Copyright and Trademark Information

© 2017 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT 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 located 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. FLEMIm and FLEXnet are trademarks of Flexera Software LLC.

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. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2008 companies.

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 ......................................................................................................................................................... vii

1. Advisories ................................................................................................................................................ viii
2. Compatibility with Previous Releases ...................................................................................................... ix
3. Installation ................................................................................................................................................ ix
4. Licensing .................................................................................................................................................. ix
5. Documentation ....................................................................................................................................... ix
6. Verification Manual ................................................................................................................................ xi
   6.1. Mechanical APDL ................................................................................................................................ x
   6.1.1. New Verification Test Cases .......................................................................................................... x
7. Online Video Access .................................................................................................................................. x
8. ANSYS Customer Portal ........................................................................................................................ xi
9. ANSYS Elastic Licensing ........................................................................................................................ xi
10. New Product Levels ................................................................................................................................ xi

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. Graphics Enhancements ................................................................................................................... 4
   1.4. Geometry Enhancements .................................................................................................................. 5
   1.5. Model Assembly and External Model Enhancements ...................................................................... 5
   1.6. Contact and Connection Enhancements .......................................................................................... 5
   1.7. Mesh Enhancements ......................................................................................................................... 6
   1.8. Linear Dynamics Enhancements ...................................................................................................... 6
   1.9. Topology Optimization Enhancements ............................................................................................ 6
   1.10. Loads/Supports/Conditions Enhancements ..................................................................................... 7
   1.11. Mapping Enhancements .................................................................................................................. 7
   1.12. Solution Enhancements .................................................................................................................. 7
   1.13. Rigid Body Solver Enhancements ................................................................................................... 7
   1.14. Explicit Dynamics Enhancements .................................................................................................. 8
   1.15. Results Enhancements .................................................................................................................... 8

2. Mechanical APDL ...................................................................................................................................... 9
   2.1. Structural ......................................................................................................................................... 9
      2.1.1. Contact ..................................................................................................................................... 9
         2.1.1.1. Beam-to-Beam Contact ...................................................................................................... 9
         2.1.1.2. Performance Improvements for Large Contact Models .................................................... 10
         2.1.1.3. Modeling Interface Damping in Assembled Structures ....................................................... 10
      2.1.2. Linear Dynamics ...................................................................................................................... 10
         2.1.2.1. Component Mode Synthesis (CMS) Enhancements ............................................................ 10
         2.1.2.2. MPRS for FSI .................................................................................................................... 10
   2.2. Solvers .............................................................................................................................................. 10
      2.2.1. Distributed ANSYS Enhancements .......................................................................................... 11
      2.2.2. GPU Acceleration Enhancements ............................................................................................. 11
   2.3. Results File ...................................................................................................................................... 11
   2.4. Commands ...................................................................................................................................... 11
      2.4.1. New Commands ........................................................................................................................ 12
      2.4.2. Modified Commands ................................................................................................................ 12
      2.4.3. Undocumented Commands ..................................................................................................... 12
   2.5. Elements ......................................................................................................................................... 13
      2.5.1. Modified Elements .................................................................................................................. 13
      2.5.2. Undocumented Elements ........................................................................................................ 13
2.6. Documentation .......................................................................................................................... 13
  2.6.1. Technology Demonstration Guide .......................................................................................... 13
2.7. ANSYS Product Improvement Program .................................................................................... 13
2.8. Known Limitations .................................................................................................................... 14

3. Autodyn ......................................................................................................................................... 15
  3.1. New Features and Enhancements .............................................................................................. 15

4. Aqwa ............................................................................................................................................ 17
  4.1. Aqwa Solver Modules ............................................................................................................... 17
    4.1.1. Wind Spectrum Seed ........................................................................................................... 17
    4.1.2. Frequency Dependent Added Mass and Damping ................................................................. 17
    4.1.3. Recalculate QTFs Based on User-Input RAOs ................................................................. 17
  4.2. Hydrodynamic Analysis Systems ............................................................................................ 17
    4.2.1. Deactivated Freedoms Object ........................................................................................... 17
    4.2.2. Morison Hull Drag Coefficients Object .......................................................................... 17
    4.2.3. Time Domain Statistics Distribution Function .................................................................. 18
    4.2.4. Generated Mesh Information ......................................................................................... 18
    4.2.5. Visualization of Splitting Force Bounding Box ................................................................. 18
    4.2.6. Fixed Points Container ................................................................................................... 18

5. ANSYS Composite PrepPost (ACP) ............................................................................................. 19
  5.1. New Features in ANSYS Composite PrepPost (ACP) 18.1 .................................................... 19
    5.1.1. Improved Tree View ......................................................................................................... 19
  5.2. Supported Platforms for ANSYS Composite PrepPost (ACP) 18.1 ........................................ 19
  5.3. Known Limitations and Incompatibilities .............................................................................. 19

II. ANSYS Fluids Products ............................................................................................................... 21
1. Fluent ........................................................................................................................................... 23
  1.1. New Features in ANSYS Fluent 18.1 ...................................................................................... 23
  1.2. Supported Platforms for ANSYS Fluent 18.1 ....................................................................... 26
  1.3. New Limitations in ANSYS Fluent 18.1 ................................................................................ 26
  1.4. Resolved Issues and Limitations ............................................................................................ 27
  1.5. Updates Affecting Code Behavior ......................................................................................... 29

2. CFX .............................................................................................................................................. 31
  2.1. Supported Platforms .............................................................................................................. 31
  2.2. New Features and Enhancements ......................................................................................... 31
  2.3. Resolved Issues and Limitations ........................................................................................... 31
  2.4. Updates Affecting Code Behavior .......................................................................................... 32

3. TurboGrid ..................................................................................................................................... 33
  3.1. Supported Platforms .............................................................................................................. 33
  3.2. New Features and Enhancements ........................................................................................... 33

4. BladeModeler ............................................................................................................................... 35
  4.1. Supported Platforms .............................................................................................................. 35

5. CFD-Post ..................................................................................................................................... 37
  5.1. Supported Platforms .............................................................................................................. 37
  5.2. New Features and Enhancements ........................................................................................... 37

6. Polyflow ....................................................................................................................................... 39
  6.1. New Features .......................................................................................................................... 39
  6.2. Supported Platforms .............................................................................................................. 39
  6.3. New Limitations in ANSYS Polyflow 18.1 .......................................................................... 40
  6.4. Past Versions of ANSYS Polyflow Release Notes ................................................................. 40

7. Forte .......................................................................................................................................... 41
  7.1. New Features and Enhancements .......................................................................................... 41
  7.2. Resolved Issues since Forte 18.0 ........................................................................................... 42
  7.3. Supported Platforms ............................................................................................................. 43
8. Chemkin-Pro .............................................................................................................. 45
  8.1. New Features and Enhancements .......................................................................... 45
  8.2. Resolved Issues since 18.0 ....................................................................................... 46
9. FENSAP-ICE .............................................................................................................. 47
  9.1. New Features and Enhancements in ANSYS FENSAP-ICE ................................. 47
  9.2. New Limitations in ANSYS FENSAP-ICE ............................................................... 50
  9.3. Resolved Issues and Limitations .......................................................................... 52
  9.4. Beta Features ........................................................................................................ 53
III. ANSYS Electronics Products ................................................................................ 55
  1. Icepak ....................................................................................................................... 57
    1.1. Introduction ........................................................................................................... 57
    1.2. New and Modified Features in ANSYS Icepak 18.1 .......................................... 57
    1.3. Resolved Issues and Limitations in ANSYS Icepak 18.1 ................................. 57
IV. ANSYS Geometry & Mesh Prep Products ................................................................ 61
  1. DesignModeler ........................................................................................................... 63
  2. SpaceClaim ............................................................................................................... 65
  3. CAD ............................................................................................................................ 67
  4. Meshing ..................................................................................................................... 69
    4.1. Changes in Interface Terminology and Product Behavior from Previous Releases 69
    4.2. Contact Enhancements ....................................................................................... 69
  5. IC Engine Release Notes .......................................................................................... 71
  6. ICEM CFD .................................................................................................................... 73
    6.1. Highlights of ANSYS ICEM CFD 18.1 ................................................................. 73
      6.1.1. Multizone Block Editing improvements ......................................................... 73
      6.1.2. Transfer Blocks ............................................................................................. 73
      6.1.3. Pipe Modeler ................................................................................................. 73
      6.1.4. Usability Improvements .............................................................................. 73
    6.2. Documentation ...................................................................................................... 74
      6.2.1. Tutorials ........................................................................................................ 74
    6.2.1. Tutorials ........................................................................................................ 74
7. Fluent Meshing ............................................................................................................ 75
  7.1. Changes in Product Behavior from Previous Releases ......................................... 75
  7.2. New Features ......................................................................................................... 75
V. ANSYS Simulation Products ...................................................................................... 77
  1. Workbench ................................................................................................................ 79
    1.1. ANSYS Workbench ............................................................................................. 79
      1.1.1. Mechanical APDL Enhancements ................................................................. 79
      1.1.2. ANSYS Workbench-Remote Solve Manager Enhancements ..................... 79
      1.1.3. ANSYS Workbench-EKM Enhancements ............................................... 79
    1.2. External Connection ............................................................................................ 79
    1.3. Engineering Data Workspace ............................................................................ 80
    1.4. External Data ....................................................................................................... 80
    1.5. External Model .................................................................................................... 80
    1.6. Enhancement to Mechanical Model Cells .......................................................... 80
    1.7. FE Modeler .......................................................................................................... 80
    1.8. System Coupling ................................................................................................. 80
    1.9. TurboSystem Release Notes ............................................................................. 80
      1.9.1. Supported Platforms .................................................................................... 81
  2. ACT ............................................................................................................................. 83
  3. Remote Solve Manager (RSM) ................................................................................ 87
    3.1. New Features and Enhancements ....................................................................... 87
    3.2. Deprecated Features .......................................................................................... 88
    3.3. Issues Resolved in this Release ......................................................................... 88
3.4. Known Issues and Limitations .............................................................................................. 88
4. **EKM** ........................................................................................................................................ 89
   4.1. New Features and Enhancements .......................................................................................... 89
   4.2. Issues Resolved in this Release ............................................................................................. 91
   4.3. Issues and Limitations .......................................................................................................... 91
5. **DesignXplorer** ...................................................................................................................... 93
6. **ANSYS Viewer** .................................................................................................................... 95
   6.1. New Features and Enhancements .......................................................................................... 95
   6.2. Known Issues and Limitations ............................................................................................ 95
VI. **ANSYS AIM** ....................................................................................................................... 97
   1. Advisories ................................................................................................................................ 99
   2. Enhancements in AIM 18.1 ...................................................................................................... 101
   3. Enhancements in AIM 18.0 ...................................................................................................... 103
   4. Enhancements in AIM 17.2 ...................................................................................................... 105
   5. Limitations ............................................................................................................................. 107
Global Release Notes

The release notes are specific to ANSYS, Inc. Release 18.1 and arranged by application/product, with the exception of:

- Advisories (p. viii)
- Compatibility with Previous Releases (p. ix)
- Installation (p. ix)
- Licensing (p. ix)
- Documentation (p. ix)
- Verification Manual (p. x)
- Online Video Access (p. x)
- ANSYS Customer Portal (p. xi)
- ANSYS Elastic Licensing (p. xi)
- New Product Levels (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 18.0
- ANSYS 17.2
- ANSYS 17.1
- ANSYS 17.0
- ANSYS 16.2
- ANSYS 16.1
- ANSYS 16.0

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
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.

At release 18.1, ANSYS ACT provides all Workbench External Connection Add-in functionality and the majority of Workbench Framework SDK coverage. Consequently, you should only use ACT for your automation and customization projects. ACT allows you to adopt the official, easy-to-use, consistent customization methodology for all ANSYS products. For the same reason, please also consider transitioning legacy ANSYS customization solutions to ACT.

Project names and paths should not include Japanese or Chinese characters. The restriction is applicable to most ANSYS, Inc. products, including the flagship products and Workbench add-ins. The restriction applies whether the user interface is localized or not.

Product Change Notification

As part of improvements being made, the ANSYS CFD-Viewer will be replaced by the ANSYS-Viewer in a future release.

A video describing the functionality of the ANSYS Viewer is available.

The ANSYS Viewer can be installed as part of the normal ANSYS product installation or can be downloaded from the ANSYS website.

From ANSYS CFX-Pre, CFD-Post, and TurboGrid you can write AVZ files from the File > Save Picture panel of those components by setting the "Format" to "AVZ (3D)".

Existing CFD-Viewer State (CVF) files previously generated from ANSYS CFD products can be converted by using an application supplied as part of the CFX, ANSYS TurboGrid and CFD-Post installations. For example: C:\Program Files\ANSYS Inc\v181\CFX\bin\cfx5cvfconvert <cvf-file>

This command will generate an ANSYS Viewer file (AVZ file) which can be loaded into the ANSYS-Viewer. The new file will be named based on the name of the CVF file but with the file extension replaced with "avz."

• 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
Note: Reports previously generated from ANSYS CFD-Post containing CVF files cannot be successfully converted at this time. This may be available in future releases.

2. Compatibility with Previous Releases

Backwards Compatibility: ANSYS 18.1 was tested to read and resume databases from the following previous versions: 17.0, 17.1, 17.2 and 18.0. Note that some products are able to read and resume databases from releases prior to 17.0. Please see the specific product sections below for more details. For those products that cannot directly read a 16.x database in 18.1, first resume it in 18.0 and then resume that database in 18.1.

Upward/Forward Compatibility: No previous release has the ability to read and resume a database from a more recent release.

3. Installation

The following features are new or changed at Release 18.1. Review these items carefully.

- ANSYS CFD-Post is now included with the FENSAP-ICE installation.

- Microsoft .NET Framework 4.6.2 is now required for running ANSYS products. If your installation computer does not have .NET 4.6.2 installed, the ANSYS installation program will install .NET 4.6.2 as a prerequisite and may require a system restart after exiting the installation and before running any ANSYS products.

- The ANSYS product installation program automatically verifies the existence of NVidia drivers on the installation computer and creates an ANSYS Workbench driver profile if necessary.

- ANSYS SpaceClaim Direct Modeler and ANSYS SpaceClaim can now be configured, post installation, by using the Product & CAD Configuration Manager or silent mode operations flags.

4. Licensing

The following enhancements were made to ANSYS, Inc. Licensing for Release 18.1:

- The Product Order File was re-ordered to reflect product changes and additions. The installation program should automatically upgrade the file for you, merging any changes that you have made.

- The current version of OpenSSL utilized with the ANSYS Licensing Interconnect is 1.1.0e.

- The current version of Java utilized by the ANSYS License Management Center is Java 8 Update 121.

5. Documentation

Online Documentation

We are debuting a beta release of a new online version of our documentation. With online documentation, you have access to the best and latest content, updated as soon as it is available. You also gain access to our help, tutorials, and videos in a single, convenient location, accessible from all your Internet-connected devices.

Your feedback is greatly appreciated as we continue to improve this new resource.
You can visit https://ansyshelp.ansys.com to access online documentation with a customer portal login, or you can set online documentation as the default source of help for your ANSYS programs. This allows you to go straight to online documentation from the program, without a login.

To set online documentation as the default source of help for your ANSYS programs, create a new environment variable with the following values:

Variable name: ANSYS_INTERNET_DOC

Variable value: https://ansyshelp.ansys.com/

Please consult the documentation for your operating system for instructions on how to create an environment variable.

**Help Viewer Enhancements**

Release 18.1 of the ANSYS Help Viewer has no new features or enhancements.

**Updated Product Documentation**

Visit the ANSYS Customer Portal Online Documentation page to view or search the latest updates to the Release 18.1 documentation.

Documentation is 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.

**6. Verification Manual**

Significant modifications and additions occurred in the Verification Manuals at 18.0. These changes provide greater coverage and accuracy in the verification of the ANSYS product suite.

The Verification Manuals for the following products were updated at 18.0:

6.1. Mechanical APDL

**6.1. Mechanical APDL**

The following sections outline the changes to the Mechanical APDL Verification Manual:

6.1.1. New Verification Test Cases

**6.1.1. New Verification Test Cases**

The following new VMs are available:

- VM298 - PSD Analysis of 40-Story Building Under Wind Load Excitation

**7. 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.
8. 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.

9. ANSYS Elastic Licensing

Pay-per-use licensing is offered in conjunction with traditional lease and paid-up licensing. ANSYS Elastic Licensing provides hourly-based access to all ANSYS products (with the exception of the Semiconductor applications and SCAD applications) through a single license called ANSYS Elastic Units.

Elastic Licensing was launched on the ANSYS Enterprise Cloud (AEC) in September 2016.

More information and announcements about ANSYS Elastic Licensing and the ANSYS Enterprise Cloud are available at ansys.com.

10. New Product Levels

The product level consolidation and upgrades introduced at Release 18.0 remain unchanged for Release 18.1. Contact your ANSYS Account Manager to move to one of these levels.

The following table outlines the capabilities of the new ANSYS CFD product levels. Each level provides single-task access to the listed applications. These new levels include access to four HPC cores with a GPU being counted as a single core. ANSYS Workbench access is included with all levels.

<table>
<thead>
<tr>
<th>Product Name</th>
<th>Capabilities</th>
</tr>
</thead>
<tbody>
<tr>
<td>ANSYS CFD Enterprise</td>
<td>This level includes everything in ANSYS CFD Premium and:</td>
</tr>
<tr>
<td></td>
<td>• ANSYS Polyflow</td>
</tr>
<tr>
<td></td>
<td>• ANSYS Forte</td>
</tr>
<tr>
<td></td>
<td>• ANSYS FENSAP-ICE</td>
</tr>
<tr>
<td></td>
<td>• ANSYS AIM Pro</td>
</tr>
<tr>
<td>Product Name</td>
<td>Capabilities</td>
</tr>
<tr>
<td>------------------------------------</td>
<td>------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>ANSYS CFD Enterprise Solver</td>
<td>Includes everything in ANSYS CFD Premium Solver and:</td>
</tr>
<tr>
<td></td>
<td>• ANSYS Polyflow Solver</td>
</tr>
<tr>
<td></td>
<td>• ANSYS Forte Solver</td>
</tr>
<tr>
<td></td>
<td>• ANSYS FENSAP-ICE Solver</td>
</tr>
<tr>
<td></td>
<td>• Can be used in combination with ANSYS CFD PrepPost to run AIM Pro</td>
</tr>
<tr>
<td>ANSYS CFD Premium</td>
<td>This level includes:</td>
</tr>
<tr>
<td></td>
<td>• ANSYS Fluent</td>
</tr>
<tr>
<td></td>
<td>• ANSYS CFX</td>
</tr>
<tr>
<td></td>
<td>• ANSYS Meshing (including ANSYS TurboGrid and ANSYS ICEM CFD)</td>
</tr>
<tr>
<td></td>
<td>• ANSYS OptiGrid</td>
</tr>
<tr>
<td></td>
<td>• ANSYS SpaceClaim Direct Modeler</td>
</tr>
<tr>
<td></td>
<td>• ANSYS CFD-Post</td>
</tr>
<tr>
<td></td>
<td>• 4 HPC</td>
</tr>
<tr>
<td>ANSYS CFD Premium Solver</td>
<td>This level includes:</td>
</tr>
<tr>
<td></td>
<td>• ANSYS Fluent Solver</td>
</tr>
<tr>
<td></td>
<td>• ANSYS CFX-Solver</td>
</tr>
<tr>
<td></td>
<td>• 4 HPC</td>
</tr>
<tr>
<td>ANSYS CFD PrepPost</td>
<td>Prior to Release 18.0, ANSYS CFD PrepPost included:</td>
</tr>
<tr>
<td></td>
<td>• ANSYS Meshing (including ANSYS TurboGrid and ANSYS ICEM CFD)</td>
</tr>
<tr>
<td></td>
<td>• ANSYS Fluent PrepPost</td>
</tr>
<tr>
<td></td>
<td>• ANSYS CFX-Pre</td>
</tr>
<tr>
<td></td>
<td>• ANSYS CFD-Post</td>
</tr>
</tbody>
</table>
## New Product Levels

<table>
<thead>
<tr>
<th>Product Name</th>
<th>Capabilities</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>• ANSYS Polyflow PrepPost</td>
</tr>
<tr>
<td></td>
<td>Release 18.0 adds the following:</td>
</tr>
<tr>
<td></td>
<td>• ANSYS SpaceClaim Direct Modeler</td>
</tr>
<tr>
<td></td>
<td>• ANSYS OptiGrid</td>
</tr>
<tr>
<td></td>
<td>• ANSYS FENSAP-ICE PrepPost</td>
</tr>
<tr>
<td></td>
<td>• Can be used in combination with ANSYS CFD Enterprise solver to run AIM Pro</td>
</tr>
<tr>
<td></td>
<td>• ANSYS Forte PrepPost</td>
</tr>
<tr>
<td>ANSYS FENSAP-ICE</td>
<td>Replaces ANSYS FENSAP-ICE 16/32/64/128/256/unlimited and includes:</td>
</tr>
<tr>
<td></td>
<td>• ANSYS FENSAP-ICE Solver</td>
</tr>
<tr>
<td></td>
<td>• ANSYS FENSAP-ICE PrepPost</td>
</tr>
<tr>
<td></td>
<td>• ANSYS FENSAP-ICE Turbo</td>
</tr>
<tr>
<td></td>
<td>• ANSYS OptiGrid</td>
</tr>
<tr>
<td></td>
<td>ANSYS FENSAP-ICE provides access to one solver core.</td>
</tr>
</tbody>
</table>

### Additional Product Availability

The ability to perform parametric analysis with ANSYS DesignXplorer is now bundled with ANSYS Autodyn, ANSYS DesignSpace, ANSYS Mechanical Pro, Premium and Enterprise in ANSYS 18.0 and 18.1.

Specific to the Mechanical products, Topology Optimization is now enabled at all license levels. Topology Optimization is a physics driven optimization that is based on a set of loads and boundary conditions provided by a preceding analysis.
Part I: ANSYS Structural Products

Release notes are available for the following ANSYS Structural products:

- Mechanical Application (p. 3)
- Mechanical APDL (p. 9)
- Autodyne (p. 15)
- Aqwa (p. 17)
- ACP (p. 19)
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. Graphics Enhancements
1.4. Geometry Enhancements
1.5. Model Assembly and External Model Enhancements
1.6. Contact and Connection Enhancements
1.7. Mesh Enhancements
1.8. Linear Dynamics Enhancements
1.9. Topology Optimization Enhancements
1.10. Loads/Supports/Conditions Enhancements
1.11. Mapping Enhancements
1.12. Solution Enhancements
1.13. Rigid Body Solver Enhancements
1.14. Explicit Dynamics Enhancements
1.15. Results Enhancements

Backwards Compatibility: ANSYS products strive to enable the reading and resuming of databases from previous releases. We currently test this capability for the previous two releases and any included point releases. This means that release 18.1 was tested and verified to be backwards compatible with release 16.0 and 17.0 as well as any associated point releases (16.x, 17.x). Although not verified for even earlier releases, ANSYS Mechanical should also allow resuming databases from them.

1.1. Incompatibilities and Changes in Product Behavior from Previous Releases

Release 18.1 includes several new features and enhancements that result in product behaviors that differ from previous releases. These behavior changes are presented below.

• **Nonlinear Formulation (Transient Thermal).** For Transient Thermal analyses, the application now specifies the Program Controlled option of the Nonlinear Formulation property (Analysis Settings>Nonlinear Controls) with the Full setting when enthalpy is present as a material property. In previous releases, the Quasi option was specified by default.

• **Mechatronics Analysis.** The name of the macro file used to export the reduced model was changed from "ExportSpaceSpaceMatrices.mac" to "ExportStateSpaceMatrices.mac".

• **Specifying Edge/Edge Contact Preference for 2D Models.** The Options preference feature (see the Connections category) now enables you to change the default setting for automatic contact detection for Edges in two dimensional (2D) models. Contact detection occurs automatically by default, but you can now change this setting (to No) so that it does not take place. This setting takes effect upon future geometry attachments.
• **Topology Optimization Solution Selection.** The **Solution Selection** object is no longer an available object in the Topology Optimization analysis. An Environment listing can now be seen under the **Definition** category of the **Solution** object in the Topology Optimization analysis.

• **Topology Optimization Objective Object.** The properties of the **Objective** object was moved to Objective Worksheet in order to combine the properties together with the user specified weights.

• **Damping Controls - Structural Damping Coefficient.** For the **Structural Damping Coefficient** property of the Analysis Settings **Damping Controls**, Mechanical previously, and incorrectly, used Hertz (Hz) as the default unit of measure. The application now uses the proper unit of radians per second (rad/s).

• **LS-DYNA (Export) system.** The LS-DYNA (Export) system is no longer fully supported. You must have the Beta features enabled in order to use it in the current release. The Workbench LS-DYNA ACT extension has enhanced the capabilities of using the LS-DYNA solver in the Workbench environment.

### 1.2. General Enhancements

The following general enhancements were made at Release 18.1:

• **Scripting in Mechanical.** A new online Help guide is now available: **Scripting in Mechanical (Quick Start Guide)**. This Help guide was added to introduce scripting in Mechanical. It examines important scripting concepts and provides illustrative examples.

• **Searchable Drop-Down Options.** Details view properties that provide a drop-down list of options, such as a list of Named Selections, now enable a search box in order to quickly locate a specific option. Entering one or more characters in the search box filters the list of options to only show the ones containing the search string. This feature is turned on by default, however, you can change the default setting and disable the feature under the **UI Controls** category of the **Miscellaneous Options**.

• **Specifying Named Selections using Worksheet Criteria.** When you select **Body** as the **Entity Type**, there is a new Worksheet **Criteria** option: **Cross Section**. Using this option, the application finds bodies using the cross section selection specified in the **Value** column.

### 1.3. Graphics Enhancements

The following graphical enhancements were made at Release 18.1:

• **Line Body Thickness:** A new line body display preference, **Line Body Thickness**, is available in Mechanical. You can now specify a display preference of **Thin** (default) or **Thick**.

• **Body Color by Cross Section.** A new **Display Type** property is available for the **Geometry** object: **By Cross Section**. When this option is selected, bodies with the same cross section are assigned the same color in the **Graphics** window.

• **Frame-by-Frame Animation.** For multi-step analyses, the Animation feature in the **Graph** window has a new option: **Update Contour Range at Each Animation**. This new option enables you to view your results on a frame by frame basis. The **Geometry** window legend dynamically changes from frame to frame and the result contours display the full range of colors from the minimum value to the maximum value.

• **Presentation Mode.** Using the **F11 hotkey**, Mechanical now enables you to quickly maximize the **Geometry** window (only) for presentation purposes.
1.4. Geometry Enhancements

The following geometry enhancements were made at Release 18.1:

- **Cross Section Object.** Mechanical now displays line body cross section data as objects in the tree. Each Cross Section object displays the type, dimensions, and attributes associated with a unique cross-section. The application automatically inserts these objects under the **Cross Sections** folder when you import geometry with **Line bodies**.

- **Line Body Alignment.** ANSYS DesignModeler now has a **Frame Alignment** property that enables you to override the default frame alignment assigned by DesignModeler. See the **Cross Section Alignment** Help section in the DesignModeler User's Guide for additional information.

1.5. Model Assembly and External Model Enhancements

The following enhancements for **Model Assembly and External Model** were made at Release 18.1.

**External Model**

The following enhancements for **External Model** were made at Release 18.1:

- **Importing ABAQUS Finite Element Data.** ANSYS Workbench and Mechanical now enable you to attach additional files to an imported parent .inp file. These support files can include additional node and element data.

- **Face Components.** You can now choose to import face 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 this new capabilities:
  - **Process Face Components**
  - **Face Component Key**

**Model Assembly**

The following enhancements for **Model Assembly** were made at Release 18.1:

- **Editing Imported Mesh-Based Data.** You can now edit the **Worksheet** content of an imported object from External Model using the new **Edit Items** option.

- **Thermal Analysis Support.** You can now import coupling and remote connection data (flexible or rigid) into thermal analyses.

1.6. Contact and Connection Enhancements

The following contact and connection enhancements were made at Release 18.1:

- **Beam Contact.** Mechanical now supports contact between the edges of line bodies (beam-to-beam contact) in a 3D structural analysis.

- **Specifying Edge/Edge Contact Preference for 2D Models.** The **Options** preference feature (see the Connections category) now enables you to change the default setting for automatic contact detection for Edges in two dimensional (2D) models. Contact detection occurs automatically by default, but you can now change this setting (to No) so that it does not take place. This setting takes effect upon future geometry attachments.
• **Face Overlap Tolerance.** Enables you to set your preference for the minimum percentage of overlap at which a contact pair will be created for two overlapping faces. This setting enables the software to obtain more precise contact pairs during automatic contact generation based on a default tolerance that is appropriate for your simulation type. The Face Overlap Tolerance setting in the Connections group of the Mechanical Options panel determines the default tolerance. You can modify the Face Overlap Tolerance property in the Details view of the Connection Group folder to override the default for the current model.

• **Edge Overlap Tolerance.** Enables you to set your preference for the minimum percentage of overlap at which a contact pair will be created for an edge and a face that overlap. This setting enables the software to obtain more precise contact pairs during automatic contact generation based on a default tolerance that is appropriate for your simulation type. The Edge Overlap Tolerance setting in the Connections group of the Mechanical Options panel determines the default tolerance. You can modify the Edge Overlap Tolerance property in the Details view of the Connection Group folder to override the default for the current model.

• **Only Beam Edges for Face/Edge Contact Detection.** You can select the new Only Beam Edges option for the Face/Edge property so that face to edge connection uses only edges of beam bodies to determine connection with all faces. In the Connections group of the Mechanical Options panel, you can set the Face/Edge property to Only Beam Edges to make this the default for face to edge connection detection. You can modify the Face/Edge setting in the Details view of the Connection Group folder to override the default for the current model.

• You can now drag and drop the Contacts folder onto the Mesh object to create a Contact Sizing control for each contact region in the folder automatically.

### 1.7. Mesh Enhancements

Refer to the 18.1 Release Notes of the Meshing application for new features and enhancements associated with Meshing in the Mechanical application.

### 1.8. Linear Dynamics Enhancements

The following enhancements were made at Release 18.1 for Linear Dynamic features and analyses:

• **Modal Analysis Commands.** For Modal analyses, there is a new Commands object property: Point Selection Mode. This property enables you to send solver commands based on the solver points of the Campbell Diagram as specified by the Rotordynamics Controls of the Analysis Settings.

### 1.9. Topology Optimization Enhancements

The following analysis enhancements were made at Release 18.1:

• The Topology Optimization Analysis now supports:
  – Optimization of objectives and constraints selected from multiple Static Structural or Modal analysis types.
  – Combined objectives through the Objective object Worksheet.
  – Response Constraints applied to selected static structural or modal analyses.

• For the Topology Optimization Analysis:
  – The Manufacturing Constraint object now provides the following new options:
→ Pull Out Direction
→ Extrusion
→ Cyclic
→ Symmetry

- The **Response Constraint** object now provides the following new options:
  → Local von-Mises Stress Constraint
  → Displacement Constraint
  → Reaction Force Constraint

- **Topology Optimization Frequency Detection.** The detection of repeating frequencies was modified. If the design objective is to optimize a frequency, then all of the repeating frequencies are optimized simultaneously. This process could change the iteration sequence compared to previous versions.

### 1.10. Loads/Supports/Conditions Enhancements

The following loads/supports/conditions enhancements were made at Release 18.1:

- **Step Dependent Tabular Loading.** For Static Structural analyses that use the MAPDL solver, the **Independent Variable** property now provides the option **Step**, enabling you to specify loading on a per step basis. The application does not use tables, but rather sends the loading to the solver as constant values for each step.

### 1.11. Mapping Enhancements

The following mapping enhancements were made at Release 18.1:

- **Maxwell-Harmonic Coupling:** Mechanical now enables you to import **Surface Force Density** data into a Harmonic analysis from an upstream Maxwell Eddy-current solution.

### 1.12. Solution Enhancements

The following solution enhancements were made at Release 18.1:

- **Worksheet Summary.** The **Worksheet** summary feature now provides an option, **List Solver Component Information**, that enables you to list, in tabular form, the Material IDs, Element Name IDs, and Element Type IDs generated during the solution process.

- **Restart Controls - Combined Restart Files.** The **Combined Restart Files** property of the Analysis Settings **Restart Controls** category now enables you to restart your downstream pre-stress analysis using a different number of cores than the static structural analysis. You will need to set this property to **Yes** prior to solving your static structural analysis.

### 1.13. Rigid Body Solver Enhancements

The following Rigid Body Solver enhancements were made at Release 18.1
• **Multi-variable input table.** The new GILTable command object allows you to input a multi-variable interpolated table based on a cloud of points. This table can be used to define loads that are dependent on other variables than time.

### 1.14. 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 following Explicit Dynamics Solver enhancements were made at Release 18.1:

• A new algorithm allows for an enhanced contact treatment which uses the manner in which surfaces (meshes) are connected. This is available when Contact Detection is set to Trajectory and the **sliding option** is set to Connected Surface.

The following LS-DYNA Solver enhancements have been made at Release 18.1:

• The **Workbench LS-DYNA** ACT extension is now included in the installation, and has enhanced the capabilities of using the LS-DYNA solver in the Workbench environment.

### 1.15. Results Enhancements

The following results enhancements were made at Release 18.1:

• **Solver Component Result Scoping.** A new result **Scoping Method** is available: **Solver Component**. This option enables you to scope results on solver generated elements, such as surface pressure elements or weak springs, that were previously only available for post-processing within Mechanical APDL.
Chapter 2: Mechanical APDL Release Notes

Release 18.1 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. 9)
• Solvers (p. 10)
• Commands (p. 11)
• Results File (p. 11)
• Elements (p. 13)
• Documentation (p. 13)
• ANSYS Product Improvement Program (p. 13)

Also see Known Limitations (p. 14) and the ANSYS Customer Portal (p. xi) for important information about this release.

Backward Compatibility: Mechanical APDL Release 18.1 can read database files from all prior Mechanical APDL releases. Due to ongoing product improvements and defect corrections, however, results obtained from old databases running in new releases may differ somewhat from those obtained previously.

2.1. Structural

Release 18.1 includes new features and enhancements for the following structural analysis disciplines:

2.1.1. Contact
2.1.2. Linear Dynamics

2.1.1. Contact

Release 18.1 includes the following enhancements for structural analyses involving contact:

2.1.1.1. Beam-to-Beam Contact
2.1.1.2. Performance Improvements for Large Contact Models
2.1.1.3. Modeling Interface Damping in Assembled Structures

2.1.1.1. Beam-to-Beam Contact

The following enhancements are available for beam-to-beam contact for both pair-based and general contact definitions:
• The 3-D line-to-surface contact element CONTA177 now supports 3-D internal beam-to-beam contact in addition to 3-D external beam-to-beam contact. Prior to this release, 3-D internal beam-to-beam contact could only be modeled with the 3-D line-to-line contact element, CONTA176.

• Prior to this release, each contact detection point of the CONTA177 3-D line contact element could interact with one target segment only. Now, up to eight target segments (controlled by KEYOPT(14)) can be involved in each contact constraint. This enhancement enables you to fully model applications such as wire clamping and multilayer coil compression with beam-to-beam contact.

• Beam-to-beam contact requires an equivalent radius for both the contact and target elements. If the contact radius and/or target radius are not specified, the program automatically calculates the equivalent radius for each individual contact and target element based on the associated geometry of underlying elements. In this case, the equivalent radius may vary along the length of a beam.

2.1.1.2. Performance Improvements for Large Contact Models

Significant performance improvements in terms of run time were made for projection-based MPC bonded contact (KEYOPT(4) = 3 and KEYOPT(2) = 2 for the contact elements).

2.1.1.3. Modeling Interface Damping in Assembled Structures

Vibration properties of most assembled mechanical systems depend on the damping mechanism at the interfaces between parts. To account for interface damping in a harmonic or mode-superposition analysis, you can define two bonded contact pairs at the interface: one for normal damping and one for tangential damping. Independent structural damping coefficients (MP,DMPR) are applied to these two bonded contact pairs. For more information, see Modeling Interface Damping in Assembled Structures in the Mechanical APDL Contact Technology Guide.

2.1.2. Linear Dynamics

Release 18.1 includes the following enhancements for structural analyses involving linear dynamics:

2.1.2.1. Component Mode Synthesis (CMS) Enhancements

2.1.2.2. MPRS for FSI

2.1.2.1. Component Mode Synthesis (CMS) Enhancements

For the generation pass in a CMS substructuring analysis, eigensolver calculation performance was improved for the Block Lanczos eigensolver. In some cases, CMS superelement generation can occur significantly faster.

The subspace (SUBSP) and supernode (SNODE) mode-extraction methods (CMSOPT) for the eigensolver calculations are now available as alternatives to the default Block Lanczos (LANB) method. The alternative options can improve calculation performance during the CMS superelement generation pass.

2.1.2.2. MPRS for FSI

Multiple point response spectrum analysis is now supported for acoustic analyses. For more information, see Spectrum Analysis in the Mechanical APDL Acoustic Analysis Guide.

2.2. Solvers

Release 18.1 includes the following improvements to the solution process:

2.2.1. Distributed ANSYS Enhancements
2.2.2. GPU Acceleration Enhancements

2.2.1. Distributed ANSYS Enhancements

The following enhancements are available for distributed-memory parallel processing (Distributed ANSYS):

- The two new domain decomposition methods introduced at Release 18.0, frequency domain decomposition and cyclic-harmonic index domain decomposition ($\text{DECOMP} = \text{FREQ}$ and $\text{CYCHI}$, respectively, on the $\text{DDOPTION}$ command), offer more capability. You can now combine either method with mesh-based domain decomposition, potentially reducing peak memory consumption by limiting the number of harmonic frequency solutions or cyclic harmonic index solutions occurring simultaneously. You can also use potentially all available CPU cores if more CPU cores are available beyond those needed to solve the number of harmonic frequencies or cyclic harmonic indices.

- Linear perturbation analyses, which inherently involve the multiframe restart process, now support the ability to change the core count between the base static or full transient analysis and the restarted analysis. For more information, see Restarts in Distributed ANSYS in the Mechanical APDL Parallel Processing Guide.

- By default, a shell script (Linux) or .bat file (Windows) is now written into the current working directory for a simulation. The script or file facilitates termination of all distributed processes created by Distributed ANSYS. You can use the script to ensure that all processes are properly terminated if you wish to end the simulation prematurely, or if the job hangs.

2.2.2. GPU Acceleration Enhancements

The following enhancements are available for the GPU Accelerator capability:

- The NVIDIA GPU driver requirements were updated. For specific driver-version requirements, see the ANSYS, Inc. Installation Guide for your platform.

- The NVIDIA CUDA libraries were updated to version 8.0, enabling better support and performance for the recently released Pascal-generation of GPU cards from NVIDIA.

- Support was added for the NVIDIA Quadro P5000, NVIDIA Quadro P6000, and NVIDIA Quadro GP100 GPU video cards.

- Several enhancements were made to the GPU Acceleration feature logic in the Sparse solver to improve performance and robustness.

2.3. Results File

The following enhancements to the results file (Jobname.RST, Jobname.RTH, etc.) appear in Release 18.1:

- A list of dead elements is written to the results file.

- Components are written to the results file.

2.4. Commands

This section describes changes to commands at Release 18.1:

- 2.4.1. New Commands
- 2.4.2. Modified Commands
- 2.4.3. Undocumented Commands
Some commands are inaccessible from menus and are available via the command input area or batch file input only. The documentation for each command indicates menu path information, if available.

### 2.4.1. New Commands

The following new commands are available:

- **EXOPTION** -- Specifies the EXPROFILE options for the Mechanical APDL to ANSYS CFX profile file transfer.
- ***SORT** -- Sorts the values of a specified vector.

### 2.4.2. Modified Commands

The following commands were enhanced or otherwise modified:

- **CMSOPT** -- Specifies component mode synthesis (CMS) analysis options. The new EIGMETH argument specifies the mode extraction method to be used during the generation pass.
- ***COMP** -- Compresses a matrix using a specified algorithm. For the SVD compression method, new options are available to output the compression part of the matrix.
- **DDOPTION** -- Sets domain decomposer options for Distributed ANSYS. The command now enables you to combine mesh-based domain decomposition with the frequency-based method or the harmonic indices-based method.
- **EALIVE** -- Reactivated an element (for the birth and death capability). Now supports tabular input.
- **EKILL** -- Deactivates an element (for the birth and death capability). Now supports tabular input.
- **EMTGEN** -- Generates a set of TRANS126 electromechanical transducer elements. The new Smethod argument enables you to choose between the augmented or full stiffness method for the generated elements. If the subsequent analysis is a linear perturbation harmonic analysis, you must select the full stiffness method ($Smethod = 1$) to achieve accurate results.
- **EREINF** -- Generates reinforcing elements from selected existing (base) elements. You can now enable or disable the limit of the angle between a MESH200 element and a base element when using the mesh-independent reinforcing method.
- **PSDGRAPH** -- Displays input PSD curves. The new DisplayKey option enables you to control the display of points markers and numbering.
- **STORE** -- Stores data in the database for the defined values. The new FREQ and Toler options enable you to define frequency values for PSD analyses.
- **SECDATA** -- Describes the geometry of a section. The input format for equivalent contact radius ($TYPE = CONTACT, Subtype = RADIUS$) was changed to model internal beam-to-beam contact.

### 2.4.3. Undocumented Commands

No commands were undocumented at this release. For information about commands that were 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 18.1:

2.5.1. Modified Elements
2.5.2. Undocumented Elements

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

2.5.1. Modified Elements

The following element was enhanced:

• COMBI214 -- Mass can now be added to the COMBI214 element via KEYOPT(6).

• CONTA177 -- This line-to-surface contact element can now model internal beam-to-beam contact (a beam sliding inside of a hollow beam or pipe sliding inside another pipe). Also, the element now supports multi-layer contact in which up to eight target segments can be involved in each contact constraint. Multi-layer contact is activated via the new KEYOPT(14).

• CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, CONTA177 -- These contact elements now report contact element force due to tangential stress as output items CNTX, CNTY, and CNTZ.

• TARGE170 -- This target segment element has a new KEYOPT(9) for specifying the type of beam-to-beam contact (external or internal) for pair-based contact involving CONTA177 elements.

2.5.2. Undocumented Elements

No elements were undocumented at this release. For information about elements that were undocumented in prior releases, see the archived release notes on the ANSYS Customer Portal (p. xi).

2.6. Documentation

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

2.6.1. Technology Demonstration Guide

The following example problem was added to the Technology Demonstration Guide:

• Active and Passive Lateral Earth-Pressure Analysis – Simulates soil behavior under active and passive earth-pressure loadings. The problem shows how the nonlinear plastic behavior of soil can be modeled with a Mohr-Coulomb material.

• Electromigration in a Solder Ball – Demonstrates a transient electromigration analysis of a solder ball. The coupled-field solution computes the deviation in atomic concentration from an initial unit value due to the combined effects of diffusion, electromigration, stress migration, and thermomigration.

2.7. ANSYS Product Improvement Program

The new ANSYS Product Improvement Program was implemented at Release 18.1. The voluntary program enables ANSYS, Inc. to collect and analyze anonymous usage data reported by Mechanical APDL. The data enable us to better understand how you use Mechanical APDL so that we can develop or enhance
product features that better address your needs. For more information, see ANSYS Product Improvement Program in the Mechanical APDL Operations Guide.

2.8. Known Limitations

The limitation related to specifying Remote Shell (rsh) on the Mechanical APDL product launcher that was reported in the 18.0 Release Notes was not corrected. For more information, including a workaround, see The Mechanical APDL Product Launcher in the Mechanical APDL Operations Guide.
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. 8)).

3.1. New Features and Enhancements

The following new features and enhancements are available in release 18.1. Refer to the product specific documentation for full details.

- A new algorithm allows for an enhanced contact treatment which uses the manner in which surfaces (meshes) are connected. This is available when Contact Detection is set to Trajectory and the sliding option is set to Connected Surface.
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 18.1. 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. Wind Spectrum Seed

The starting seed for a wind spectrum can now be set separately from the seed of a wave spectrum. See Wind Spectra Definition and Wind for more information.

4.1.2. Frequency Dependent Added Mass and Damping

The restriction on combining user-defined added mass and damping with convolution in Aqwa-Naut and Aqwa-Drift was removed. See The WAMS/WDMP Data Record - Wave Frequency Added Mass Matrix and Wave Frequency Damping Matrix.

4.1.3. Recalculate QTFs Based on User-Input RAOs

The new URAO option allows you to recalculate QTFs based on RAOs input with the TRAO and RRAO Data Records in Data Category 7. See Administration and Calculation Options for the Aqwa Suite.

4.2. Hydrodynamic Analysis Systems

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

4.2.1. Deactivated Freedoms Object

In Stability and Time Domain analyses you can add a Deactivated Freedoms object to restrict the degrees of freedom of a structure. See Deactivated Freedoms.

4.2.2. Morison Hull Drag Coefficients Object

The Morison Hull Drag Coefficients object allows you to input a matrix of Morison Hull Drag Coefficients using tabular input. These coefficients are used to calculate hull drag forces and moments in a similar way to that for a Morison element. See Morison Hull Drag Coefficients.
4.2.3. Time Domain Statistics Distribution Function

The Time Domain Statistics result object can now be switched between a Probability Density Function (PDF) and a Cumulative Distribution Function (CDF). See Time Domain Statistical Results.

4.2.4. Generated Mesh Information

Details of the mesh now show the numbers of Line Body nodes and elements. See Mesh.

4.2.5. Visualization of Splitting Force Bounding Box

When defining the data required for a Splitting Forces result, you can now display the bounding box and over-turning moment calculation coordinate. See Splitting Forces (RAO).

4.2.6. Fixed Points Container

Fixed Points are now grouped together in their own parent object, producing a more compact object tree. See Add Fixed Points.
Chapter 5: ANSYS Composite PrepPost (ACP)

The following enhancements are available in release 18.1. Refer to the product specific documentation for full details of the new features.

5.1. New Features in ANSYS Composite PrepPost (ACP) 18.1
5.2. Supported Platforms for ANSYS Composite PrepPost (ACP) 18.1
5.3. Known Limitations and Incompatibilities

5.1. New Features in ANSYS Composite PrepPost (ACP) 18.1

The following new features were added to ANSYS Composite PrepPost (ACP) for the 18.1 release.

5.1.1. Improved Tree View

5.1.1. Improved Tree View

The ACP tree view now features a search bar that allows you to search and filter the object tree. It assists in locating objects in an ACP model.

**Known Limitation** Care should be taken when defining objects while tree filtering is active. The object selection must be consistent with what is displayed in the filtered tree. It is recommended that you clear any tree filters while defining new objects.

5.2. Supported Platforms for ANSYS Composite PrepPost (ACP) 18.1

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

5.3. Known Limitations and Incompatibilities

There are no known incompatibilities with previous releases for ANSYS Composite PrepPost in Release 18.1.
Part II: ANSYS Fluids Products

Release notes are available for the following ANSYS Fluids products:

- Fluent (p. 23)
- CFX (p. 31)
- TurboGrid (p. 33)
- ANSYS BladeModeler (p. 35)
- CFD-Post (p. 37)
- Polyflow (p. 39)
- Forte (p. 41)
- ANSYS Chemkin-Pro (p. 45)
- FENSAP-ICE (p. 47)
Chapter 1: Fluent Release Notes

The following sections contain release information for ANSYS Fluent 18.1.

1.1. New Features in ANSYS Fluent 18.1  
1.2. Supported Platforms for ANSYS Fluent 18.1  
1.3. New Limitations in ANSYS Fluent 18.1  
1.4. Resolved Issues and Limitations  
1.5. Updates Affecting Code Behavior

Backwards Compatibility: In most instances, version 18.1 of ANSYS Fluent can read case and data files from all past Fluent releases. However, due to product improvements and defect fixes, results obtained from old cases running in new releases may differ to some degree from the previously obtained results. Additionally, infrequent changes made in UDF macros over time could lead to some user-defined functions failing to compile without modification.

1.1. New Features in ANSYS Fluent 18.1

New features available in ANSYS Fluent 18.1 are listed below. Where appropriate, references to the relevant section in the User's Guide are provided.

User Interface

- Surface selection and context menus are now available in the graphics window, allowing you to modify certain settings directly on the model. For additional information, see Graphics Windows in the Fluent User's Guide.

- Using the mouse-probe functionality on displayed meshes in the graphics window now prints their zone-id and zone-name, in addition to the surface group, surface-name, and surface-id.

- The tree now lists the zone-id along with the zone type for cell zones and boundary conditions.

- For additional information about the user interface, see Graphical User Interface (GUI) in the Fluent User's Guide.

Solver-Meshing

- The user interface for creating and managing non-conformal mesh interfaces is enhanced with new functionality and improved for ease of use:
  - It is now possible to edit the setup of individual or multiple selections of mesh interfaces, rather than having to delete and recreate each one individually.
  - The ability to automatically create multiple mesh interfaces from a list of interface zones (previously available through the define/mesh-interfaces/auto-pairing text command) is now available in a new Mesh Interfaces dialog box, as well as the right-click menu that is available when multiple interface zones are selected in the tree.
The workflow is streamlined by the removal of the **Mesh Interfaces** task page and the incorporation of its functionality into the **Mesh Interfaces** dialog box.

For details, see Using a Non-Conformal Mesh in ANSYS Fluent in the *Fluent User's Guide*.

- For dynamic meshes, it is now possible to enable local face remeshing (with or without feature detection) for a boundary zone whose motion / deformation is defined by a user-defined function. *(User-Defined Motion)*

- The following improvements were made for overset meshes:
  - The overset domain connectivity will now be retained by default when writing case files in the hierarchical data format (HDF), thus ensuring a consistent overset interface in all ANSYS Fluent sessions. *(Writing and Reading Overset Files)*
  - 2D axisymmetric flows are now supported for overset meshing. *(Compatibilities)*

**Solution Convergence**

- Convergence conditions can be deactivated without deleting the condition. For additional information on setting convergence conditions, see Convergence Conditions in the *Fluent User's Guide*.

**Models**

- Heat Transfer and Radiation
  - Thermal data transfer at non-conformal mesh interfaces with the **Mapped** or **Coupled Wall** options enabled is now supported between ANSYS Fluent and ANSYS Mechanical via System Coupling.
  - You can now use the Monte Carlo radiation model with polyhedral meshes.
  - **Mesh Coarsening** is now available for the Monte Carlo radiation model. This feature reduces the number of effective cells in the domain that are used by the radiation model, and thereby increases the speed of the radiation calculation and reduces peak memory usage. For complete details, see Setting Up the MC Model in the *Fluent User's Guide*.

- Combustion and Species Transport
  - For species transport cases with a single phase flow, the ability to display only selected boundary species was extended to the **Patch** dialog box, the **Hybrid Initialization** dialog box, and the **Solution Initialization** task page. For more information, see Patching Values in Selected Cells, Steps in Using Hybrid Initialization, and Initializing the Entire Flow Field Using Standard Initialization in the *Fluent User's Guide*.

- Discrete Phase Model
  - Escaped Mass DPM report definitions are now supported as a full feature. The mass of particles escaping through one or more boundaries can be reported either for all or selected injections. *(DPM Report Definition)*
  - The ability to create an extended DPM summary report using the text user interface is now available as a full feature. The report can be created either for all or selected injections. For additional information, see report/ in the Fluent Text Command List.
New Features in ANSYS Fluent 18.1

- It is now possible to enable a dynamic load balancing that is designed for use with the **Use DPM Domain** option in the **Discrete Phase Model** dialog box. ([Parallel Processing for the Discrete Phase Model](#))

- **Eulerian Multiphase Model**
  - For Eulerian and Mixture multiphase cases with species transport, you can now select different species mass transfer models and options for multiple species-mass-transfer mechanisms in the same simulation. For more information, see [Including Mass Transfer Effects in the Fluent User's Guide](#).

- **Eulerian Wall Film Model**
  - Specifying the **Minimum Thickness** in the **Eulerian Wall Film** dialog box is no longer required.

**Material Properties**

- For NIST real gas models, you can now create NIST Lookup saturation tables for binary materials. ([Creating Binary Mixture Saturation Tables for Binary Mixtures](#))

**Cell Zones and Boundary Conditions**

- Torque-speed coupling with GT-POWER is now available. This enables Fluent to receive a shaft speed from GT-POWER and return torque. Note that GT-POWER v2017.2 or later is required to use this feature. For more information, see [Torque-Speed Coupling with GT-POWER in the Fluent User's Guide](#).

- A mass-flow outlet boundary condition is now available. This replaces the functionality that was previously available through a mass-flow inlet defined to use the **Outward Normals** direction specification method or defined with a **Direction Vector** that pointed out of the domain. Defining your boundary as a mass-flow outlet streamlines your setup, as you will no longer be prompted with irrelevant fields and models. ([Mass-Flow Outlet Boundary Conditions](#))

**Parallel Processing**

- A binary that is optimized for the AVX2 instruction set can be used to enhance performance when running on processors that support AVX2. This optimized binary is only available for the parallel version of ANSYS Fluent on Linux, and can be invoked by using the following command line option: `-platform=intel`. Performance will improve most for cases with polyhedra cells that also use the coupled solver; other cases will see minor improvements.

**Graphics, Postprocessing, and Reporting**

- Scene colormap locations are retained and saved with the case file.

- Pathlines can be included in scenes. For additional information on scenes, see [Displaying a Scene in the Fluent User's Guide](#).

- The orientation of animation objects can be specified as a view in the **Animation Definition** dialog box.

- Report files and plots can be deactivated so they are not written or plotted during the calculation. The files and plots can be re-activated as required.

- You can clear report files and plots via the text commands `solve/report-files/clear-data` and `solve/report-plots/clear-data`. 
• The statistic monitors functionality from previous releases is now accomplished by creating report files and/or plots for the flow-time, delta-time, iterations-per-timestep, periodic-pressure-gradient, and periodic-bulk-temperature-ratio report definitions. This means that you can choose to separate these quantities into different files and plots or include them all in the same file and plot (for unit consistency, only flow-time and delta-time can be included in the same report plot). For additional information on statistic monitors, see Monitoring Statistics in the Fluent User’s Guide.

Add-Ons

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 18.1 Beta Features Manual, which is available on the ANSYS Customer Portal.

1.2. Supported Platforms for ANSYS Fluent 18.1

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

1.3. New Limitations in ANSYS Fluent 18.1

The following is a list of new or recently discovered limitations known to exist in ANSYS Fluent 18.1. Where possible, suggested workarounds are provided.

• Models
  – For discrete phase model simulations, the Use DPM Domain option (in the Discrete Phase Model dialog box) is not used in any of the following situations:
    → if the case includes a wall / shadow pair
    → if you are not running with distributed memory on a cluster

  These limitations existed for previous releases.

• Platform Support and Drivers
  – Fluent may terminate abnormally during launch when running on Community Enterprise OS (CentOS) 7.3 or Red Hat Enterprise Linux (RHEL) 7.3 when DISPLAY is set to a Virtual Network Computing (VNC) session. To attempt to resolve this, verify that you are using a supported graphics card and update the graphics card drivers (directly from the graphics card vendor website). If the issue persists, you can do one of the following: set the DISPLAY to a local machine; set the LD_PRELOAD environment variable to /usr/lib64/libstdc++.so.6; or use the alternative drivers x11 or null (either by defining it in the HOOPS_PICTURE environment variable or using the -driver Fluent command line option).

• Graphics, Reporting, and Postprocessing
  – The use of node weights for node-based gradients in postprocessing is not available for cases with overset meshes.

  – It is possible that the graphics window may become [Out of Date] (which would be indicated at the top of the graphics window), if you make changes to items that are already displayed.
To resolve this out of date state, right-click in the graphics window and click **Refresh Display** in the context menu that appears.

*If a context menu does not appear on a right-click of the graphics window, ensure that the right-mouse button is set to an action other than **mouse-probe** with **long description** or **mouse-zoom** (**Viewing** ribbon tab, **Mouse** group box).*

- (Windows only) When using the context menus in the graphics window, an artifact of a previous menu may linger in the display. If it occurs, you can stop this phenomenon by disabling the **Fade or slide menus into view** setting in Windows. To access this setting, open **Advanced system settings** and click **Settings...** in the **Performance** group box (**Advanced** tab of the **System Properties** dialog box).

- If you create an **Animation Definition** for **residuals**, you must ensure that **Plot** is enabled in the **Residual Monitors** dialog box or you may see an error printed in the console when you run the calculation.

- The view specified in the **Animation Definition** dialog box is the view that you will see during animation playback, regardless of the specified **Storage Type**. This is also true for MPEGs created using the **Playback** dialog box.

- **Data Import and Export**
  - The maximum number of profiles that can be read into a single Fluent session is 50.
  - When exporting solution data to the PATRAN and/or AVS formats, Fluent may terminate abnormally if the number of selected variables is more than 40. As a workaround you can perform multiple exports, each with a maximum of 40 variables selected.

- **User-Defined Functions (UDFs)**
  - In some situations you may encounter an error with interpreted UDFs related to the C preprocessor (CPP) distributed with Fluent. As a workaround, you can enter the full path to the system’s preprocessor in the **CPP Command Name** field in the **Interpreted UDFs** dialog box; for example, `/usr/bin/cpp` or `gcc -E` (Linux only).

- **For a list of ongoing Fluent limitations listed in previous ANSYS, Inc. Release Notes, refer to Known Limitations in ANSYS Fluent 18.1 in the Fluent Getting Started Guide.**

### 1.4. Resolved Issues and Limitations

This section lists issues and limitations that existed in previous releases, but that are resolved and removed in ANSYS Fluent 18.1.

- **Data Import and Export**
  - A fix was introduced to avoid abnormal terminations when performing the following export operations in the ABAQUS, NASTRAN, Mechanical APDL Input, I-deas Universal, and PATRAN formats:
    - surface export of the temperature and heat transfer coefficient from periodic boundaries
    - domain export of the heat transfer coefficient for cases containing periodic boundaries

- **Models**
- A fix was introduced for discrete phase model cases that have the **Use DPM Domain** option enabled, to fully support particles crossing non-conformal interfaces and avoid abnormal terminations of the simulation.

- A fix was introduced for parallel cases that include shell conduction, in order to avoid nonphysical temperatures (and subsequent solution divergence or abnormal termination) that could result if you manually reordered the domain during the simulation (but after shell creation). For a workaround when running on earlier versions of Fluent, you can try only reordering prior to starting the calculation.

**Boundary Conditions**

- The **High Roughness (Icing)** model for wall zones is now supported with the density-based solver.

**Mesh**

- A fix was introduced for overset cases, to avoid abnormal terminations that could occur when displaying node values of field functions with the **Global Range** option disabled.

**Graphics, Postprocessing, and Reporting**

- Scene colormap locations are now retained and saved with the case file.

- MPEG movies created from PPM Image animations will contain colormaps if they originally contained a colormap when viewed within Fluent.

- Animation object views are saved in the **Animation Definition** dialog box and persist with the case file.

- A fix was introduced so that you can create output parameters using the `define/parameters/output-parameters/create/report-definition` text command.

- A fix was introduced so that animation playback from a previously saved case and data file is available for the **PPM Image** storage type (assuming the relevant animation sequence file *.cxa is available in the appropriate folder).

- The **Clear History** command (accessed by right-clicking the animation definition under the **Calculation Activities**>**Solution Animations** branch in the tree or by using the text command `solve/animate/objects/clear-history`) now applies not only to animations created using the **HSF File** storage type, but also to those created using the **In Memory** and **PPM Image** storage types.

- A fix was introduced for **PPM Image** animations, in order to reduce flickering in the graphics window and retain the object orientation when you continue a calculation with the **Playback** dialog box open.

- User defined report definitions can be created through the **Solving** ribbon tab (**Solving/Reports/Definitions/User Defined...**) or by right-clicking the **Report Definitions** branch in the tree (under **Solution**).

- A fix was introduced to prevent fatal signals that could result from using the right-click probe on contour or vector objects in Linux.

**Parallel**
A corrected version of the Intel MPI was included in ANSYS Fluent version 18.1, which ensures that performance on Linux with an Intel SkyLake processor is optimal when the Intel MPI is selected as the MPI choice for Fluent; therefore, it is no longer necessary to use the following command line option when starting Fluent: -mpiopt="-genv _MPI_INTRANODE_EAGER_THRESHOLD=8".

1.5. Updates Affecting Code Behavior

This section contains a list of code changes implemented in ANSYS Fluent 18.1 that may cause behavior and/or output that is different from the previous release.

Note

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

Solver-Meshing

• The 
  Create/Edit Mesh Interfaces dialog box can now only be opened by clicking the Manual Create... button in the Mesh Interfaces dialog box.

• Cases with mesh interfaces created in version 18.0 or later (including those with interfaces created as part of a sliding mesh problem) may see a reduction in quality issues (such as left-handed faces) and/or convergence problems compared to earlier versions. For case files in which the interface was created in an earlier version, you can obtain such reductions if you recreate the interface in version 18.0 or later by entering the following Scheme command in the console: (recreate-sliding-interfaces).

• For overset meshes, the least squares interpolation method was improved and also made suitable for use in Volume of Fluid (VOF) multiphase flows (previously this combination was not recom- mended).

• Improvements have been made in the handling of overset meshes, and so overset interfaces and/or results may change compared to previous releases.

Mesh Morpher/Optimizer

• The content of the "Modeling Flows Using the Mesh Morpher/Optimizer" chapter in the Fluent User’s Guide was moved and combined with information about the adjoint solver, and can now be found in the following chapter: Design Analysis and Optimization.

Discrete Phase Model

• For parallel DDPM simulations, a correction was made to the dispersed phase velocity along partition boundaries, which may produce more accurate particle tracking results compared to previous releases. Note that it is no longer necessary to use the user-defined function provided on the ANSYS Customer Portal as a workaround.

Cell Zones and Boundary Conditions

• For case files created in previous versions, any mass-flow inlet defined to use the Outward Normals direction specification method will automatically be converted to be a mass-flow outlet in version 18.1, and this will not affect the solution results; however, any journals or scripts that
set up such case files must be revised manually to use the define/boundary-conditions/mass-flow-outlet text command. Mass-flow inlets defined with a Direction Vector that pointed out of the domain will remain unaltered. (Mass-Flow Outlet Boundary Conditions)

- There was an API change in version 18.1 with GT-POWER v2017.1 that might cause solutions with Mass Flux coupling to differ from previous versions of ANSYS Fluent.

**Adjoint Solver Module**

- In order to increase the ease of use, the adjoint solver module will now be automatically loaded the first time any part of the adjoint solver user interface is accessed; therefore, it is no longer necessary to enter the define/models/addon-module text command prior to setting up and using the adjoint solver. The documentation for the adjoint solver was also moved from the Fluent Advanced Add-On Modules manual to the Fluent User's Guide and the Fluent Text Command List (see Design Analysis and Optimization and adjoint/, respectively).

**Graphics, Reporting, and Postprocessing**

- The Save and Close buttons in the Animation Definition dialog box were changed to OK and Cancel. Clicking OK saves and closes the dialog box, Cancel closes the dialog box without saving any changes.

- Clicking Compute in a report definition dialog box no longer automatically creates that report definition: now it only calculates and prints the value of that report. Report definitions are only created when you click OK.
Chapter 2: CFX Release Notes

The following sections contain release information for Release 18.1 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.

Note

Cray MPT 7.x is supported by the default solver. The solver that supports Cray MPT 5.x was discontinued as of this release.

2.2. New Features and Enhancements

This section lists features and enhancements that are new in Release 18.1 of ANSYS CFX.

• Face set topology simplification, which can improve solver performance for models with a large number of 2D primitives, is now on by default. This has some implications for the Edit Run In Progress command. For details, see the description for expert parameter topology simplification in Physical Models Parameters in the CFX-Solver Modeling Guide.

• There is a new Mesh Deformation option, Periodic Regions of Motion, which can save computational time compared to the Regions of Motion Specified option for transient cases involving wall boundaries that move with a periodic motion (for example, Transient Blade Row cases involving blade flutter). By taking advantage of the periodic nature, mesh motion equations are solved only once: at the start of the simulation. For details, see Periodic Regions of Motion in the CFX-Solver Modeling Guide.

• There is a new mesh stiffness option that automatically combines sensitivity to boundary proximity and sensitivity to small volumes. For details, see Blended Distance and Small Volumes in the CFX-Solver Modeling Guide.

2.3. Resolved Issues and Limitations

This section lists issues and limitations that existed in previous releases, but that were resolved and removed in Release 18.1 of ANSYS CFX.

The discretization change for the energy equation at stage interfaces that was introduced in Release 17.0 adversely affected the conservation of stationary frame Total Enthalpy across a Stage (Mixing-Plane) interface in cases that involved all of the following conditions:
1. The Stage model (also known as the Mixing-Plane model) is used as the frame change/mixing model, with Downstream Velocity Constraint set to Constant Total Pressure.

2. A rotor is on the downstream side of the interface.

3. The energy equation is solved.

In Release 18.1, the conservation of stationary frame Total Enthalpy is no longer adversely affected by that discretization change.

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 18.1 and Release 18.0.

- There is now a three-level hierarchy for boundary conditions with mesh motion. That is, for mesh vertices shared between two or more boundary conditions, the applied condition is calculated from the attached boundary conditions based on the following priorities:
  - Stationary takes precedence over a specified value: either Specified Displacement or Specified Location.
  - A specified value takes precedence over a sliding condition (for example, Parallel to Boundary or Surface of Revolution).
  - Any condition takes precedence over the Unspecified condition.

- The mesh deformation model numerics were changed for Transient Blade Row analyses in order to aid convergence and robustness for blade flutter simulations. In order to use the previous numerics, the expert parameter meshdisp optimization level must be set to 0.
Chapter 3: TurboGrid Release Notes

The following sections contain release information for Release 18.1 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 18.1 of ANSYS TurboGrid.

• By default, TurboGrid makes use of multiple CPU cores to improve performance. This ability can be controlled by a setting, which is described in Threading in the TurboGrid User's Guide.

• The Traditional with Control Points method is no longer supported.
Chapter 4: ANSYS BladeModeler Release Notes

The following sections contain release information for Release 18.1 of BladeGen and BladeEditor.

4.1. Supported Platforms

4.1. Supported Platforms

Platform/OS levels that are supported in the current release are posted on the ANSYS website.
Chapter 5: CFD-Post Release Notes

The following sections contain release information for Release 18.1 of ANSYS CFD-Post.

5.1. Supported Platforms
5.2. New Features and Enhancements

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 18.1 of ANSYS CFD-Post.

• Cartesian vector variable definitions are now created automatically for CGNS files if suitable scalar components are present in the file. Scalar variables which are superseded by the automatically-created CGNS vector variables are retained for backwards compatibility reasons but marked as “Deprecated” if they do not have the same names as the components of the new vector variables. The automatic vector creation can be disabled by turning off the preference Define vector variables if the new vector variable definitions cause problems with loading state and session files from previous releases. These problems are likely to occur if the old state and session files contain User Variables with the same names as the new vector variables (or vector components). In this case, the User Variables can simply be deleted from the old state or session files, or the preference can be used to revert to the behaviour of previous releases if desired.
Chapter 6: Polyflow Release Notes

The following sections contain release information for ANSYS Polyflow 18.1.

6.1. New Features
6.2. Supported Platforms
6.3. New Limitations in ANSYS Polyflow 18.1
6.4. Past Versions of ANSYS Polyflow Release Notes

6.1. New Features

The new features in ANSYS Polyflow 18.1 are as follows:

- Multiple-material coextrusion flows can be modeled in ANSYS Polyflow. See Using Multiple Materials in Polymer Flows for details.
  - To mitigate possible convergence difficulties in an extrusion problem, you should apply evolution to the moving boundaries. The Enable incremental involvement of moving boundaries option in the Numerical parameters menu was replaced by the Enable evolution on moving boundaries option. See Convergence Strategies for details.
  - Also, Evolution for non-isothermal flows was renamed to Enable convergence strategies for thermal flows in the Numerical parameters panel. See Using Evolution in Heat Conduction and Nonisothermal Flow Calculations for details.

- The documentation for porous wall conditions was clarified to include the normal force calculation and user inputs. See Porous Wall Condition for details.

- The documentation was clarified to reflect the fact that CSV files created by CFD-Post can be read into ANSYS Polyflow.

- The documentation for the adaptive meshing technique based on distance with respect to molds was clarified. See Basing the Calculation on Distance for details.

- For simplified viscoelastic flows, a check was added to make sure that the weighting coefficient should be positive or zero.

- Naming conventions for viscoelastic field variables was more fully described. See Field Names for Viscoelastic Flows for details.

- Additional ANSYS Polyflow Workbench templates are available. See Choosing a Polyflow Project Template for details.

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 18.1

There are no new limitations to note for ANSYS Polyflow 18.1. For limitations that are present in ANSYS Polyflow 18.1 but that were discovered during previous releases, see Known Limitations in ANSYS Polyflow 18.1 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\v181\polyflow\polyflow18.1.x\help\index.htm

- For Linux:
  
  path \ansys_inc\v181\polyflow\polyflow18.1.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 polyflow18.1.0).
Chapter 7: Forte Release Notes

The enhancements and defect corrections listed below are relative to ANSYS 18.0.

7.1. New Features and Enhancements
7.2. Resolved Issues since Forte 18.0
7.3. Supported Platforms

7.1. New Features and Enhancements

This section lists new features and enhancements in Release 18.1 of ANSYS Forte CFD, organized by topic.

Simulation Interface

• Added the Compression Ratio Calculator, a utility that can verify an engine compression ratio, based on the Forte setup, and facilitate repositioning of the piston to achieve a specific compression ratio.

• Two new options are available in the Turbulence options panel, for using Large Eddy Simulation (see Engineering Models and Computation below).

• Added a Time Reference Frame option, which allows settings that are time-based or crank-angle-based to be specified relative to a reference time (such as the top-dead-center piston location of a reference cylinder). This facilitates the setup of multi-cylinder cases.

• Change the default selections on Spatially Resolved Outputs to not include soot particle-related outputs and other uncommon outputs, which reduces the solution file size. [DE146379]

Job Submission, Monitoring, and Running Options

• A new, simple text-based command-line control system is now in place, which facilitates command-line control of simulation settings and general scripting for batch processing, eliminating the need to open the user interface for minor changes.

• Added cylinder-specific monitoring for multi-cylinder engine configurations.

Engineering Models and Computation

• A new option is available for handling arbitrary-size sectors with periodic boundaries, while using the automatic mesh generation with solution adaptive meshing. This extends the sectors allowed to any arbitrary size, in addition to the 90- and 180-degree options previously available.

• Two new options are available for modeling turbulence using Large Eddy Simulation (LES). These options are the classic Smagorinsky method and the (recommended LES option) of Dynamic Structure method. Details are available in the Forte User Guide.

• Eight more fuels are available to use with the Flame Speed Library. These fuels are methyl butanoate, methyl crotonate, CO, cyclopentadiene, 2-methyl-2-butene, n-decane, o-xylene and iso-hexane.
Further improvements to parallel scaling, which reduce simulation turnaround time for large numbers of cores.

### 7.2. Resolved Issues since Forte 18.0

#### Simulation Interface

- Improved the validation and correction logic for correcting input profiles that do not exactly cover \([0,\ldots,720]\) crank-angle range for 4-stroke engines or \([0,\ldots,360]\) for 2-stroke engines, as required for engine cyclical simulations. [DE142241]

- Added the ability to export data using the Models->Chemistry->Flame Speed->Table Lookup->Flame Tables->Editor menu. [DE142515]

- Improved the behavior of the Initial Condition panel, such that data is displayed automatically, without any need to push the Apply button. [DE143321]

- Fixed an issue that caused a failure during project saving after importing an STL geometry. [DE143605]

- Fixed an issue that prevented user specification of initial particle surface site fractions in the Simulation Interface, for use by the Moment Method in soot particle size tracking. [DE143978]

- Corrected the default turbulence-parameter input selections for Inlet and Outlet boundary condition panels. [DE145369]

- Fixed an issue that prevented the Surface Checker from being run multiple times when issues are found by the check. [DE146258]

#### Job Submission, Monitoring, and Running Options

- Fixed an issue that resulted in an error in reading the *.ftind files, when the mesh exceeded 10 million cells.

- Fixed a problem using wall-sampling probes with spray. [DE143608]

- Addressed issues in automatic mesh generation partitioning logic that caused runtime failures when running on a large number of cores, for some specific engine setups. [DE143407, DE146410, 147075]

- Corrected the message reporting the Global Mesh Size in the MONITOR file at the start of the simulation. [DE143639]

- Addressed an issue in the parallel load balancing component that caused a failure when running across multiple nodes using a particular version of Infiniband on a Linux cluster. [DE144593]

- Moved the Monitor Probe data storage, so that the time history is not included in the *.ftavg file, to reduce the file size when many Monitor points or probes are included. [DE145158]

- Removed the inconsequential message regarding “Server Out of Loop” that was being reported at the end of the run in the Monitor text. [DE146363]

#### Engineering Models and Computation

- Fixed a bug causing an access violation during the simulation when the auto-ignition-induced flame-propagation model is initiated. [DE144334]
• Fixed a bug causing prechamber regions defined as separate regions from a neighboring chamber from being properly handled for flame propagation and chemistry calculations. [DE143312]

• Fixed an issue with automatic mesh generation that caused a runtime failure. [DE144743]

• Corrected the initial velocities applied for closed-cylinder, 90-degree periodic-boundary, automatic-mesh-generation cases when a swirl-ratio is specified. [DE147612]

7.3. Supported Platforms

Information about present and future operating system and platform support is viewable via the ANSYS website.

Information about present and future operating system and platform support is viewable via the ANSYS website.
Chapter 8: Chemkin-Pro Release Notes

The following sections contain release information for Release 18.1 of Chemkin-Pro:

8.1. New Features and Enhancements
8.2. Resolved Issues since 18.0

8.1. New Features and Enhancements

Chemkin-Pro

• Added the option to consider real-gas effects, based a cubic form of the equation of state (EOS), for 0-D and plug-flow reactor models, as well as flame simulators. The option is available when the thermo-dynamic data for the chemistry set includes the necessary real-gas data for all or for major gas-phase species in the chemistry set.

• In addition to the default inlet boundary condition that allows for back diffusion, it is now possible to specify a fixed-composition inlet for the flame simulators, including: Premixed Laminar Burner-stabilized Flame Simulator, Premixed Laminar Burner-stabilized Stagnation Flow Flame Simulator, Flame Speed Simulator, Opposed-flow Flame Simulator, and Extinction Flame Simulator.

• Added the option to specify a heat-transfer profile to control the wall heat transfer for the internal combustion engine (ICE) spark-ignition zonal model, where a separate profile may be applied to the unburned and burned regions. [DE143686]

• In the multi-zone HCCI engine model, changes were made to allow more flexibility in specifying the initial conditions in each zone, including:
  – Specification of different initial compositions for each zone, which can be entered in the multi-zone table in the reactor-model setup panel. [DE143994]
  – Initial fuel-air equivalence ratios and exhaust-gas recirculation (EGR) rates can now be specified as varying from one zone to another; this information can be entered in the multi-zone table in the reactor-model setup panel.

• Added a capability to calculate diameter-based coagulation efficiency for particle collisions when using the Sectional Method for Particle Tracking.

• Added the option to view surface reaction rates when post-processing the Shear Flow Reactor results in the Chemkin Visualizer. [DE144006]

• Added the option to display 2-D Contour Plots in the Chemkin Visualizer.

Reaction Workbench

• Allow the Multizone Extraction Utility, which sets up the Chemkin-Pro Multizone HCCI Engine Model initialization based on an imported CFD solution, to use an ANSYS Forte Solution based on automatic mesh generation. [DE141349]
8.2. Resolved Issues since 18.0

This section lists issues and limitations that existed in previous releases, but that were resolved and removed in Release 18.1.

Chemkin-Pro

- Fixed the calculation of density for Plasma Plug-flow Reactors, which resulted in inconsistent pressure and density values for high levels of electron density. [DE142401, 142403]

- For reactor networks involving Plasma reactor models, assure that the electron temperature is set in the downstream reactor to be consistent with the upstream value. [DE142402]

- For Flame Simulators, deprecated the option to use a “TRACE” species formulation for transport properties from the User Interface. Also, added an error if this option is selected in combination with a user-defined Lewis number to avoid an un-known result of incompatible options. [DE142660]

- Fixed an issue that caused incorrect units to be provided for variables when using the Chemkin Visualizer. [DE145085]
Chapter 9: FENSAP-ICE Release Notes

The following sections contain release information for ANSYS FENSAP-ICE 18.1.

9.1. New Features and Enhancements in ANSYS FENSAP-ICE
9.2. New Limitations in ANSYS FENSAP-ICE
9.3. Resolved Issues and Limitations
9.4. Beta Features

9.1. New Features and Enhancements in ANSYS FENSAP-ICE

FENSAP/DROP3D

• Rotational periodicity algorithm improved:
  – Cases with large periodic angles (>30 degrees) can now be run with the default CFL number.
  – Existing small angle rotationally periodic cases converge better.
  – Modifications to the stabilization scheme for the centerline nodes improve solution stability and accuracy.

• Improvements in the load-balancing algorithm increase the parallel speed-up of the wall distance calculation with large numbers of CPUs.

• 6000- and 7000-type BCs are now frozen during mesh morphing with ALE so that the shape of those surfaces can be preserved.

• Inlet and exit surfaces that are in contact with ice-accreting walls are now included in the list of surfaces that undergo grid displacement due to ice growth.

• Negative-volume elements detected in a grid are now listed in the log file with their element id rather than being written into individual element grid files.

• Rigid motion also supported in the unsteady icing mode:
  – The unsteady solution mode can now simulate oscillation and translation of the grid while surfaces are morphing due to ice accretion.
  – Rigid motion is applied uniformly to the entire grid using a prescribed set of equations.
  – Cyclic translation and rotation modes are available, which can be used to simulate icing on pitching and plunging wings and airfoils.
  – The unsteady icing tutorial was updated to include a demonstration of rigid motion.

• Appendix O: The 10-droplet diameter distributions recently proposed by the FAA in AC 25-28 for the FDE and FRE environments and their MVD <40 micron and MVD >40 micron sub-classes were introduced to complement the distributions already offered by FENSAP-ICE.
• Introduced a warning message and disabled automatic remeshing for grids containing prisms alongside any of the other element types (hexa, tetra, pyramid) on the same boundary condition family.

• Dry initialization now initializes droplet velocities to air velocities. Previously this was only possible by setting droplet initial velocity components to 0 0 0. Now, doing so will set droplet initial velocity components to 0 0 0.

**C3D/CHT3D**

• C3D now supports non-conformal grid interfaces inside solid grids.

• CHT3D can work with solid grids containing non-conformal interfaces.

• CHT3D with roughness can now use the variable roughness computed with the beading model in ICE3D.

• Improved automatic time-stepping for electro-resistive heating.

• The execution of C3D automatically stops and the solution file is written when a non-physical temperature is detected at a node.

**ICE3D**

• The constant time-stepping algorithm was stabilized by introducing conservative flux-limiting. This is beneficial:
  – In rotational cases where substantial film flow can push the auto-time step to very low values to satisfy the stability condition.
  – To permit the use of larger time steps.
  – This option must be used with care: large constant time steps may decrease solution accuracy.

• New boundary condition type - Enabled-sliding:
  – Enables film flow calculations on surfaces where the nodes are allowed to slide on the reconstructed CAD of the surface.
  – Surfaces on which ice shape is not computed can now be included in the water film calculations.
  – Film flow and reinjection on sliding hub and shroud surface sof turbomachinery stages are now included in the solution.

• Manual execution termination now forces the mass and heat flow tables to be written to the log files.

**OPTIGRID**

• Improved read/write support for Fluent:
  – Periodic boundaries are now fully supported.
  – To reduce file size of the interpolated solution file (.dat), a new Interpolated solution = Subset option is available in the Optigrd input panel. The output file will include only the variables required for a Fluent simulation restart.

• Support for multi-material or multi-domain FENSAP grid adaptation:
- The domains will be conserved in the output grid.
- Enables mesh adaptation for the solid domain in C3D and CHT3D.

• ADAPT-DROP adaptation sequence can now also be used with ice crystals.

ICEM-CFD

• The FENSAP mesh export tool in ICEM CFD was updated and supports the following options:
  - ASCII or binary file output.
  - Additional surface boundary condition identifier: PERIO boundary condition, for periodic BC 5000.
  - Volume boundary condition identifiers: MAT to separate C3D domain materials, DOM to write multi-domain grids for FENSAP.

• The writefensap ICEM-CFD conversion tool included in FENSAP-ICE was updated to support the same mesh writing options offered by the ICEM CFD User Interface.

FLUENT

• Conversion to FENSAP (fluent2fensap):
  - Import of the periodicity from Fluent is now supported automatically.
  - Centerline topologies are also supported (tetra/prism).

• Conversion from FENSAP to Fluent (fensap2fluent):
  - FENSAP grids converted to Fluent will have proper periodicity shadow zone pairs.
  - Centerline is also supported.
  - Rotational periodic grids require the axis of rotation and a shadow zone periodicity type to be set-up in Fluent.

GENERAL

• FENSAP-ICE can now leverage ANSYS licensing for academic users.

• Windows:
  - The software can be installed and used from network UNC paths (\\fileserver\path).

WORKBENCH ADD-ON

• Component systems:
  FENSAP, DROP and ICE can be set-up individually as component systems.

• Mesh import system:
  - The Mesh Import system can be connected to ANSYS Meshing, Fluent Meshing and ICEM CFD (ICEM CFD import in Workbench is also handled through the Fluent format).

• FENSAP-ICE analysis system - sequences and restarts:
– It is now possible to connect a chain of FENSAP-ICE analysis systems and restart from the previous shot with simple settings (Input mode = Workbench connection and Restart from input = Enabled).

– The analysis system will restart from the results of the previous system (airflow, droplet, icing), and, if applicable, will use the beading information from the previous shot.

9.2. New Limitations in ANSYS FENSAP-ICE

DROP3D

• Setting initial droplet velocity to 0 0 0 will negate the Dry Initialization setting.

ICE3D

• When running unsteady icing, the heat fluxes will be updated only at solution write. For most accurate results, set the solution output to every 1 iterations.

MULTISHOT-REMESHING

• Multishot with remeshing sequence does not support particle distributions when breakup is enabled.

CHT3D

• When using FLUENT as the flow solver, the built-in journal file used to apply temperature onto the walls might not be compatible with your physical model (notably: for rotating domains and moving walls). The issue can be corrected by crafting a case specific journal file and specifying it instead of the $JOURNAL variable in the FLUENT command line settings of CHT3D. The CHT3D centercone tutorial, using FLUENT, provides details on the method chosen to avoid this issue.

OPTIGRID

• Fluent mesh adaptation only support single domain case files. Case files with multiple "interior" facet zones are not supported.

FENSAP-ICE TURBO

• ICE3D-TURBO: At least one of the walls of each row should have icing enabled.

• DROP3D-TURBO: In a multidomain turbomachinery run, the solver execution on a specific row may converge and end before the specified number of iterations required for printout. In this case, the boundary conditions required for the next row are not written and the code stops prematurely. The temporary solution is to increase the frequency of print-outs to fall within the total iteration count.

WORKBENCH

• Some FENSAP-ICE modules and capabilities are currently not available within Workbench, and can only be performed in standalone mode:
  – CHT3D: Anti-icing and de-icing simulations.
  – FENSAP-ICE TURBO: Turbomachinery analyses.
  – OptiGrid: Grid optimization.
• FENSAP-ICE Preferences and Unit settings are only available through the FENSAP-ICE Project Manager, which can be accessed by right-clicking on the first cell of a FENSAP-ICE based system and selecting Show project manager.

• Using FENSAP-ICE with the Remote Solve Manager (RSM) is currently not supported.

• Connecting Mesh and Geometry component systems directly to FENSAP-ICE based systems is not a currently supported feature.

• The Geometry and Mesh cells are currently not part of Fluid Flow - Icing analysis systems.

• Some common Workbench options are not yet offered with Fluid Flow - Icing analysis systems, including:
  – Clear Generated Data
  – Import Final Data

• The following Workbench commands are limited in their functionality or may not function as expected with CFX/Fluent/FENSAP-based icing systems:
  – Reset - It is effective only for the Setup Flow cell in a Fluid Flow – Icing (FENSAP) analysis system and/or for the Setup Droplets cell in a Fluid Flow - Icing (CFX/Fluent) analysis system: the local and generated data for the entire Fluid Flow - Icing analysis system is deleted, such that the states of the downstream cells appear as Unfulfilled (û).
  – Interrupt – does not stop a FENSAP-ICE calculation immediately as expected. Instead, the Interrupt option from the Workbench’s Progress view safely stops the current cell calculation after completing the total number of iterations specified. To stop a calculation immediately, use the Interrupt option from the Workbench’s Progress view together with the Stop command from the FENSAP-ICE graphical window. When you interrupt a calculation, the state of the cell appears as Up-to-Date and not as Interrupted, Update Required.
  – Abort - This option does not function by itself. However, when you Abort a calculation from the Workbench’s Progress view, together with the Kill command from the FENSAP-ICE graphical window, the calculations will stop immediately. When you Abort a calculation, the state of the cell appears as Up-to-Date and not as Update Failed, Update Required.
  – Shared data connections - are not currently supported.
  – Transfer data connections – Currently supported connections for the Fluid Flow – Icing analysis systems are:
    → Fluid Flow – Icing (FENSAP) - Result component systems.
    → Fluid Flow – Icing (CFX/Fluent) – Result, CFX and Fluent component systems.

• Stop - will stop the calculation immediately, but will not interrupt the Workbench update process for the downstream cells. When you Stop a calculation, the state of the cell appears as Up-to-Date and not as Interrupted, Update Required. Currently, the Interrupt option from Workbench’s Progress view, can only work in combination with the Stop command.

• Kill – See Abort option limitation above.
• When viewing FENSAP-ICE data using ANSYS CFD-Post, only one CGNS file can be postprocessed at a time; multiple simultaneous CGNS files are not supported inside ANSYS CFD-Post. To do data set comparisons between two or more result files use VIEWMERICAL instead.

• Currently the Custom feature under Import Options inside the Properties window for CFX/Fluent-based Icing systems is not fully supported, and it is recommended to use the automatic shared data connection option.

9.3. Resolved Issues and Limitations

FENSAP/DROP3D

• Fixed an issue where automatic remeshing after icing was disabled for grids that contained wall boundaries with non-prism elements.

• Corrected some problems with the actuator disk when running with the conservative energy equation:
  – The enthalpy injection to match the work done on the flow by the actuator disk was not yet in place in the conservative energy equation.

• Bug fix in the heat flux calculation that caused some cases to hang when searching for the 2nd order node stencil in the boundary layer.

• Corrected a bug introduced in 18.0 in the stopping criteria check for DROP3D where, for some cases, the code continued to run even after the convergence criteria was met.

C3D/CHT3D

• Bug fix for CHT3D that occurred with periodic grids when the new boundary condition system introduced in 18.0 triggered a crash at the start of the C3D execution (solid heat conduction), (pseudo-periodic elements on the interface lacked multipass-interpolated boundary condition data.

ICE3D

• A bug was fixed in the grid displacement algorithm to correct an issue where in some cases the displacement progress remained stuck at a certain percentage.

• Fixed the problem of slow file I/O when running on some Windows platforms.

• Corrected the integration tables for 2D periodic grids where the integrated values were shown as half of what they should have been.

OPTIGRID

• Correction of possible issues with periodic grids.

FLUENT

• Fluent file conversion only processed absolute frame solutions. A warning was displayed when converting zones with relative frame solutions. With 18.1, fluent2fensap will automatically convert relative frame velocity to the absolute frame, as required for each cell zone.

WORKBENCH ADD-ON
• Workbench project access path or project name with spaces are now supported with the FENSAP-ICE workbench add-on.

9.4. Beta Features

(Available only if advanced/beta features are enabled in the FENSAP-ICE preferences - Settings → Preferences → General)

• CFD-Post native reader for FENSAP-ICE solution files. This mode permits to load FENSAP-ICE (grid and solution) directly in CFD-Post:
  
  – In the FENSAP-ICE preferences:
    
    → CFD-Post - Native can be selected as the default post-processor.
    
    • Requirement: CFD-Post 18.1 must be used, and the beta options need to be enabled also in the CFD-Post Preferences panel, prior to the first launch from FENSAP-ICE.

  – In the FENSAP-ICE workbench add-on:
    
    → The FENSAP-ICE / Results component can connect to CFD-Post using the native reader. Choose the Native (Beta) options in Results settings - Type.

• OptiGrid Workbench component system:

  – In FENSAP-ICE Workbench add-on: Can be used to adapt a FENSAP grid (mesh smoothing), a FENSAP grid with the airflow solution, or a Fluent grid with the airflow solution.

  • An ice-shedding option added in de-icing simulations, based on the amount of melt-water obtained on a surface element, was introduced.

  • An ice-shedding option was introduced in ICE3D to compare the adhesive forces between the ice and the surface to the aerodynamic and body forces experienced by the mass of ice. Currently ice-steel and ice-aluminum interface properties are available to determine the adhesive force between the surface and the ice. The shedding of ice is triggered using a user-defined percentage of delaminated ice mass vs. total accreted ice mass.
Part III: ANSYS Electronics Products

Release notes are available for the following ANSYS Electronics products:

Icepak (p. 57)
Chapter 1: Icepak Release Notes

Release 18.1 of the ANSYS Icepak application offers most of the capabilities from previous releases plus many new features and enhancements.

• Introduction (p. 57)

• New and Modified Features in ANSYS Icepak 18.1 (p. 57)

• Resolved Issues and Limitations in ANSYS Icepak 18.1 (p. 57)

1.1. Introduction

ANSYS Icepak 18.1 is a release of ANSYS Icepak that has new features and resolved issues and limitations.

1.2. New and Modified Features in ANSYS Icepak 18.1

• Graphical User Interface
  – Added capability to perform transient coupling between Icepak and HFSS, Maxwell, and Q3D.

1.3. Resolved Issues and Limitations in ANSYS Icepak 18.1

• Import/Export
  – When importing a .brd trace file, the metal fractions calculated do not match the trace layout. (136134)
  – ECXML assemblies’ offsets are not being exported properly. (140488)
  – When importing Gerber files, values displayed on the vias tab have incorrect units. (140630)
  – Importing CTM files does not function when importing files with fewer than 100 sources. (141429)
  – When importing geometry that was exported from SpaceClaim, only four significant digits are imported. (141449)

• Meshing
  – The mesher fails after changing a polygon to a prism block for specific model. (117517)
  – For a specific model, the multi-level mesh for an inclined cylinder is skewed and may cause the solution to diverge. (121592)
  – When a model contains assemblies using different mesh type, the Mesh type in the Mesh control panel changes after a solution is complete (141347)
  – For a specific model, the HD mesher fails to mesh an inclined plate. (142234)

• Model Building
When an internal opening is placed in partial contact with a solid object (e.g., a solid block), the .srp file is set to zero enthalpy for all openings. (122552)

Sometimes monitor points created from object sides are not located in the center of the side. (142075)

**Parametric Trials**

- Terminating the convergence plot of one parametric trial terminates all subsequent trials. (118962)
- For parametric trials, if the trial name contains a space, a warning message is displayed. (127660)
- Clicking the Trials tab after selecting Selected Values in the Parameters and optimization panel causes Icepak to hang. (127729)
- When running parametric trials with Fast trials enabled, the residuals are not plotted and the parametric table results are not populated. (141646)
- Double-clicking Restart ID in the Parameters and optimization panel causes an error. (142796)
- For a specific model containing a heat sink, the heat sink parameters (fin thickness, fin spacing, fin length, etc.) are unable to be parameterized. (143559)

**Post-processing**

- For a specific model's transient solution, the surface monitor does not function as expected. (143000)
- For a specific model, when run in parallel, post-processing contours are displayed incorrectly. (143548)
- Dynamic plotting does not happen inside a selected window through zooming at solving. (145425)

**Reports**

- For a Source object, the specified and calculated power do not match even after convergence. (125160)
- For a transient simulation, when using one of the functions (for example, linear, exponential, etc.) to specify any parameter as a function of time, the units for the coefficient 'a' are incorrect. (135764)
- For a PCB with imported ECAD data, specific heat does not appear to be dependent on metal fractions. (142049)
- There is no unit for relative humidity (RH%) for species analysis of water. (142492)

**Solution**

- For a specific model solving for joule heating, the ground boundary condition is applied to the wrong side when viewing the results. (111172)
- Changing the value for a solid material property is not reflected when performing a full data restart. (119397)
- The block type for a solid block automatically changes to fluid when specifying a material name as the same as an existing material, though with different case (i.e., lower case or upper case). (138369)
- For a specific model, the Fluent solver fails to start despite acceptable mesh. (141453)
- For a specific model, the solution converges in serial mode but diverges in parallel. (142046)
- For a specific model, the Fluent solver fails to start despite acceptable mesh. (142252)
- For a specific model coupling with Slwave, a high temperature is reported. (142256)
- For transient models, results for surface monitors are not displayed. (142281)
- Full data restart fails when there is Joule heating is modeled. (142490)
- Full data restarts do not work when Species transport is also being solved. (142614)
- For a transient solution, the recirculating opening time interval setting causes divergence of direct ordinates radiation equation. (142870)

• Solver Setup

- While natural convection is disabled on the Basic parameters General setup tab and ideal gas law is enabled on the Advanced tab, natural convection settings are applied. (143304)
Part IV: ANSYS Geometry & Mesh Prep Products

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

- DesignModeler (p. 63)
- SpaceClaim (p. 65)
- CAD Integration (p. 67)
- Meshing (p. 69)
- IC Engine (p. 71)
- ICEM CFD (p. 73)
- Fluent Meshing (p. 75)
Chapter 1: Geometry Release Notes

This section summarizes the new features in DesignModeler Release 18.1. Topics include:

Frame Alignment Property

You can use the new **Frame Alignment** property to override the default frame alignment that DesignModeler would select on its own. It is important for frame alignment to be set correctly because it is used during meshing to determine the orientation of individual elements. Most of the time DesignModeler assigns the frame correctly on its own, but it may not always be able to make the optimal assignment.

For more information, see Cross Section Alignment in the DesignModeler User's Guide.

Solid/Shell Surface Coatings

DesignModeler supports a new multibody part configuration for analyses involving solid/shell surface coatings. In this type of analysis, surface bodies that are coincident to the faces of solid bodies are defined, and the surface bodies and solid bodies are grouped into the same part. The surface bodies merge with the faces of the solid bodies, but transfer downstream with the surface bodies intact to simulate coatings.

For more information, see Shared Topology and Multibody Part Configurations in the DesignModeler User's Guide.
Chapter 2: SpaceClaim

For detailed information specific to SpaceClaim 2017.1, see the SpaceClaim 2017.1 Release Notes on the ANSYS Customer Portal (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

This section summarizes the new features in CAD Integration Release 18.1.

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

Geometry Interfaces Update for New CAD Releases

Geometry interfaces are updated to support new CAD releases including:

- CATIA V6 R2016x (Reader)
- Creo Elements/Direct Modeling 20.0 (Plug-in)
- Creo Parametric 4.0 (Plug-in)
- Inventor 2017 (Reader)
- NX 11.0 (Reader)
- Parasolid 29.0 (Reader)
- Solid Edge ST9 (Reader)
- SolidWorks 2017 (Plug-in)

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. Changes in Interface Terminology and Product Behavior from Previous Releases
4.2. Contact Enhancements

Many of the enhancements detailed in the Mechanical Application Release Notes (p. 3) are also relevant to the Meshing application.

4.1. Changes in Interface Terminology and Product Behavior from Previous Releases

This release includes the following changes in interface terminology and product behavior:

- The Element Midside Nodes property was renamed Element Order, and its Dropped and Kept options were renamed Linear and Quadratic respectively. This change affected the Element Midside Nodes property that was located in the Defaults group in the Details View of Mesh objects, in the Details View of mesh Method controls, and in the Details view of Gasket objects.

- The Repair Overlapping Contact Regions option was renamed Check Overlapping Contact Regions, and its behavior was changed. This option no longer repairs overlapping contact regions automatically. Instead, it checks for overlapping contact regions and identifies any it finds, so that you can resolve them manually. The option to repair the regions automatically is no longer needed due to the introduction of the Face Overlap Tolerance and Edge Overlap Tolerance options, which give you greater control over contact pair generation.

- The Initial Size Seed option was changed. In previous releases, you could set Initial Size Seed to Active Assembly, Full Assembly, or Part. For this release, Active Assembly was eliminated, and you can choose between Assembly (equivalent to Full Assembly in previous releases) or Part. Additionally, the Initial Size Seed option is exposed only when Size Function is set to Adaptive. For all other size functions, Assembly is set internally and cannot be modified.

4.2. Contact Enhancements

This release includes the following enhancements related to contact:

- The new Face Overlap Tolerance option enables you to set your preference for the minimum percentage of overlap at which a contact pair will be created for two overlapping faces. This setting enables the software to obtain more precise contact pairs during automatic contact generation based on a default tolerance that is appropriate for your simulation type. The Face Overlap Tolerance setting in the Connections group of the Mechanical Options panel determines the default tolerance. You can modify the Face Overlap Tolerance property in the Details view of the Connection Group folder to override the default for the current model.

- The new Edge Overlap Tolerance option enables you to set your preference for the minimum percentage of overlap at which a contact pair will be created for an edge and a face that overlap. This setting enables the software to obtain more precise contact pairs during automatic contact generation based on a default
tolerance that is appropriate for your simulation type. The **Edge Overlap Tolerance** setting in the **Connections group of the Mechanical Options panel** determines the default tolerance. You can modify the **Edge Overlap Tolerance** property in the Details view of the **Connection Group folder** to override the default for the current model.

- You can select the new **Only Beam Edges** option for the **Face/Edge** property so that face to edge connection uses only edges of beam bodies to determine connection with all faces. In the **Connections group of the Mechanical Options panel**, you can set the **Face/Edge** property to **Only Beam Edges** to make this the default for face to edge connection detection. You can modify the **Face/Edge** setting in the Details view of the **Connection Group folder** to override the default for the current model.

- You can now drag and drop the **Contacts** folder onto the **Mesh** object to create a **Contact Sizing control** for each contact region in the folder automatically.
Chapter 5: IC Engine Release Notes

Release 18.1 has no new features or enhancements.
Chapter 6: ICEM CFD Release Notes

This section summarizes the new features in ICEM CFD Release 18.1. Topics include:

- 6.1. Highlights of ANSYS ICEM CFD 18.1
- 6.2. Documentation

6.1. Highlights of ANSYS ICEM CFD 18.1

Release 18.1 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 18.1 includes new features and improvements in the following areas:

- 6.1.1. Multizone Block Editing improvements
- 6.1.2. Transfer Blocks
- 6.1.3. Pipe Modeler
- 6.1.4. Usability Improvements

6.1.1. Multizone Block Editing improvements

The following enhancements were made to improve block editing:

- **Blocking → Split Block → Imprint Face** was enhanced. It now supports multiple free blocks (2D) or multiple free faces (3D) as a target(s). This enhancement also applies to **Split Free Block → By Imprint**.

- Defeaturing was improved.

- Support for embedded edges in 2D.

6.1.2. Transfer Blocks

A control was added to the **Edit Block** dialog box. You can now **Transfer Blocks** to the Output Block topology by selection.

6.1.3. Pipe Modeler

An item was added to the **File → Blocking** menu that allows you to **Create Pipe Blocking from File**. A properly formatted text file can be used to define geometry and blocking for a network of pipes.

6.1.4. Usability Improvements

The following enhancements were made to improve usability:

- The **Blocking → Pre-Mesh → Find Worst elements** interface was expanded and improved. You can now specify a range and the graphics display allows you to view the elements or step through the identified elements.
• Replay scripting for isocurves, midline and arc was improved.

• The **Import Model** process now allows you to import a **Local Coordinate System** along with your model from certain CAD packages.

• A new menu item was added to **Parts** in the **Display tree**. You can now **Show Parts Info** to see which parts contain geometry/mesh/blocking along with other statistics, without selecting the part(s) in the graphics window.

### 6.2. Documentation

All documentation for **ANSYS ICEM CFD Release 18.1** 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](http://support.ansys.com/training). The Customer Portal also contains links for training, and PDF versions of the Tutorial manual or of individual tutorials.
Chapter 7: Fluent Meshing Release Notes

The following sections contain release information for ANSYS Fluent Meshing Release 18.1:

7.1. Changes in Product Behavior from Previous Releases
7.2. New Features

7.1. Changes in Product Behavior from Previous Releases

• When using Auto Mesh after importing a CAD or .tgf file, the following default settings will be applied:
  – The option to retain dead cell zones is switched off, by default.

    The Auto Mesh dialog box contains an option to Keep Dead Cell Zones, which is switched off by default. Also, the option to Delete Dead Zones in the Tet and Hexcore dialog boxes is enabled.

  – For hexcore meshing, the default number of Buffer Layers is 2 and the Peel Layers is 1.

  – When the sizing method is set to Geometric, the default Growth Rate will be set to 1.2.

• For zone-specific prism meshing, the following default settings will be applied:
  – The options for improving prism mesh quality by swapping or smoothing edges, and smoothing nodes are switched off by default.

    The options Swap Edges, Smooth Edges, and Smooth Nodes are switched off in the Improve tab of the Prisms dialog box.

  – The layer-by-layer improvement of cell quality by smoothing normals in the current layer is switched off by default.

  – The checking of left-handed faces is switched off by default.

• Tet initialization controls will now be respected for object-based meshing.

• The face zone name and ID will be retained when triangulating quad face zones with the replace option.

7.2. New Features

The new features available in the meshing mode in Fluent include enhancements to existing features, and improved robustness through defect fixes.

User Experience

The following enhancements were made:

• The Auto Mesh dialog box contains an option to Keep Dead Cell Zones.

• The Auto Mesh dialog box contains additional Volume Fill Options for the volume fill methods available.
The option to select Octree-based hexcore meshing is available via the Auto Mesh and Hexcore dialog boxes.

Parallel Meshing
The following enhancements were made:

• The toolbars are now available in parallel mode.
• You can write the mesh file for the distributed mesh.
• Periodic meshes are now supported.

Scripting Improvements
New API functions are available for the following:

• Defining and setting up controls leading to propagation of sizes along the surface instead of the volume (geodesic sizing).
• Wrapping for automotive applications for underhood thermal management (UTM) and external-aero analysis.
• Reporting selected mesh check statistics.
• Marking faces based on deviation from the size field and node degree.

Miscellaneous Improvements
The following improvements were made:

• For object-based Auto Mesh using tet/poly volume fill, you can choose to use the maximum cell size specified instead of recomputing the value based on the objects, when the volume mesh is generated.
• You can now use specific sizes for finding and fixing holes while wrapping the model.
• A new option for Quick Edge Proximity enables speed up of proximity calculations for geometries with a large number of small feature edges.
Part V: ANSYS Simulation Products

Release notes are available for the following ANSYS Simulation products:

- Workbench (p. 79)
- ANSYS Customization Toolkit (ACT) (p. 83)
- RSM (p. 87)
- EKM (p. 89)
- DesignXplorer (p. 93)
- ANSYS Viewer (p. 95)
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

1.1. ANSYS Workbench

Enhancements were made to the following areas:

1.1.1. Mechanical APDL Enhancements
1.1.2. ANSYS Workbench-Remote Solve Manager Enhancements
1.1.3. ANSYS Workbench-EKM Enhancements

1.1.1. Mechanical APDL Enhancements

The following enhancements were made to the Mechanical APDL component system for Release 18.1:

- When you right-click the Analysis cell and select Add Reference File(s), you can use Ctrl-click or Shift-click to select multiple files when adding reference files from the browse window.

1.1.2. ANSYS Workbench-Remote Solve Manager Enhancements

No enhancements were made in Workbench with regard to Remote Solve Manager.

1.1.3. ANSYS Workbench-EKM Enhancements

No enhancements were made in Workbench with regard to EKM.

1.2. External Connection

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

- The tables in Appendix A. ANSYS Workbench Component Inputs and Outputs were updated and a new "Topology Optimization" table was added.

- The table in Appendix D. Addin Data Types and Data Transfer Formats was updated.
1.3. Engineering Data Workspace

For Release 18.1, the Engineering Data Workspace has no new features or enhancements.

1.4. External Data

No enhancements were made to the External Data add-in.

1.5. External Model

For Release 18.1, External Model has no new features or enhancements. However, you may wish to refer to the Model Assembly and External Model Enhancements section of the Mechanical release notes for enhancements in the Mechanical application that are based on importing data through the External Model system.

1.6. Enhancement to Mechanical Model Cells

For Release 18.1, the Mechanical Model Cell has no new features or enhancements.

1.7. FE Modeler

Release 18.1 for FE Modeler has no new features or enhancements.

1.8. System Coupling

No enhancements were made to System Coupling.

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 Introduction in the TurboSystem User’s Guide.

These release notes cover:

- Performance Map System
- Turbo Setup System
- 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.
Chapter 2: ACT

The following enhancements are available in ANSYS ACT 18.1. All referenced topics are in the ANSYS ACT Developer's Guide.

ACT Console Enhancements

In an ongoing effort to improve the ACT Console, it now provides the following new features:

• Ability to search the command history for text that is currently entered in the command line, allowing you to easily find this text in other command entries. For more information, see Using the Command History.

• Suggestions for autocompletion display square icons for member types and circle icons for return types, making command-building more intuitive. For more information, see Using Autocompletion.

Guided Simulations and Guided Setups for AIM

When creating an extension for AIM, you set the context attribute in the <wizard> block to Study. This creates a custom template. For more information, see Creating a Custom Template.

If you want to create a guided simulation or guided setup, you must also set an additional attribute in the <wizard> block.

• A guided simulation displays your own end-to-end custom workflow in a persisted view, remaining open until you close it. To create a guided simulation, you set the persistent attribute to true. For more information, see Creating a Guided Simulation.

• A guided setup displays on the Guide Me context (right-click) menu when a compatible object and geometry entity are selected. To create a guided setup, you set the subcontext attribute. For example, subcontext = "Physics Solution/Face" causes the extension to display on the Guide Me menu for a task group of type "Physics Solution" when an AIM user right-clicks a selected face. For more information, see Creating a Guided Setup.

ACT App Builder (Beta)

The ACT App Builder is a standalone utility for creating ACT extensions in a visual environment. Introduced in 18.0 as a beta feature, the ACT App Builder can be launched from either Workbench or AIM to create and edit XML code for an ACT extension within a graphical user interface. In 18.1, the useability of this beta feature is further enhanced. For more information, refer to the beta documentation on the ANSYS Customer Portal. After selecting Downloads > ACT Resources to display the ACT Resources page, expand the Help & Support section, where you will find a link for viewing the App Builder beta document.

More Efficient Method to Retrieve Mechanical Results (Beta)

You can use a beta version of a new method to retrieve Mechanical results for postprocessing. This method uses an external executable in addition to the Mechanical process. By using the postprocessing
API via this method, you transparently control the instantiation of the results reader and postprocess results without interfering with Mechanical, especially in the case of analysis over several time steps. This method improves postprocessing performance by avoiding unnecessary actions that could be automatically executed in a standard use of Mechanical. This method is available only with the Windows platform.

To enable or disable the use of this method, you set the variable `ExtAPI.UsesStandaloneActResultReaderImplementation` to `True` or `False`. This variable is set to `False` by default.

Example code:

```csharp
# Activate postprocessing via external executable
ExtAPI.UsesStandaloneActResultReaderImplementation = True

# Read the Stress from the specified result file
rst = r"C:\\test\file.rst"
reader = ExtAPI.Tools.GetResultsDataFromFile(rst)

nbSteps = reader.ResultSetCount
rs = reader.GetResult("S")
for i in range(nbSteps):
    reader.CurrentResultSet = i+1
    s = rs.GetElementValues(elemIds,False)
reader.Dispose()

# Deactivate postprocessing via external executable
ExtAPI.UsesStandaloneActResultReaderImplementation = False
```

As the result reader is in a separate process, to ensure good performance, it is necessary to use the two following methods to get the nodal results and the elemental results.

**Method to Retrieve Nodal Results (Beta)**

To retrieve nodal results efficiently, you must use one call to get the results for a set of nodes. The method `GetNodeValues` takes as arguments an array of node indices and returns an array with the values of each selected component of the required result at each node sequentially.

Declaration syntax:

```csharp
public double[] GetNodeValues(int[] nodeIds)
```

Where `nodeIds` is the array of integers containing the list of the node indices for which result values are required.

Example code:

```csharp
reader=ExtAPI.DataModel.AnalysisList[0].GetResultsData()
mesh=reader.CreateMeshData()
mesh=ExtAPI.DataModel.MeshDataByName("Global")
nodeIds = mesh.NodeIds
ru=reader.GetResult("U")
u = ru.GetNodeValues(nodeIds)
> nodeIds = [ 1, 2, ...]
> u = [ U1X, U1Y, U1Z, U2X, U2Y, U2Z, ...]
```

**Method to Retrieve Elemental Results (Beta)**

To retrieve element results efficiently, you must use one call to get the results for a set of elements. The method `GetElementValues` takes as arguments an array of element indices and returns an array with the values of each selected component of the required result for each element sequentially.

Declaration syntax:
public double[] GetElementValues(int[] elementIds, boolean computeMidSideNodes)

Where:

- **elementIds** is the array of integers containing the list of element indices for which result values are required.
- **computeMidSideNodes** indicates if the method is to return only values at corner nodes or values at both corner nodes and midside nodes. When set to False, only values at corner nodes are returned. When set to True, values at both corner nodes and midside nodes are returned.

Example code:

```java
reader=ExtAPI.DataModel.AnalysisList[0].GetResultsData()
mesh=reader.CreateMeshData()
mesh=ExtAPI.DataModel.MeshDataByName("Global")
elemIds = mesh.ElementIds
re=reader.GetResult("S")
s = re.GetElementValues(elemIds,False)
```

```java
>elemIds = [1, 2, ...]
```

In this example, each element has three nodes. The stress values are located at the nodes of the elements. Thus, $S_{ijX}$ is the value of component X for the stress result at node j of element i.

**Remarks**

In the case of a 2D geometry, the result array returned by the method `GetNodeValues` or `GetElementValues` stays dimensioned as for a 3D geometry. However, the values for the third dimension are dummy values.

For shell elements, the method `GetElementValues` returns the values for the three positions in this order:

- Bottom
- Top
- Middle

Example code:

```java
elemIds = mesh.ElementIds
re=reader.GetResult("S")
re.SelectComponents(["X"])
s = re.GetElementValues(elemIds,False)
>elemIds = [1, 2, ...]
>s = [ S11X_bottom, S12X_bottom, S13X_bottom, S11X_top, S12X_top, S13X_top, S11X_middle, S12X_middle, S13X_middle, S21X_bottom, S22X_bottom, S23X_bottom, ...]
```

**Limitations**

Degenerated elements are not supported.
Chapter 3: Remote Solve Manager (RSM)

The following sections contain release information for ANSYS Remote Solve Manager 18.1:

3.1. New Features and Enhancements
3.2. Deprecated Features
3.3. Issues Resolved in this Release
3.4. Known Issues and Limitations

3.1. New Features and Enhancements

If you have a previous installation of RSM, here are the key changes you should know about:

• In a single-node ANSYS RSM Cluster (ARC), cluster services are now started automatically when jobs are submitted to the single node. Also, when users submit jobs to a single-node ARC, password caching is no longer necessary.

• To configure a multi-node ANSYS RSM Cluster (ARC), you are no longer required to switch to advanced mode, as this is now the default mode. The concept of 'basic mode' no longer exists. Rather, ARC configuration is based on whether the cluster is a single-node or multi-node cluster.

• The RSM launcher service was renamed Ans.Rsm.Launcher.exe (formerly Ans.Rsm.JMHost.exe).

• On Windows, the command used to install the RSM launcher service is now AnsConfigRSM.exe -launcher (formerly AnsConfigRSM.exe -mgr). Similarly, the command to uninstall it is AnsUnconfigRSM.exe -launcher.

• On Linux, the rsmmanager service script was renamed rsmlauncher. For example, to start the RSM launcher service, you would run [RSMInstall]/RSM/Config/tools/linux/rsmlauncher start.

• A new migration utility enables you to automatically transfer cluster configurations, queue definitions and RSM application settings from a previous version of RSM to a new version.

• A new arcconfig migration command enables you to automatically transfer ANSYS RSM Cluster (ARC) cluster configurations, queue definitions, ARC-related application settings and ARC node configurations from one version of RSM to another, eliminating the need to redefine your ARC every time you upgrade to a new version.

• A new arcnodes command enables you to view the cores, memory and disk space available for job execution on ANSYS RSM Cluster (ARC) execution nodes at any given time.

• A new arcqueues command enables you to display the status and details of all defined ANSYS RSM Cluster (ARC) cluster queues.

• When defining an ARC cluster queue, a new -n <numJobs> option enables you to specify the maximum number of jobs that can be run from the queue.

• When multiple versions of RSM are running, a new ARC_ROOT environment variable enables you to point all versions of RSM to a specific ARC version.
• A standalone **RSM Job Monitoring** application is now available for monitoring RSM and Portal jobs outside of Workbench or EKM.

### 3.2. Deprecated Features

• Integration with Windows HPC running on HPC Server 2008 is no longer supported.

   The minimum supported operating system for Windows HPC is Windows Server 2012 R2 (Standard) with Microsoft HPC Pack 2012 R2.

### 3.3. Issues Resolved in this Release

• The issue of an RSM job monitor becoming temporarily unresponsive when attempting to delete a large number of jobs was resolved.

• When using the 'My Computer, Background' solve process setting on Linux, a solve job will no longer fail if the project folder contains a space.

### 3.4. Known Issues and Limitations

All issues and limitations known at the time of release are listed in the **Known Issues and Limitations** section of the *Remote Solve Manager User's Guide*. 
Chapter 4: ANSYS EKM Release Notes

ANSYS Engineering Knowledge Manager (EKM) 18.1 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 18.1:

4.1. New Features and Enhancements
4.2. Issues Resolved in this Release
4.3. Issues and Limitations

4.1. New Features and Enhancements

If you have used previous versions of EKM, Release 18.1 offers many significant changes and improvements:

Installation and Configuration

- When specifying custom content to be displayed in the Sign in to EKM box, you can use HTML markup if desired.

- When defining a property for a custom type, a new Hidden option enables you to specify that the property will not be displayed when users view or edit object properties.

- When defining a custom interface, a new multiFileUpload component can be used to add a browse feature for selecting multiple files to upload to EKM.

- When defining a script for a custom type or interface, if a macro returns a URL string, the URL will be fired by the browser on completion of the macro.

- When defining a custom application, a new Embedded option enables you to specify whether or not the application is associated with a process. Marking an application as Embedded prevents it from appearing in the application launcher. Such applications should only be launched from their associated processes.

- A Clear log button was added to the Script log dialog box when using the Administration > Scripting > Show log feature. This enables you to instantly clear the log when the maximum size is reached so that more data can be written to it.

- Support for REST technology was added. The REST API library enables you to develop applications that interact with the EKM server.

- The Sun Java Development Kit (JDK) that is set up by the EKM Server product installation has been upgraded to version 1.8.0_121.
The Jython library for Python scripting has been upgraded to version 2.7.0.

**Important**

In some cases, Python scripts used with EKM will need to be modified for compatibility with the new Jython version. In particular, statements that import Java classes and packages may be impacted. It is advised to use fully-qualified class and package names when accessing Java classes and packages from a Python script.

For example, the following statement imports a Java package:

```python
from java import util
```

To work correctly under Jython 2.7.0, this statement should be written as:

```python
import java.util as util
```

**Data Management**

- Data is now extracted from ANSYS Electronics Desktop Archive (.aedtz) files.
- When adding a String property to an object, specifying a URL as the property value renders the text as a hyperlink when viewing the object's properties on the Details tab.
- When performing an advanced search for job objects, the Status expression now displays a drop box of valid job statuses instead of a text box.
- When checking out an object that is under version control, you can optionally allow other users to check out and modify the object while you have it checked out.
- When viewing an image on the Image tab, you can now change the mouse button assignments for zooming, panning and rotating.

**Job Management**

- An LS-DYNA Batch job template was added for explicit dynamic analyses using LS-DYNA.
- When starting an interactive job, if you navigate away from the Job Monitor page or sign out of EKM while an interactive session is being created, you will receive an email notification once the interactive session is created (in other words, the job status has changed from Pending to Running).
- When running a CFX server-mode job, live monitors are now displayed when using an .mdef input file.

**Process Management**

- When creating a process template in EKM Studio, a new Select assignee using custom ui option in the process template attributes enables process owners to set task assignees using a custom interface instead of the standard dropdown list.
Usability Enhancements

• The method of selecting users when editing permissions, group membership and cache server assignments was simplified, and includes type ahead search functionality.

• You can now add an ANSYS Viewer File gadget to your dashboard which displays a selected ANSYS Viewer Archive (.avz) file in an embedded, interactive viewer.

4.2. Issues Resolved in this Release

Below are the major issues that were resolved since the release of 18.0.

• If you cancel a job that is executing, and then submit and execute another batch job that uses the same working directory, selecting the cancelled job in the Job Monitor no longer displays the monitors of the newly executed job.

• When a Fluent job running on Windows is interrupted from EKM, the job status is no longer reported as 'Failed'.

• When EKM is configured to use a cache server, and a Workbench user downloads files or gets changes from the EKM repository, the cache server is now correctly used for the requested operation.

4.3. Issues and Limitations

All issues and limitations known at the time of release are listed in Appendix A: Known Issues and Limitations in the EKM Troubleshooting Guide.
Chapter 5: DesignXplorer

The following enhancements are available in ANSYS DesignXplorer 18.1. All referenced topics are in the ANSYS DesignXplorer User's Guide.

Allowed Values for Continuous Input Parameters

In previous releases, a Manufacturable Values checkbox existed so that only the real-world manufacturing or production values that you specified for a continuous input parameter were taken into account during postprocessing. In 18.1, the Allowed Values option replaces this checkbox. When the default setting of Any is selected, any value within the range defined by the lower and upper bounds is allowed. If you want to further limit values, you change Allowed Values from Any to either Snap to Grid or Manufacturable Values.

- When Snap to Grid is selected, you specify a grid interval, which is the distance that must exist between adjacent design points when generating new design points.

- When Manufacturable Values is selected, levels display in the Table view so that you can specify the real-world manufacturing or production values to take into account during postprocessing.

For more information, see Defining Continuous Input Parameters.

Min-Max Search Improvements

The algorithm used to search the output parameter space for the minimum and maximum values now depends on the input parameters. If at least one input parameter is continuous with manufacturable values, DesignXplorer uses MISQP for the Min-Max search, which significantly improves performance. Otherwise, DesignXplorer uses NLPQL. In 18.1, the warning about long search times displays only if there is one or more discrete input parameters. For more information, see Min-Max Search.

Parameters Parallel Chart Enhancement

The Parameters Parallel chart for the Design of Experiments (DOE) component now supports sliders at the upper and lower bounds of each axis. When you place the mouse cursor on the graph, you can use the sliders that appear to easily filter for each parameter. DOE points that fall outside of the bounds defined by the sliders are dynamically hidden.

DesignXplorer Extension Compatibility Updates

The following extensions were updated for compatibility with the 18.0 release:

- Direct Optimization from RSO
- DOE from Correlation
- Full Factorial DOE
- Import Parameters and DOE
• LHS with Parameter Relationships

• MATLAB Optimizers

• Parameter Sweep

• Response Surface Reader

Compatibility updates for the 18.1 release are in progress and will be released promptly upon completion.

To access DesignXplorer extensions, go to the ANSYS App Store and filter the apps available by typing DesignXplorer in the Search Apps field and clicking the search button. To further filter the results, you can make a selection in the Product Version field to the left.
Chapter 6: ANSYS Viewer

The following sections contain release information for ANSYS Viewer 18.1:

6.1. New Features and Enhancements
6.2. Known Issues and Limitations

6.1. New Features and Enhancements

If you have a previous installation of ANSYS Viewer, here are the key changes that you should know about:

• A new **Viewports** action enables you to split the view window into two or four viewports, and load a file into each viewport. This enables you to view multiple files simultaneously.

• When probing an object with your mouse, a **Create Probe** action enables you to automatically turn the probe data (displayed in a tooltip) into dynamic text. This provides an additional way of marking up a file.

• A new **Settings** option enables you to change the mouse button assignments for zooming, panning and rotating.

• Support for legends was added.

• Improvements to the way in which **ANSYS Viewer Help** is presented make the help easier to access and browse.

6.2. Known Issues and Limitations

All issues and limitations known at the time of release are listed in the **Known Issues and Limitations** section of the **ANSYS Viewer User's Guide**.
Part VI: ANSYS AIM

The following enhancements are available in ANSYS, Inc. Release 18.1 (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 17.2 and Release 18.0 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). First-time users of the customer portal must register to create a password. See the ANSYS Customer Portal for information about ANSYS service packs, and any additional items not included in the Known Issues and Limitations document.
Chapter 2: Enhancements in AIM 18.1

The following enhancements were made to ANSYS AIM for Release 18.1.

- The ability to model bi-linear isotropic hardening plasticity to simulate plastic strains and permanent deformations of metallic materials.

- Enhanced solver file management for nonlinear structural and thermal simulations to reduce the amount of file storage.

- Enhanced solver messages for structural and thermal simulations to provide guidance and troubleshooting information.

- The ability to model temperature-dependent material properties, including electric conductivity, relative permittivity, and relative permeability for electromagnetic simulations.

- The ability to specify a temperature condition for electromagnetic simulations.

- The ability to simulate one-way thermal-magnetic coupling where volumetric temperatures are mapped from a thermal analysis to an electromagnetic simulation.

- Improved conservative algorithm for one-way electromagnetic-thermal coupling.

- The ability to specify an isotropic porous medium to model the momentum loss of flow through filters, perforated plates, packed beds, etc.

- Enhanced fluid solver convergence, which is less sensitive to the number of parallel processes.

- Improved robustness for fluid solution automatic initial guess when the model setup includes both pressure inlets and outlets, supersonic outlets, or mixed (supersonic/subsonic) inlet boundary conditions.

- Enhanced HPC scaling for conjugate heat transfer solutions via physics-based partitioning.

- The ability to post-process mesh quality metrics and solution residual information for fluids solutions.

- Enhanced user experience for specifying boundary layers (near wall refinement) for fluid simulations.

- Enhanced AIM start page for resuming existing projects, launching simulation process templates, and defining new simulation workflows.

- Enhanced simulation process templates that include multiple steps and automatically launch geometry modeling if required.

- The ability to select existing geometry tasks to build new simulations when running templates.

- Enhanced performance of transient updates, reducing the number of times updates need to be performed.

- Enhanced geometry modeling, including faceting tools for organizing, modifying, and smoothing faceted geometry for simulation.
Enhancements in AIM 18.1

- Enhanced custom applications that include guided steps.
- The option to expose custom applications in the context of a task/object via a right-click context menu.
- Enhanced user experience by further aligning user interactions between modeling and physics.
- The capability to export to a CSV file from solution monitors, and to select only the last two hundred points to make it easier to review recent changes in value.

Updates Affecting Code Behavior

Listed below are code changes implemented in Release 18.1 that may cause output that is different from the previous release.

As a result of a correction, Winding current is calculated differently than it was in Release 18.0, provided that the number of branches used in the 18.0 project is greater than one. Opening a project that was created and solved in Release 18.0, then re-solving in Release 18.1 may generate different results for the static calculation type. See the Winding section of the product documentation for more information.

The inflation feature in AIM was renamed to Boundary Layer for an improved user experience. Any user-defined apps that use Inflation will need to be updated to the new behavior.
Chapter 3: Enhancements in AIM 18.0

The following enhancements were made to ANSYS AIM for release 18.0.

- The ability to simulate magnetic frequency response including eddy/displacement currents.
- Enhanced material properties that include frequency dependent magnetic material properties.
- Enhanced magnetics template that automatically creates surrounding enclosure.
- The ability to simulate solid heating due to induction heating by using physics coupling to transfer heat rate to a thermal simulation.
- The ability to include field variables and quality functions (average, minimum, maximum, etc.) in expressions to define fluid conditions.
- The ability to transfer mesh and selection set data from AIM to Fluent via a Workbench project schematic connection.
- Enhanced AIM templates enable model transfer from AIM to either Fluent or Mechanical.
- Enhanced material properties that include the ability to define the state of matter - solid, liquid or gas - for a given material definition.
- The ability to continue a fluid solution from the previous solution if the only change is the number of iterations.
- The ability to map surface fluid force using a conservative mapping algorithm with an automatic gap tolerance to a structural simulation modeled with shell elements.
- The ability to define either a spatially varying pressure or force per unit area using a position dependent expression.
- The ability to define a cylindrical reference frame for displacement and support conditions.
- Enhanced prescribed displacements that include the display of a vector arrow.
- Enhanced bolt pretension that enables a single bolt pretension to represent a collection of bolts, and the ability to post-process the bolt adjustment and the bolt working load.
- Enhanced group view for the results task that allows the display of multiple results in the same graphics scene.
- Enhanced calculated values that include the display of an annotation for the calculated value in the graphics scene.
- The ability to specify the AIM user interface color theme - light (default), white or dark - based on user preference.
Enhancements in AIM 18.0

- Enhanced multiphysics workflow that automatically copies material assignments from the previous physics task when connected to the same geometry task.

- Enhanced monitoring of calculated values that includes a user interface shortcut to add calculated values to a monitor chart.

- Enhanced simulation steps that includes access to the simulation step manager from boundary conditions, and the ability to visualize applied factors.

- Enhanced definition of results that includes the ability to automatically use variable names when defining new results.

- The ability to display the AIM user interface in the Chinese language.

- The ability to execute geometry modeling operations via a Python script.

- The ability to simulate conjugate heat transfer for polymer extrusion including fluid-solid, fluid-fluid and solid-solid region interfaces.

- Enhanced material models for polymer extrusion that include a simplified viscoelastic model to account for extrudate swelling.
Chapter 4: Enhancements in AIM 17.2

The following enhancements were made to ANSYS AIM for Release 17.2.

• The ability to perform static structural stress simulation in multiple steps. Structural loads and physics coupling interfaces can be activated or deactivated in each step.

• The ability to apply bolt pre-tension loads in structural simulation including an option to lock the pre-tension and apply loads (additional bolt tensioning or operational) in subsequent simulation steps.

• The ability to perform time-dependent (transient) thermal simulations. Define initial temperatures, apply time varying thermal conditions through tabular data or expressions, post process results over time.

• The ability to define one or more momentum sources/sinks as fluid physics conditions.

• The ability to define heat sources/sinks fluid and fluid-thermal simulations.

• The ability to define rotational or translational periodic region interfaces.

• Enhanced wall boundary condition to allow specification of wall roughness height.

• Enhanced fluid viscosity definition to allow viscosity to be defined as a function of strain rate for fluid flow simulation.

• The ability to monitor calculated values during the solution of a fluid flow simulation.

• The ability to include thermal effects in polymer extrusion simulation.

• Enhanced, generalized Newtonian, viscosity models for polymer extrusion.

• The ability to include generalized slipping models for polymer extrusion.

• Enhanced performance for 1-way physics coupling through the use of 2-core parallel processing for all surface and volumetric data mapping.

• The ability to transfer geometry, mesh and selection sets to a Mechanical system from the AIM study.

• The ability to display the AIM UI in Japanese language.

• The ability to create an HTML report to provide a summary of all simulation data.

• Enhanced animation controls in graphics.

• Enhanced usage of conditionally up-to-date state messaging.

• Enhanced performance for geometry with large number of facets.

• Automatic selection of primary field for editing when working with the immersive data entry panel.
• Enhanced field level help. Some fields now include short video demonstrations within the field level help window. You can double-click on the video to launch a larger version in your default video player.
Chapter 5: Limitations

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.

When selecting cylindrical faces using Location Criteria Functions, some functions with attributes set to x,y or z may not return correctly. **Work-around:** Manually select the cylindrical faces for simpler geometries or use the area attribute if applicable. (147577)