "Customization" in the installed ANSYS Documentation.

Chapter 3: Remote Solve Manager (RSM)

In this release of Remote Solve Manager, changes and enhancements include:

[3.1. New Features and Enhancements](#)

[3.2. Resolved Issues and Limitations](#)

3.1. New Features and Enhancements

- Custom cluster integration has been extended to allow the customization of job status and job ID parsing for all supported clusters. This makes it possible to customize unsupported clusters as well, without having to modify the RSM code.
- In the job list view, a new context menu on column headers contains the following items:
 - Select All
 - Home
 - End
 - Item Numbering
- The transfer speed has been improved for file transfers made through a user proxy.
- When running cluster jobs in a scratch directory local to the execution node in a Windows HPC cluster, there is no longer a constraint to make the local scratch directory the same as the Shared Cluster Directory. In the **Compute Server Properties** dialog box, a new **Share Path for Local Scratch** setting on the **File Management** tab enables you to specify a UNC path to any shared directory.

3.2. Resolved Issues and Limitations

- There are no known issues at this time.

Chapter 4: ANSYS EKM Release Notes

ANSYS Engineering Knowledge Manager (EKM) 17.0 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 17.0.

4.1. New Features and Enhancements

If you have used previous versions of EKM, Release 17.0 offers many significant changes and improvements:

Installation and Configuration

- Each EKM server is now installed in cluster configuration. Cluster configuration affords the flexibility to scale EKM to meet changing workloads. An EKM server can be installed as single-node or multi-node, and additional EKM cluster nodes may be added as needed.
- EKM now features centralized server installation and configuration. This simplifies EKM configuration and management, and reduces duplication of files and configuration in a multi-node server.
- EKM now supports the next-generation `mod_cluster` load balancer for integration with a front-end webserver. `mod_cluster` replaces `mod_jk`, providing wider protocol support (HTTP, HTTPS, AJP), more flexible load balancing options and improved management.
- A new tiered licensing system enables you to better control how individual users access and modify repository data.
- EKM workspaces are no longer categorized as Individual or Shared. A user's access to data in a workspace is determined by the **Access Level** assigned to the user in their profile, and the type of license that the user is holding when he or she accesses the workspace.
- A new `clusterVisibleDirectories` setting in the `ekm.xml` file enables you to specify the directories that are visible to all compute cluster nodes. This determines where the job data directory can reside, and ensures that RSM jobs run directly in their specified working directory, without any unnecessary file transfers.
- When a template, application, action or macro fails, administrators can view a script log that enables them to debug scripts in that item.
- When defining type attributes for a custom type, you can use the new `customStatusFlagsMacro` to specify the name of a macro that will display custom status flags and icons when an object of that type is displayed or queried in the list view.
- When defining actions for a custom type, you can remove selected built-in actions to prevent them from appearing on an object's context menu or toolbar.
- The `supportedVersions` variable has been removed from the built-in CFX, Fluent and MAPDL job templates. Solver versions are now specified globally in a new `solverVersions` element in the `ekm.xml` file.

- The way in which you configure RSM when integrating it with EKM has changed. Key changes include:
 - The Compute Server service (Ans.Rsm.SHHost.exe) and RSM-RPC Server (Ans.Rsm.XmlRpcServer) are no longer used. Only the RSM Job Manager service (Ans.Rsm.JMHost.exe) is required.
 - All configuration is done through XML configuration files instead of the RSM Admin interface. This includes configuring RSM to integrate with a cluster, and defining RSM queues.
 - EKM no longer caches passwords with RSM. Your EKM credentials are automatically sent to the RSM Launcher service when RSM is needed.
 - A new **Job Management** preference enables you to specify different job execution credentials if needed.

For more information, refer to [Integrating EKM with Remote Solve Manager \(RSM\)](#) in the *EKM Administration Guide*.

- The EKM Connector End User API has been updated.

Data Management

- The **Copy** action now copies selected files and folders to the clipboard. A new **Paste** action enables you to quickly complete the copy operation. The **Move** action has been replaced by **Cut** and **Paste** actions.
- Simulation details are now extracted from `.rstp` files (Mechanical APDL results file format), as well as `.cas.h5` files (Fluent HDF).
- Simulation details are now extracted from ANSYS AIM projects, and the resulting simulation details report includes AIM Study properties.
- Icepak `.ice.xml` and `.tzz` files have been added as built-in types in EKM, and simulation details are extracted from them.
- SpaceClaim `.scdoc` files have been added as a built-in type in EKM. When you upload such a file to the repository, you can view the SpaceClaim model in the interactive 3D viewer.
- The 3D viewer has been enhanced to include additional zoom/pan/rotate functionality as well as a scene selector for files that support multiple scenes.
- A new `disableDataExtraction` setting enables you control whether or not metadata is extracted from specific file types when they are uploaded to EKM.
- A new **Transfer Details** button in the **Download From EKM** dialog box calculates and displays the total download size of the files that you have selected for download.
- A new **Views** tab in the **Custom Type** dialog box enables you to define custom views for objects.
- When defining a custom interface, you can now include images that reside in the EKM repository, as well as links to repository objects.
- A more robust 3D viewer has been integrated into the **Image** tab. You can use keyboard shortcuts to display the model in various predefined rotations.
- You can now download a report as a Word document.
- The option to re-extract data is now available if you cancel an extraction on the **Extraction Monitor** page.

- A new **Connections** feature enables you to capture your sign-in credentials for other EKM repositories. These predefined connections can be used to quickly expand a search or access another repository through a URL object.

Job Management

- When starting an interactive session, the session status is now displayed in the **Job Monitor** instead of a dialog box.
- If you encounter issues during an interactive session, you can use the new **Show session log** action to view a log of activity from EnginFrame, including VNC server information and task outputs and errors.
- When using the built-in **Electronics** job template, you can now select the version of Electronics Desktop that you want to use. When adding **Batchoptions** during job setup, a new **Show registry key entries** drop box enables you to specify that you want to display all or specific registry key categories. A **Display only frequently used** check box enables you to control whether only frequently used entries or all entries are displayed for the selected category.
- Legacy projects created in Simplorer, Maxwell, HFSS, Designer and Q3D will be converted to .aedt files and submitted to ANSYS Electronics Desktop for execution.
- The Fluent job template now supports Fluent HDF (.cas.h5 and .dat.h5) files.
- When the **Show RSM job log** action is selected, EKM now retrieves a full, pre-formatted job report from RSM.
- In the **Edit Job Template** dialog box, a new **Job status view** option on the **UI** tab enables you display advanced monitoring and controls when a selected solver is launched in server (aas) mode.
- When running a Mechanical APDL server-mode job, the **Job Monitor** now shows the progress of individual solution steps.
- When updating a Workbench project, you no longer need to specify the component execution mode. EKM uses the server-mode startup options provided by Workbench. Also, the job status refreshes automatically as the job moves from *Queued* to *Executing*, eliminating the need for a manual refresh.

Process Management

- A new **Multi-line value** option enables you to create multi-line text fields in the following contexts:
 - When defining a text box in a custom dialog box
 - When defining properties for a custom type
 - When defining a variable for a process template in EKM Studio
- Upon selecting a process template that has not yet been approved, you are now presented with the option to request approval at that time.
- The **Edit > Alerts** action can now be captured in a journal.
- When defining a custom dialog node in a process template, you can now incorporate an **On-change macro** for Text Box, Check Box, Reference, Date, and Dropdown widgets.
- Delete permission has been added to approved process templates.

- When defining a batch node in a process template, a new **Show working directory** option lets you control whether or not users can access the job's working directory.
- Administrators can use the scripting interface to test or debug a process template. APIs have been added for this purpose.

Usability Enhancements

- A new **Edit Gadget** action enables you to edit the name of a dashboard gadget, or what it displays.
- The object view window can now be maximized and restored down as needed.
- Check boxes have been added to the design points table for Workbench server projects to facilitate the selection of design points.
- The majority of toolbar buttons show only updated icons, giving the interface a cleaner look.

4.2. Issues Resolved in this Release

Below are the major issues that have been resolved since the release of 16.2.

- If a newly created workspace fails to initialize, users are no longer prevented from signing in to EKM.
- In a custom dialog box, using `ekm.raiseException` in a macro that is called from an ajax button will no longer produce a blank exception page.
- When using the **Parallel** processing option in a Fluent batch job, clicking the down arrow of the **Number of processes** field now displays a correct value in the field.

4.3. Issues and Limitations

All issues and limitations known at the time of release are listed in [Appendix B](#) in the *EKM User's Guide*.

Chapter 5: DesignXplorer

5.1. Enhancements

The following enhancements have been added to ANSYS DesignXplorer 17.0:

Export Response Surfaces as DX-ROM

You can export a solved DesignXplorer response surface as a lightweight, independent reduced-order model either in the DX-ROM format (.dxrom file extension) or as an FMU (.fmu file extension). It can then be used in software that implements the Functional Mock-up Interface (FMI), such as ANSYS Simpler and Mathworks MATLAB®.

For more information, see [Exporting Response Surfaces](#).

DX-ROM Response Surface Readers Package

The Response Surface Readers package delivers DesignXplorer's custom DX-ROM tools. If you've exported a DesignXplorer response surface as a .dxrom file, these tools enable you to incorporate it into your simulation. The following three tools are included:

- Response Surface Reader System: ACT extension exposing a new Design Exploration system template in the Workbench Toolbox
- Excel DX-ROM Add-in: Exposed as an ANSYS DESIGNXPLOER tab in the Excel application
- DX-ROM Postprocessing Utility: Lightweight tool to evaluate your exported DX-ROM

The package and related documentation are available in the [ANSYS ACT Application Store](#) on the [ANSYS Customer Portal](#).

For more information on DX-ROM, see [Exporting Response Surfaces](#).

Genetic Aggregation as Default Response Surface Type

Genetic Aggregation is now the default response surface type.

For more information, see [Genetic Aggregation](#).

Genetic Aggregation Performance Enhancements

Enhancements to the Genetic Aggregation response surface provide improved reliability and reduced computing time.

For more information, see [Genetic Aggregation](#).

Updated MATLAB Optimization Extension

The ACT MATLAB optimization extension has been updated for use with the ANSYS Workbench 17.0 release. The extension and related documentation are available in the [ANSYS ACT Application Store](#) on the [ANSYS Customer Portal](#).

5.2. Resolved Defects and Limitations

The following issues and limitations have been resolved:

- After an interrupt/abort operation, you can successfully re-update a MOGA or AMO optimization without needing to change optimization settings.
- For input parameters using Discrete or Manufacturable values, the parameter Starting Value (for MISQP) and Lower Bound/Upper Bound properties remain editable in the Optimization component.
- Optimization objectives defined on input parameters are now handled as expected in charts. For a study containing only objectives defined on input parameters, the Samples and Trade-off charts can now be generated. For a study combining objectives and constraints on both input and output parameters, the input parameters are now correctly taken into account when calculating the Pareto fronts.

Part VI: ANSYS AIM

The following enhancements are available in ANSYS, Inc. Release 17.0 (ANSYS AIM). Accessible via the Help Viewer in the product and online via the ANSYS Customer Portal, the release notes are intended to provide an overview of the product. Enhancements published in the Release 16.1 and Release 16.2 release notes are included for reference.

Chapter 1: Advisories

In addition to any incompatibilities noted within the release notes, known non-operational behavior, errors and/or limitations at the time of release are documented in the *ANSYS, Inc Known Issues and Limitations* document, accessible via the ANSYS Customer Portal (account required). The AIM download page is not publicly-accessible. First-time users of the customer portal must register to create a password. See the ANSYS Customer Portal for information about ANSYS service packs, **Customization**, and any additional items not included in the *Known Issues and Limitations* document.

Chapter 2: Enhancements in AIM 17.0

The following enhancements have been made to ANSYS AIM for Release 17.0.

- The ability to create and edit geometry directly within the AIM Study.
 - Based on the ANSYS SpaceClaim Direct Modeler.
 - Includes enhanced performance for geometry updates and automated parameter creation.
- The ability to manage, update and review design points directly from within the AIM Study using Design Point Dashboard.
- Enhanced graphics display renders bodies based on material appearance.
 - Material library enhanced to include appearance settings for many common materials.
- The ability to model strain-life based (low cycle) fatigue.
- The ability to apply force loads remotely on model topology or a construction point.
- The ability to automatically generate region interfaces for conjugate heat transfer based on surface proximity.
- Enhanced boundary conditions for compressible flow including a far field boundary condition to model free-stream compressible flow at infinity.
- Enhanced solution monitoring including multiple convergence and residual monitors, and the ability to review multiple charts during the solution update.
- The ability to interrupt, interrogate results and continue the solution from an interrupted state for all physics.
- Enhanced documentation including field level help for any property.
- The ability to connect to the simulation portal (EKM) from within the AIM Study.
- Enhanced volumetric mapping including radial basis function mapping for volumetric temperature data.
- The ability to automate a simulation in AIM by executing a Workbench Project wizard on the AIM project tab.
- Enhanced custom simulation process templates including the ability to include field level help, radio and multi-select lists as part of a custom template.
- Improved fluid initial conditions - consistent with boundary conditions, more granular.

Chapter 3: Enhancements in AIM 16.2

The following enhancements were made to ANSYS AIM for Release 16.2.

- The ability to model conjugate heat transfer including the flexibility to use different physics models in different regions of the same model.
 - The ability to specify region interfaces, which allow connections between fluid and solid regions (and within a region), with support for both conformal and non-conformal mesh connections.
 - The ability to specify temperature dependent material properties (thermal conductivity and specific heat) for solid materials for conjugate heat transfer models.
- The ability to model compressible gas flows across the subsonic, transonic and supersonic range including a variable gas density using the Ideal Gas Law as well as temperature dependent specific heat, dynamic viscosity and thermal conductivity.
- The ability to model buoyancy-driven flows for both incompressible and compressible fluids using the Boussinesq model for incompressible fluids, and the Density difference model for compressible gases.
- Fluid solver discretization changes:
 - Peclet blending of outflow face values, improves convergence of the energy and turbulence equation when the flow is tangential to the boundary surface.
- Fluid solver algorithmic changes:
 - Default time scale factor for the energy equation has been increased to 10.
 - Coupled multi-grid linear solver coarsening heuristic has been improved.
 - Reverse Cuthill-McKee reordering is applied prior to solving the linear system.
 - Solver transcript has been restructured to aid in judging convergence.
- Enhanced one-way fluid-structure interaction including both profile preserving volumetric temperature mapping and conservative surface force mapping.
- The ability to model nonlinear surface-to-surface contacts with options for frictionless, frictional, rough, bonded and no-separation contact behavior.
- Automatic nonlinear structural solution control with automatic determination if a large deflection solution is required.
- The ability to model stress-life based (high cycle) fatigue including an option to determine a simplified S-N curve from the material ultimate strength.
- Enhanced lighting and shading for all display options including geometry, mesh and solution results.

- The ability to interrupt the execution of a mesh generation or solver execution including the ability to post-process a partially updated solution.
- The ability to create custom guided process extensions using ANSYS ACT, which can be added as custom templates to ANSYS AIM.

Chapter 4: Enhancements in AIM 16.1

The following enhancements were made to ANSYS AIM for Release 16.1.

- Immersive editing has been enhanced to include the ability to select and/or create selection sets using the mini data panel while working in the graphics window. This allows easy access to selection sets when working immersively for defining mesh controls, boundary conditions and/or post-processing objects.
- Structural boundary conditions (force, moment, and displacement) have been enhanced to allow specification using magnitude and direction. This capability aligns the boundary condition with a geometric surface or an edge, which simplifies the specification of boundary conditions versus specifying vector components.
- Interface conditions now include structural joints to model a variety of kinematic conditions. Fixed, hinge, translational, slot, cylindrical, universal, spherical, planar, and general joint behaviors are available to model a variety of kinematic conditions.
- The results legend has been enhanced to include a user specified variable range, and an option for a logarithmic scale.

Chapter 5: Limitations

The AIM product documentation details limitations to the Volume Creation task and Geometry Modeling task.

In addition, the *Known Issues and Limitations* document is accessible via the ANSYS Customer Portal (account required). Via Knowledge Resources> Online Documentation, open the General section to view the current *Known Issues and Limitations* document. First-time users of the customer portal must register to create a password.

