
# Release Notes

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

Release 18.2
August 2017

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 ... x
6.1. Workbench ... x
6.1.1. New Verification Test Cases ... x
7. Online Video Access ... x
8. ANSYS Customer Portal ... x
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 ... 5
1.4. Geometry Enhancements ... 5
1.5. Material Enhancements ... 6
1.6. External Model Enhancements ... 6
1.7. Model Transfer Enhancements ... 7
1.8. Contact and Connection Enhancements ... 7
1.9. Mesh Enhancements ... 8
1.10. Analysis Enhancements ... 8
1.11. Linear Dynamics Enhancements ... 8
1.12. Topology Optimization Enhancements ... 8
1.13. Loads/Supports/Conditions Enhancements ... 9
1.14. Mapping Enhancements ... 9
1.15. Solution Enhancements ... 9
1.16. Rigid Body Solver Enhancements ... 10
1.17. Explicit Dynamics Enhancements ... 10
1.18. Results Enhancements ... 10
**2. Mechanical APDL** ... 13
2.1. Structural ... 13
2.1.1. Contact ... 13
2.1.1.1. Small-Sliding Logic ... 13
2.1.1.2. Linear Contact ... 14
2.1.2. Material and Fracture Modeling ... 14
2.1.2.1. Drucker-Prager Concrete ... 14
2.1.2.2. Viscoelasticity ... 14
2.1.2.3. Microplane Damage ... 14
2.1.2.4. Nominal Strain for Hyperelastic Material ... 15
2.2. Multiphysics ... 15
2.2.1. Acoustics ... 15
2.2.2. Diffusion and Coupled-Field ... 15
2.3. Solvers ... 15
2.3.1. Sparse Solver Enhancements ... 16
2.3.2. Distributed ANSYS Enhancements ... 16
2.3.3. GPU Acceleration Enhancements ... 16

2.4. Commands ... 16
2.4.1. New Commands ... 16
2.4.2. Modified Commands ... 16
2.4.3. Undocumented Commands ... 17
2.5. Elements ... 18
2.5.1. Modified Elements ... 18
2.6. Other ... 18
2.6.1. Mechanical APDL Product Launcher ... 18
2.7. Documentation ... 18
2.7.1. Documentation Updates for Programmers ... 18
2.7.2. *Feature Archive* ... 19
2.8. Known Incompatibilities ... 19
2.8.1. Real Constant TNOP for Contact Elements ... 19
**3. Autodyn** ... 21
3.1. New Features and Enhancements ... 21
**4. Aqwa** ... 23
4.1. Aqwa Solver Modules ... 23
4.1.1. Aqwa-Wave with Forward Speed ... 23
4.1.2. New Items in AH1 File ... 23
4.2. Hydrodynamic Analysis Systems ... 23
4.2.1. Current Calculation Position ... 23
4.2.2. Current Object ... 23
4.2.3. Slender Tube ... 23
4.2.4. Nonlinear Roll Damping ... 23
4.2.5. Yaw-Rate Drag ... 24
4.2.6. Frequency Domain Statistics ... 24
**5. ANSYS Composite PrepPost (ACP)** ... 25
5.1. New Features in ANSYS Composite PrepPost (ACP) 18.2 ... 25
5.1.1. New Failure Criteria Formulations ... 25
5.1.2. Collapse Tree Button ... 25
5.1.3. Distance Measure Tool ... 25
5.1.4. Suppress Materials in Postprocessing ... 25
5.2. Supported Platforms for ANSYS Composite PrepPost (ACP) 18.2 ... 25
5.3. Known Limitations and Incompatibilities ... 26
**II. ANSYS Fluids Products** ... 27
**1. Fluent** ... 29
1.1. New Features in ANSYS Fluent 18.2 ... 29
1.2. Supported Platforms for ANSYS Fluent 18.2 ... 33
1.3. New Limitations in ANSYS Fluent 18.2 ... 33
1.4. Resolved Issues and Limitations ... 34
1.5. Updates Affecting Code Behavior ... 36
**2. CFX** ... 39
2.1. Supported Platforms ... 39
2.2. New Features and Enhancements ... 39
2.3. Incompatibilities ... 39
2.4. Updates Affecting Code Behavior ... 39
**3. TurboGrid** ... 41
3.1. Supported Platforms ... 41
3.2. New Features and Enhancements ... 41
**4. BladeModeler** ... 43
4.1. Supported Platforms ... 43
**5. CFD-Post** ... 45

5.1. Supported Platforms ... 45
5.2. New Features and Enhancements ... 45
**6. Polyflow** ... 47
6.1. New Features ... 47
6.2. Supported Platforms ... 47
6.3. New Limitations in ANSYS Polyflow 18.2 ... 47
6.4. Past Versions of ANSYS Polyflow Release Notes ... 47
**7. Forte** ... 49
7.1. New Features and Enhancements ... 49
7.2. Resolved Issues since Forte 18.1 ... 50
7.3. Supported Platforms ... 50
**8. Chemkin-Pro** ... 51
8.1. New Features and Enhancements ... 51
8.2. Resolved Issues since 18.1 ... 51
8.3. Supported Platforms ... 52
**9. FENSAP-ICE** ... 53
9.1. New Features and Enhancements in ANSYS FENSAP-ICE ... 53
9.2. Beta Features ... 55
**III. ANSYS Electronics Products** ... 57
**1. Icepak** ... 59
1.1. Introduction ... 59
1.2. New and Modified Features in ANSYS Icepak 18.2 ... 59
1.3. Resolved Issues and Limitations in ANSYS Icepak 18.2 ... 59
**IV. ANSYS Geometry & Mesh Prep Products** ... 61
**1. DesignModeler** ... 63
**2. SpaceClaim** ... 65
**3. CAD** ... 67
**4. Meshing** ... 69
4.1. Meshing Application Advisories ... 69
4.2. Resuming Databases from Previous Releases ... 69
4.3. Sizing Enhancements and Changes ... 70
4.4. Contact Enhancements ... 71
4.5. Selective Meshing Enhancements ... 71
4.6. Documentation ... 71
**5. IC Engine Release Notes** ... 73
**6. ICEM CFD** ... 75
6.1. Multizone Block Editing improvements ... 75
6.2. Usability Improvements ... 75
**7. Fluent Meshing** ... 77
7.1. Changes in Product Behavior from Previous Releases ... 77
7.2. New Features ... 77
7.3. Known Issues and Limitations ... 78
**V. ANSYS Simulation Products** ... 81
**1. Workbench** ... 83
1.1. ANSYS Workbench ... 83
1.1.1. Design Point Update Enhancements ... 83
1.1.2. Units ... 84
1.1.3. Project Toolbox ... 84
1.1.4. Mechanical APDL Enhancements ... 84
1.1.5. ANSYS Workbench-Remote Solve Manager Enhancements ... 84
1.1.6. ANSYS Workbench-EKM Enhancements ... 84
1.2. Fluid Flow (CFX) ... 84

1.3. External Connection Add-In ... 85
1.4. Engineering Data Workspace ... 85
1.5. External Data ... 85
1.6. External Model ... 85
1.7. Enhancement to Mechanical Model Cells ... 85
1.8. FE Modeler ... 85
1.9. System Coupling ... 86
1.10. TurboSystem Release Notes ... 86
1.10.1. Supported Platforms ... 86
**2. ACT** ... 87
**3. Remote Solve Manager (RSM)** ... 93
3.1. New Features and Enhancements ... 93
3.2. Issues Resolved in this Release ... 93
3.3. Known Issues and Limitations ... 93
**4. EKM** ... 95
4.1. New Features and Enhancements ... 95
4.2. Issues Resolved in this Release ... 95
4.3. Issues and Limitations ... 96
**5. DesignXplorer** ... 97
**6. ANSYS Viewer** ... 99
6.1. New Features and Enhancements ... 99
6.2. Known Issues and Limitations ... 99
**VI. ANSYS AIM** ... 101
**1. Advisories** ... 103
**2. Enhancements in AIM 18.2** ... 105
**3. Enhancements in AIM 18.1** ... 107
**4. Enhancements in AIM 18.0** ... 109
**5. Limitations** ... 111

# Global Release Notes

The release notes are specific to ANSYS, Inc. Release 18.2 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. x)
- 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. x) for the following:

- ANSYS 18.1
- ANSYS 18.0
- ANSYS 17.2
- ANSYS 17.1

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

The Release Documentation files include the following:

- ANSYS Platform Support Strategy & Plans
- ANSYS, Inc. Installation and Licensing Tutorials
- ANSYS, Inc. Known Issues and Limitations
- ANSYS, Inc. Licensing Guide
- ANSYS Quick Start Installation Guide
- ANSYS Quick Start Licensing Guide

- ANSYS, Inc. Release Notes
- Linux Installation Guide
- SpaceClaim Release Notes (as applicable)
- Windows Installation Guide

# 1. Advisories

In addition to the incompatibilities noted within the release notes, known non-operational behavior, errors and/or limitations at the time of release are documented in the **Known Issues and Limitations** document, although not accessible via the ANSYS Help Viewer. See the ANSYS Customer Portal (p. x) 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.

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, with ANSYS 18.2, the ANSYS Viewer has replaced

the former ANSYS CFD-Viewer which remains functional until further notice.

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 and CFD-Post reports (HTML) that reference CVF files that have been generated in previous versions can be converted by using applications supplied as part of the CFX, ANSYS TurboGrid and CFD-Post installations. For example:

```
C:\Program Files\ANSYS Inc\v182\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".

And

```
C:\Program Files\ANSYS Inc\v182\CFX\bin\cfx5htmlconvert <html-file>
```

will generate an ANSYS Viewer file (ARZ file) which can be loaded into the ANSYS Viewer. The new file will be named based on the name of the HTML file but with the file extension replaced with "arz". All 2D and 3D image files previously included in the directory associated with the report will be included in the ARZ file.

## Academic Product Notification

If you are running one of the "Academic Mechanical and CFD" licenses (Associate, Research, or Teaching) with a solver license that supports license sharing (such as CFX), then you will be able to run an FSI simulation with a single license.

# 2. Compatibility with Previous Releases

**Backwards Compatibility**: ANSYS 18.2 was tested to read and resume databases from the following previous versions: 17.0, 17.1, 17.2 and 18.0, 18.1. 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 18.2, first resume it in 17.x and then resume that database in 18.2.

**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 18.2. Review these items carefully.

- A number of general enhancements were made to the installation interface to provide support for high definition (4K) monitors.

# 4. Licensing

The following enhancements were made to ANSYS, Inc. Licensing for Release 18.2:

- The current version of OpenSSL utilized with the ANSYS Licensing Interconnect is 1.1.0f.
- Enhanced License Manager security by using a SHA256 certificate and increasing the encryption strength to 4096 bit.

# 5. Documentation

## Online Documentation

A beta release of the online version of our product documentation is available. 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.

## Documentation Installation

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. Workbench

## 6.1. Workbench

The following sections outline the changes to the Workbench Verification Manual:

6.1.1. New Verification Test Cases

### *6.1.1. New Verification Test Cases*

The following new VMs are available:

- VMMECH100 - 3-D Acoustic Modal Analysis with Temperature Change
- VMMECH101 - Natural Frequency of a Submerged Ring

# 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 18.2. 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.

| Product Name | Capabilities |
|---|---|
| ANSYS CFD Enterprise | This level includes everything in ANSYS CFD Premium and:<br>• ANSYS Polyflow<br>• ANSYS Forte<br>• ANSYS FENSAP-ICE<br>• ANSYS AIM Pro<br>• ANSYS Simplorer Entry<br>• ANSYS DesignXplorer |
| ANSYS CFD Enterprise Solver | Includes everything in ANSYS CFD Premium Solver and:<br>• ANSYS Polyflow Solver<br>• ANSYS Forte Solver<br>• ANSYS FENSAP-ICE Solver<br>• Can be used in combination with ANSYS CFD PrepPost to run AIM Pro |

| Product Name | Capabilities |
|---|---|
| ANSYS CFD Premium | This level includes:<br>• ANSYS Fluent<br>• ANSYS CFX<br>• ANSYS Meshing (including ANSYS TurboGrid and ANSYS ICEM CFD)<br>• ANSYS OptiGrid<br>• ANSYS SpaceClaim Direct Modeler<br>• ANSYS CFD-Post<br>• 4 HPC |
| ANSYS CFD Premium Solver | This level includes:<br>• ANSYS Fluent Solver<br>• ANSYS CFX-Solver<br>• 4 HPC |
| ANSYS CFD PrepPost | Prior to Release 18.0, ANSYS CFD PrepPost included:<br>• ANSYS Meshing (including ANSYS TurboGrid and ANSYS ICEM CFD)<br>• ANSYS Fluent PrepPost<br>• ANSYS CFX-Pre<br>• ANSYS CFD-Post<br>• ANSYS Polyflow PrepPost<br>Release 18.0 adds the following:<br>• ANSYS SpaceClaim Direct Modeler<br>• ANSYS OptiGrid<br>• ANSYS FENSAP-ICE PrepPost<br>• Can be used in combination with ANSYS CFD Enterprise solver to run AIM Pro<br>• ANSYS Forte PrepPost |
| ANSYS FENSAP-ICE | Replaces ANSYS FENSAP-ICE 16/32/64/128/256/unlimited and includes: |

| Product Name | Capabilities |
|---|---|
| | • ANSYS FENSAP-ICE Solver<br>• ANSYS FENSAP-ICE PrepPost<br>• ANSYS FENSAP-ICE Turbo<br>• ANSYS OptiGrid<br>ANSYS FENSAP-ICE provides access to one solver core. |

**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, 18.1, and 18.2.

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. 13)
Autodyn (p. 21)
Aqwa (p. 23)
ACP (p. 25)

# 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. Material Enhancements
1.6. External Model Enhancements
1.7. Model Transfer Enhancements
1.8. Contact and Connection Enhancements
1.9. Mesh Enhancements
1.10. Analysis Enhancements
1.11. Linear Dynamics Enhancements
1.12. Topology Optimization Enhancements
1.13. Loads/Supports/Conditions Enhancements
1.14. Mapping Enhancements
1.15. Solution Enhancements
1.16. Rigid Body Solver Enhancements
1.17. Explicit Dynamics Enhancements
1.18. 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 18.2 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 18.2 includes several new features and enhancements that result in product behaviors that differ from previous releases. These behavior changes are presented below.

- **Modal Analysis Frequency Values**. In previous releases of Mechanical, for a damped modal system, the application reported negative modes in the **Tabular Data** window (via the **Solution** object) if the solution consisted of any rigid body modes. The application now reports only rigid body modes and the positive frequencies in **Tabular Data**.

- **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 has been 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 have been 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).

- **Application Generated Messages**. Now, when Mechanical issues a warning/error/information message, the message is displayed in a pop-up window for five seconds at the bottom of the interface before being automatically hidden. The Messages window is no longer automatically displayed when new messages are issued. All messages are available to view in the **Messages Window**.

- **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 18.2:

- **Environment Duplication**. Mechanical now enables you to duplicate an entire analysis environment, including all of its child-objects, simply by selecting the environment object, right-clicking, and selecting the new **Duplicate** option.

- **Adding an Analysis System in Mechanical**. From within the application, Mechanical now enables you to add an analysis to your model from the new **Standard Toolbar** drop-down menu, **New Analysis**. A corresponding analysis system, with the appropriate connections, is also included in the Project Schematic.

- **Element Face Selections**:

  - **Graphical Selections**. You can now graphically select element faces on your mesh using the new **Element Face** option on the Graphics toolbar (or hotkey combination **Ctrl**+**K**). With Element Face selections, you can view element information in **Selection Information** window, create Named Selections, and scope results, including User Defined Results, to element faces and element face-based Named Selections.

  - **Named Selections using Worksheet Criteria**. A new **Worksheet** criteria option, **Element Face**, is now available. This option enables you to convert geometry face selections or node selections to element face selections.

  - **Sending Element Face Named Selections to Solver**. You can now send element face-based Named Selections to the solver as either a nodal component or a set of Mesh200 elements.

- **Named Selections Icons**. The tree object icon for Named Selections now illustrates the corresponding geometric entity or mesh type.

- **Coordinate System APDL Name**. The new **Coordinate System** object property, **APDL Name**, enables you to create an APDL parameter (in the input file) and assign its value to the reference number of the coordinate system. This facilitates easy programmatic identification of the coordinate system for later use/reference in a Command object.

- **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 18.2:

- **Full Screen Mode**. Using the F11 hotkey, Mechanical now enables you to quickly maximize the **Geometry** window. This can be for presentation purposes as well as when you are preparing an analysis.

- **Hotkey Additions**. When the Geometry window has focus, the **H** and **Z** keys now enable you to zoom in on a selected geometric entity (**Z**) and then return the view to the default Isometric View (**H**).

- **Model Rotation**. Model rotation has been improved. When selecting the model in the **Geometry** window, the middle mouse button now defines the rotational center upon selection, enabling you to immediately rotate the model around that point.

- **Graphics Annotations**. A new **Notes** field has been added for User Defined annotations. This replaces the ability to add notes in the **Value** column in previous releases. You can display your **Notes** as multiple lines in the **Geometry** window using the backspace (\) keyboard character in your **Note**. All text following the backspace is considered a new line.

- **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 **Geometry** window.

- **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**.

## 1.4. Geometry Enhancements

The following geometry enhancements were made at Release 18.2:

- **Surface Coating**. Mechanical now enables you to create a shell surface, or Surface Coating, on an existing 3D face of your model. This feature enables you to accurately evaluate surface stresses or to overlay your structure with thin parts, for example, to model Thermal Barrier Coatings. To accurately model this type of application, Mechanical enables you to specify the thickness, stiffness behavior, coordinate system, and material.

- **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 Section** 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. Material Enhancements

Refer to the 18.2 Release Notes of the Engineering Data Workspace application for new features and enhancements associated with materials in the Mechanical application.

## 1.6. External Model Enhancements

The following enhancements for External Model were made at Release 18.2.

- **Process Mesh200 Elements**. For .cdb files only, External Model supports importing Mesh200 elements that are included in your mesh file.

- **Importing Contacts**. You can now transfer contact surface to surface (solid or shell bodies) data/objects through the **External Model** system. For thermal analyses, this includes thermal conductivity. This new data type utilizes the **Worksheet** in order to better manage large amounts of finite element data.

- **Sorting Imported Worksheet Data**. For imported data from an **External Model** source, you can now sort **Worksheet** table data by clicking on a column heading.

- **Importing ABAQUS Finite Element Data**. ANSYS Workbench and ANSYS 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 Mechanical (.cdb) and Abaqus Input (.inp) 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**

- **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, contact, and remote connection data (flexible or rigid) into thermal analyses.

## 1.7. Model Transfer Enhancements

The following contact and model transfer enhancements were made at Release 18.2:

- **Transferring System Data from AIM**. You can now transfer your geometry, mesh, Material Assignments, and Contact conditions from AIM to Mechanical.

## 1.8. Contact and Connection Enhancements

The following contact and connection enhancements were made at Release 18.2:

- **Beam Connection Length**. The **Beam Connection** feature has a new property: **Beam Length**. This read-only property displays the length of the beam based on the end points from the scoping of the **Reference** and **Mobile** categories.

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

- **Small Sliding**. A new contact property is now available that enables you to specify whether a contact interface will have a small sliding motion.

- **Contact Scoping**. You can now scope Contact Regions to element faces.

- **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.

## 1.9. Mesh Enhancements

Refer to the 18.2 Release Notes of the Meshing application for new features and enhancements associated with Meshing in the Mechanical application.

## 1.10. Analysis Enhancements

The following analysis enhancements have been made at Release 18.2:

- **Acoustics Analyses**. ANSYS Workbench now provides Harmonic Acoustics and a Modal Acoustics analysis types. Modal Acoustics will enable you to model the acoustic and optionally structural domain together to determine frequencies and standing wave patterns. Harmonic Acoustics analyses are used to determine the steady-state response of an acoustic region or a structure and the surrounding fluid medium to loads and excitation that vary sinusoidally with time.

## 1.11. Linear Dynamics Enhancements

The following enhancements were made at Release 18.2 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.

- **Modal Analysis Frequency Values**. In previous releases of Mechanical, for a damped modal system, the application reported negative modes in the **Tabular Data** window (via the **Solution** object) if the solution consisted of any rigid body modes. The application now reports only rigid body modes and the positive frequencies in **Tabular Data**.

## 1.12. Topology Optimization Enhancements

The following Topology Optimization analysis enhancements were made at Release 18.2:

- 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
- The Topology Optimization Analysis now:

- Supports optimization of objectives and constraints selected from multiple Static Structural or Modal analysis types.
- Now supports a new option (**All Supports**) for the **Exclusion Region** category as defined by the **Boundary Condition** property for the Optimization Region object.
- Supports combined objectives through the **Objective** object Worksheet.
- Supports Response Constraints applied to selected static structural or modal analyses.
- Supports the addition of exclusions to the **Optimization Region** object using the new **Exclusion Region** object.
- Enables you to specify a range for the **Mass Constraint** and **Volume Constraint** response types. The new **Define By** property enables you to specify the minimum and maximum percent to retain.
- Supports linear springs using **Compliance** Response Type for the Objective.
- Enables you to use the **Stop** button on the Solution Status window to stop your Topology Optimization solution. See the Understanding Solving section of the User's Guide.

• **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.

## 1.13. Loads/Supports/Conditions Enhancements

The following loads/supports/conditions enhancements were made at Release 18.2:

• **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.

## 1.14. Mapping Enhancements

The following mapping enhancements were made at Release 18.2:

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

## 1.15. Solution Enhancements

The following solution enhancements were made at Release 18.2:

• **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.

- **Tabular Data Load Step Display via the Solution Object**. For Static Structural and Transient Structural analyses using the Mechanical APDL solver, when you select the **Solution** object once the solution is complete for a multi-step analysis, the **Tabular Data** window displays the **Time** associated and now also includes each **Step** of the analysis as well as each **Substep** as available.

## 1.16. Rigid Body Solver Enhancements

The following Rigid Body Solver enhancements were made at Release 18.2:

- 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.

## 1.17. 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 have been made at Release 18.2:

- A new Mechanical ACT Extension, MechanicalDropTest, has been included in the installation. The extension includes a wizard which automates the setup of a drop test analysis in an Explicit Dynamics system. To use the Drop Test Wizard, you only need to provide the geometry of the object to be dropped.

The following LS-DYNA Solver enhancements have been made at Release 18.2:

- The Workbench LS-DYNA documentation is now fully incorporated into the online help.

## 1.18. Results Enhancements

The following results enhancements were made at Release 18.2:

- **Multiple Result Sets**. The **By** property, used to review result contours from the multiple result sets, has two new options: **Minimum Over Time** and **Time of Minimum**. These options enable you to display the minimum result value for a result set.
- **Automatic Result Creation**. Mechanical now provides two new contextual (right-click) menu options to automatically create new results from solution generated result data:
  - **Create Results**. This option is available in the **Tabular Data** window. It enables you to create result objects from solution-based **Tabular Data** content.
  - **Create Results at All Sets**. This option is available on solved result objects. It enables you to automatically create a group of results based on all of the available **Result Sets** for a given solved result item/object.
- **Result File Item (formerly Solver Component) Result Scoping Option**. A new result **Scoping Method** is available: **Result File Item**. In the previous release of Mechanical, this option was named **Solver Component**. This result scoping capability has been updated and renamed. This option still enables you to scope results on solution generated Material IDs, Element Name IDs, and Element Type IDs, but now you can also scope results to solver components. In addition, graphical interface options now enable you to highlight elements in the **Geometry** window, collapse consecutive Material/Element IDs, and there are previous/next options for large **Worksheet** tables of data.

- **Surface Coating**. **Surface Coating** is a new result **Scoping Method** that enables you to view results on Surface Coating objects.

- **Exporting ANSYS Viewer Files**. When exporting result objects to the ANSYS Viewer, the application now automatically opens the viewer after you have saved your AVZ file. This default behavior can be changed the under the Export preference of the **Options** dialog.

- **Fatigue Tool**. For Harmonic Response analyses, the **Fatigue Tool** has two new properties: **Frequency Selection** and **Sweep Rate**. The **Frequency Selection** property enables you to specify whether a single frequency, multiple frequencies, or a Sine Sweep analysis is requested. If you select Sine Sweep, you then need to specify a **Sweep Rate**, the rate of frequency sweep.

- **Animation**. For the Animation feature:

  – The **Graph** window has a new option for multi-step analyses: **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.

  – You can now reverse or advance an animation frame-by-frame using the new **Previous Frame** and **Next Frame** buttons. In addition, the **Stop** and **Play** (or **Pause**) options now have Hotkeys associated with them: the **S** and **P** character keys, respectively.

# Chapter 2: Mechanical APDL Release Notes

Release 18.2 of the Mechanical APDL application offers most of the capabilities from prior releases plus many new features and enhancements. Areas where you will find changes and new capabilities include the following:


- Structural (p. 13)
- Multiphysics (p. 15)
- Solvers (p. 15)
- Commands (p. 16)
- Elements (p. 18)
- Other (p. 18)
- Documentation (p. 18)


See also Known Incompatibilities (p. 19) for more information about this release.

**Backward Compatibility:** Mechanical APDL Release 18.2 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 18.2 includes the new features and enhancements for the following structural analysis disciplines:


2.1.1. Contact
2.1.2. Material and Fracture Modeling


### 2.1.1. Contact

Release 18.2 includes the following enhancements for structural analyses involving contact:


2.1.1.1. Small-Sliding Logic
2.1.1.2. Linear Contact


#### *2.1.1.1. Small-Sliding Logic*

Prior to this release, the program tracked only finite-sliding contact, reforming nodal connectivity of the contact element at each equilibrium iteration. In this release, a new small-sliding logic is available for modeling relatively small-sliding motions between contact and target surfaces. The nodal connectivity of the contact element is formed only once in the beginning of the analysis and remains unchanged throughout.

The small-sliding logic reduces solution cost and improves solution robustness. It can easily solve certain complex contact models for which the finite-sliding logic might have difficulty, especially models with a low-quality geometry or mesh and non-smooth contact interfaces. Furthermore, the sparse solver can reuse the same matrix structure throughout the simulation, avoiding the costly sequential step of equation-ordering at every equilibrium iteration and leading to significant performance improvements and better scalability in a distributed-memory parallel run.

For more information, see Selecting a Sliding Behavior in the *Mechanical APDL Contact Technology Guide*.

#### 2.1.1.2. Linear Contact

The use of kinematic multipoint constraints (MPCs) to represent linear contact behavior has been available for several releases. In this release, a general linear contact capability for all contact algorithms (penalty, MPC, and Lagrange multiplier) is now available. If no other nonlinearities exist (plasticity, large deformation, or unilateral contact), a nonlinear solution is generally not required to obtain an accurate solution. In this case, the contact searching and the factorized stiffness matrix are performed only once in the beginning of the linear analysis, significantly improving solution performance. For more information, see Specifying Linear Contact in the *Mechanical APDL Contact Technology Guide*.

### 2.1.2. Material and Fracture Modeling

Release 18.2 includes the following enhancements to material modeling and fracture analysis technology used in structural analyses:

2.1.2.1. Drucker-Prager Concrete
2.1.2.2. Viscoelasticity
2.1.2.3. Microplane Damage
2.1.2.4. Nominal Strain for Hyperelastic Material

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

#### 2.1.2.1. Drucker-Prager Concrete

To represent tension behavior in concrete models, you can now use a composite surface consisting of a Rankine tension failure surface and a Drucker-Prager (DP) surface in compression. The Rankine surface is useful for representing the brittle tensile behavior of concrete. The composite DP-Rankine surface supports the linear, exponential, and fracture energy HSD models. For more information, see Drucker-Prager Concrete in the *Mechanical APDL Material Reference*.

#### 2.1.2.2. Viscoelasticity

For viscoelastic materials, you can now specify the maximum equivalent viscous strain increment within a time step, affording you more control over the balance between higher accuracy (smaller increments) and faster simulation runtimes (larger increments). For more information, see Dissipation in the *Mechanical APDL Material Reference* and **CUTCONTROL**.

#### 2.1.2.3. Microplane Damage

Relevant postprocessing commands now support macroscopic and maximum microplane damage (general item MPLA). For more information, see **ANSOL**, **ESOL**, **ETABLE**, **PLESOL**, **PLNSOL**, **PRESOL**, and **PRNSOL**.

#### 2.1.2.4. Nominal Strain for Hyperelastic Material

Relevant postprocessing commands now support nominal strain (general item NS) for hyperelastic material. For more information, see **ANSOL**, **ESOL**, **ETABLE**, **PLESOL**, **PLNSOL**, **PRESOL**, and **PRNSOL**.

## 2.2. Multiphysics

Release 18.2 includes the following enhancements for analyses involving multiphysics environments:

2.2.1. Acoustics
2.2.2. Diffusion and Coupled-Field

### 2.2.1. Acoustics

The following acoustic analysis enhancements offer greater flexibility for postprocessing acoustic results:

- You can now postprocess acoustic quantities over multiple load steps and substeps, or over a selected frequency range (**PRAS**, **PLAS**, **PRFAR**, **PLFAR**).
- The sound pressure level and A-weighted sound pressure level over octave bands are now computed at far-field points (**PRFAR**, **PLFAR**).
- The equivalent radiated power from structural surfaces is now available for structure-borne sound (**PRAS**, **PLAS**).
- A waterfall diagram is now available for equivalent radiated power, radiated sound power level, and far-field parameters (**PLAS**, **PLFAR**).
- You can now list and visualize normal velocity (VNS) on the structural surface (**PRNSOL**, **PLNSOL**).

### 2.2.2. Diffusion and Coupled-Field

The following enhancements are available for defining loads and material properties in a diffusion or coupled-field analysis:

- TABLE type array parameters now support the concentration degree of freedom (CONC) as a primary variable. For more information, see the ***DIM** command.
- The pure-diffusion and coupled-field elements (PLANE223, SOLID226, SOLID227, PLANE238, SOLID239, and SOLID240) now support material properties defined as a function of primary variables via TABLE type array parameters. For more information, see Defining Materials Using TABLE Type Array Parameters in the *Mechanical APDL Basic Analysis Guide*.
- In addition to other primary variables, the diffusion-flux (DFLUX) and diffusion-substance-generation (DGEN) loads can now be defined as functions of concentration (CONC) via TABLE type array parameters. For more information, see Applying Loads Using TABLE Type Array Parameters in the *Mechanical APDL Basic Analysis Guide*.

These enhancements provide greater flexibility for specifying complex loading and material definitions, for example in analyses involving electromigration, thermomigration, or stress migration.

## 2.3. Solvers

Release 18.2 includes the following improvement to the solution process:

2.3.1. Sparse Solver Enhancements

2.3.2. Distributed ANSYS Enhancements
2.3.3. GPU Acceleration Enhancements

### 2.3.1. Sparse Solver Enhancements

The performance of the sparse solver (**EQSLV**,SPARSE) is now significantly improved when running on the newest generation of Intel Xeon processors (code-named Skylake). The sparse solver can now make use of the AVX-512 instruction set, increasing the overall performance of the solver on processors that support the instructions.

The performance of the sparse solver has been improved for substructuring analysis.

The scalability of the sparse solver has been improved when used in a distributed-memory parallel solution (Distributed ANSYS), particularly for simulations that use more than 256 cores.

### 2.3.2. Distributed ANSYS Enhancements

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

- Added support for residual-vector and residual-response calculations (**RESVEC**), including the use of residual vectors (or responses) for modal analysis, and residual vectors for spectrum analysis and harmonic and transient analyses using the mode-superposition method.
- Improved performance of the domain-decomposition step when hundreds of thousands of contact pairs exist in the model.

### 2.3.3. GPU Acceleration Enhancements

The GPU accelerator capability now supports substructuring analysis. For CMS substructures, support includes the ability to accelerate the requisite eigensolver calculations involved in generating the superelement.

## 2.4. Commands

This section describes changes to commands at Release 18.2:

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:

- **MRPM** – Defines the revolutions per minute (RPM) for a machine rotation.
- **PLAS** – Plots a specified acoustic quantity during postprocessing of an acoustic analysis.

### 2.4.2. Modified Commands

The following commands have been enhanced or otherwise modified:

- **ANSOL** – Specifies averaged nodal data to be stored from the results file in the solution coordinate system. This postprocessing command now supports nominal strain for hyperelastic material (NS) and microplane damage (MPLA).
- **CUTCONTROL** – Controls time-step cutback during a nonlinear solution. A new VSLIMIT argument sets the maximum equivalent viscous strain increment allowed within a time step for viscoelastic materials.
- ***DIM** – Defines an array parameter and its dimensions. The concentration degree-of-freedom (CONC) has been added as a primary variable.
- **ESOL** – Specifies element data to be stored from the results file. This postprocessing command now supports nominal strain for hyperelastic material (NS) and microplane damage (MPLA).
- **ETABLE** – Fills a table of element values for further processing. This postprocessing command now supports nominal strain for hyperelastic material (NS) and microplane damage (MPLA).
- **/FORMAT** – Specifies format controls for tables. This postprocessing command now enables you to specify whether the exponent contains two or three digits. (Default = 3.)
- **KEYOPT** – Sets element key options. New *ITYPE* labels, CONT and TARG, have been added to set element key options for all contact and all target elements, respectively.
- **PLESOL** – Displays solution results as discontinuous element contours. This postprocessing command now supports nominal strain for hyperelastic material (NS) and microplane damage (MPLA).
- **PLFAR** – Plots pressure far fields and far-field parameters. The command now supports more options for plotting the far-field parameters.
- **PLNSOL** – Displays solution results as continuous contours. This postprocessing command now supports nominal strain for hyperelastic material (NS) and microplane damage (MPLA). It also supports normal velocity on the structural surface (VNS) for acoustic analysis.
- **PRAS** – Prints a specified acoustic quantity during postprocessing of an acoustic analysis. The command now supports additional acoustic output quantities.
- **PRESOL** – Prints the solution results for elements. This postprocessing command now supports nominal strain for hyperelastic material (NS) and microplane damage (MPLA).
- **PRFAR** – Prints acoustic far-field parameters. The command now supports more options for printing the far-field parameters.
- **PRNSOL** – Prints nodal-solution results. This postprocessing command now supports nominal strain for hyperelastic material (NS) and microplane damage (MPLA). It also supports normal velocity on the structural surface (VNS) for acoustic analysis.
- **TB** – Activates a data table for material properties or special element input. A Rankine tension-failure surface has been added as an option on the concrete plasticity model (**TB**,CONCR).

### 2.4.3. Undocumented Commands

The following commands have been undocumented:

| Command | Reason |
|---|---|
| PLST | Functionality replaced by the **PLAS** and **PRAS** commands for postprocessing an acoustic analysis. |
| SPOWER | Functionality replaced by the **PLAS** and **PRAS** commands for postprocessing an acoustic analysis. |

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

## 2.5. Elements

This section describes changes to elements at Release 18.2:

2.5.1. Modified Elements

### 2.5.1. Modified Elements

The following elements have been enhanced:

- CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, CONTA177 – These contact elements have a new KEYOPT(18) to specify the sliding behavior: finite-sliding (default) or small-sliding.
- PLANE223, SOLID226, and SOLID227 – These coupled-field elements have been enhanced to accept TABLE type array parameters for material-property definitions.
- PLANE238, SOLID239, and SOLID240 – These diffusion elements have been enhanced to accept TABLE type array parameters for material-property definitions.

## 2.6. Other

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

### 2.6.1. Mechanical APDL Product Launcher

The High Performance Computing Setup tab of the Mechanical APDL Product Launcher has been improved to reflect current program defaults and preferences. Shared-memory parallel processing with two processors is now selected by default, and the distributed-memory processing and GPU acceleration options have been simplified. A new remote shell option has been added for Linux. For more information, see The High Performance Computing Setup Tab in the *Mechanical APDL Operations Guide*.

## 2.7. Documentation

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

### 2.7.1. Documentation Updates for Programmers

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

### 2.7.2. *Feature Archive*

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

## 2.8. Known Incompatibilities

The following incompatibility with prior releases is known to exist at Release 18.2:

2.8.1. Real Constant TNOP for Contact Elements

### 2.8.1. Real Constant TNOP for Contact Elements

For contact elements CONTA171 through CONTA177, real constant TNOP is the chattering-control parameter, representing the maximum allowable tensile-contact pressure. The default TNOP value is now 1 percent of the force-convergence tolerance divided by the contact area (compared to 10 percent in previous releases). The new default behavior improves solution accuracy, but often requires more iterations to achieve contact-element convergence.

# Chapter 3: Autodyn Release Notes

The ANSYS Autodyn product encompasses of 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. 10)).

3.1. New Features and Enhancements

## 3.1. New Features and Enhancements

Release 18.2 for Autodyn has no new features or enhancements.

# 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 18.2. 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 capabilties in the Aqwa solver modules:

### 4.1.1. Aqwa-Wave with Forward Speed

The core solver has been enhanced so that Aqwa-Wave now works with forward speed.

### 4.1.2. New Items in AH1 File

New items, such as center of buoyancy and added mass at high frequency are output in the `*.AH1` file if the AHD1 option is on.

## 4.2. Hydrodynamic Analysis Systems

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

### 4.2.1. Current Calculation Position

Currents can now be calculated at variable depth that moves depending on structure position. For more information, see Define Parts Behavior.

### 4.2.2. Current Object

Current objects now have several options available for definition: constant velocity, varies with depth (dimensional or non-dimensional), and formulated, including 1/7 power law. For more information, see Current.

### 4.2.3. Slender Tube

Slender tube local axes and defining information are now imported from DesignModeler/SpaceClaim Direct Modeler. Orientation is also correctly imported. For more information, see Line Body.

### 4.2.4. Nonlinear Roll Damping

Nonlinear roll damping is now included in analyses by adding the Nonlinear Roll Damping object and defining information there. For more information, see Nonlinear Roll Damping.

### 4.2.5. Yaw-Rate Drag

The Yaw-Rate Drag object can now be added to define drag based on the yaw rotational velocity. For more information, see Yaw-Rate Drag.

### 4.2.6. Frequency Domain Statistics

The Frequency Domain Statistics table now allows you to present statistics as probable maximum, median maximum, expected extreme, and percentile extreme. For more information, see Frequency Domain Tabular Results.

# Chapter 5: ANSYS Composite PrepPost (ACP)

The following enhancements are available in release 18.2. Refer to the product specific documentation for full details of the new features.


5.1. New Features in ANSYS Composite PrepPost (ACP) 18.2
5.2. Supported Platforms for ANSYS Composite PrepPost (ACP) 18.2
5.3. Known Limitations and Incompatibilities


## 5.1. New Features in ANSYS Composite PrepPost (ACP) 18.2

The following new features were added to ANSYS Composite PrepPost (ACP) for the 18.2 release.


5.1.1. New Failure Criteria Formulations
5.1.2. Collapse Tree Button
5.1.3. Distance Measure Tool
5.1.4. Suppress Materials in Postprocessing


### 5.1.1. New Failure Criteria Formulations

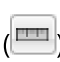
The formulations of the Core Failure and Hashin failure criteria have been updated. The previous implementation of the Core Failure criterion was too conservative, and has been replaced by a new implementation which distinguishes between isotropic and orthotropic core materials.

The Hashin criterion has been brought in line with its published definition. There were some minor differences to the published definition in the previous implementation of this failure criterion in ACP.

### 5.1.2. Collapse Tree Button

The new collapse button in the toolbar of the tree () will collapse folders and objects into their original state.

### 5.1.3. Distance Measure Tool

The new distance measure button () in the toolbar allows you to measure the distance between two points on the mesh. For more information, see Other Features.

### 5.1.4. Suppress Materials in Postprocessing

The Ignore for Postprocessing flag in ACP-Post is now saved with the model. Previously, the setting was only temporary when it was set in ACP-Post. For information on the setting, see Fabric.

## 5.2. Supported Platforms for ANSYS Composite PrepPost (ACP) 18.2

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 18.2.

# Part II: ANSYS Fluids Products

Release notes are available for the following ANSYS Fluids products:


Fluent (p. 29)
CFX (p. 39)
TurboGrid (p. 41)
ANSYS BladeModeler (p. 43)
CFD-Post (p. 45)
Polyflow (p. 47)
Forte (p. 49)
ANSYS Chemkin-Pro (p. 51)
FENSAP-ICE (p. 53)

# Chapter 1: Fluent Release Notes

The following sections contain release information for ANSYS Fluent 18.2.


1.1. New Features in ANSYS Fluent 18.2
1.2. Supported Platforms for ANSYS Fluent 18.2
1.3. New Limitations in ANSYS Fluent 18.2
1.4. Resolved Issues and Limitations
1.5. Updates Affecting Code Behavior


**Backwards Compatibility:** In most instances, version 18.2 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 18.2

New features available in ANSYS Fluent 18.2 are listed below. Where appropriate, references to the relevant section in the User's Guide are provided.

### Solver-Numerics

- If you have selected **Second Order Implicit** or **Bounded Second Order Implicit** for the **Transient Formulation** for a case with a fixed or sliding mesh, you now have the option of using a variable time step size formulation. This formulation reduces errors in the time derivative evaluation that result when you introduce arbitrary time step size variation. For details, see Second-Order Time Integration Using a Variable Time Step Size in the *Fluent Theory Guide*.

### Solver-Meshing

- The polyhedral unstructured mesh adaption (PUMA) method is now available as a full feature. This adaption method can be used to refine all 3D cell types (polyhedra, tetrahedra, hexahedra, and so on), does not create hanging nodes, and consumes less memory during mesh refinement when compared to the hanging node method. For details, see Polyhedral Unstructured Mesh Adaption in the *Fluent Theory Guide* and Mesh Adaption Controls in the *Fluent User's Guide*.

- When automatically creating multiple mesh interfaces from a list of interface zones, it is now possible to specify that the mapped option is applied to all of the resulting interfaces for which at least one side consists of only solid zones. This feature replaces what was formerly known as the **Enforced** option in the **Interface Options** dialog box, which could only be used after the interfaces were created. Setting the mapped option at the time of creation rather than after creation can greatly reduce the processing time for cases that have many interfaces. (Auto Create Options Dialog Box)

- For dynamic meshes, it is now possible to enable solution stabilization for boundary zones of motion type stationary, deforming, or user-defined; previously, this option was only available for boundary zones undergoing rigid body motion with the six degree of freedom (six DOF) solver or system coupling motion. This option may help with convergence for cases with strong fluid-structure interaction. Note that it is not recommended for boundary zones of type interior. (Solution Stabilization for Dynamic Mesh Boundary Zones)

- For overset meshes, the baseline (BSL) and shear-stress transport (SST) k-ω turbulence models are now supported. Note that the latter has always been supported, but this fact was not previously mentioned in the documentation.

**Models**

- Turbulence
  - There are two new fields for postprocessing: **Lambda 2 Criterion** is used to see the resolved turbulent vortices (similar to the Q Criterion), and the time derivative of pressure (**dp-dt**) helps to identify the radiated sound pattern in the compressible flow simulations
- Heat Transfer and Radiation
  - You can now specify an under-relaxation factor for Monte Carlo radiation sources used in the energy equation through the text user interface (TUI).
- Combustion and Species Transport
  - The Dynamic Cell Clustering (DCC) chemistry acceleration method is now extended to steady-state flow simulations.
  - The Dynamic Cell Clustering (DCC) method can now be used in conjunction with the ISAT chemistry acceleration method.
  - For the partially premixed combustion FGM model, you can now solve additional transport equations for selected slow-forming species. (Scalar Transport with FGM Closure)
  - For the partially premixed combustion model, a new method called laminar-flame-speed-library is now available for the **Laminar Flame Speed** material property. The new method enables you to access pre-built laminar flame speed tables for the most common hydrocarbon fuels. (Laminar Flame Speed)
- Discrete Phase Model
  - For droplet and multicomponent particles, a new material property, Thermolysis Model, is available. The model enables the calculation of the mass transfer of the vaporizing species from the droplet to the bulk phase. A typical application where using this material property is important is the design of Selective Catalytic Reduction (SCR) systems. For accurate modeling, it is recommended that you use the single-rate Thermolysis Model for the urea-liquid component in the urea-water particle-mixture. (See Mass Transfer During Law 2—Thermolysis , Multicomponent Particle Definition (Law 7) in the *Fluent Theory Guide*, and Setting Material Properties for the Discrete Phase in the *Fluent User's Guide*.)
  - During the injection setup, the position and orientation for the following injection types will now be displayed in the graphics window:

    → single
    → group
    → all cone injections
    → all atomizer injections

This will help minimize the risk of incorrect injection specification. The injection position and orientation will not be displayed if you use profiles to specify the injection position and direction. If you use the `DEFINE_DPM_INJECTION_INIT` user-defined function, the displayed injection position and orientation will reflect only the default point properties for the specific injection type.

- Volume of Fluid
  - An open-channel flow with more than two phases is now supported for all applicable boundaries under the two-layer assumption. (Determining the Secondary Phase for the Outlet)
- Eulerian Multiphase Model
  - For Mixture multiphase simulations run in double-precision, you can now specify the **Minimum Vol. Frac. for Matrix Solution** limit (in the **Solution Limits** dialog box) for solving phase-specific equations such as Species and UDS.
  - For Eulerian and Mixture multiphase cases, you can now model the following mass transfer effects in the same simulation using multiple mass transfer mechanisms:
    - → cavitation and species mass transfers
    - → cavitation mass transfers with different models and model constants (see Mass Transfer Mechanisms in the *Fluent User's Guide*)
- Electric Potential Model
  - You can now specify the Contact Resistance at one-sided walls and at two-sided internal walls. (Using the Electric Potential Model)
  - You can now fix a value of potential and specify a potential source in a zone. (Using the Electric Potential Model)
- Acoustics
  - You can now plot area-averaged surface pressure level values from the Acoustic Sources FFT dialog box, and write this data to an external file. (Using the FFT of Acoustic Sources)

**Cell Zones and Boundary Conditions**

- The exit corrected mass flow rate specification method is now available as a full feature for mass-flow outlets. This method is intended mainly for rotating machinery applications. It adjusts the mass flow rate to the total conditions at the outlet, and allows you to sweep through the complete machine operational range, including machine operating points from choked flow to stall conditions. (Defining the Mass Flow Rate or Mass Flux)
- When defining a pressure outlet, it is now possible to specify a factor by which the gauge pressure will be multiplied. This is primarily intended to be used when the gauge pressure is defined by a profile file or a user-defined function, as it allows for easy scaling (for example, when creating a performance map of a turbomachine at different mass flow rates). (Defining Static Pressure)
- The fixing of velocity components in a cell zone is now compatible with the pressure-based coupled solver. (Fixing the Values of Variables)

### Parallel Processing

- The accelerated discrete ordinates (DO) solver is now available on Windows and is now supported in serial, although the most benefit will be seen when used in parallel. (Accelerating Discrete Ordinates (DO) Radiation Calculations in the *Fluent User's Guide*)

### Journal Files

- A new text user interface (TUI) command allows you to hide any new TUI prompts that were added for version 18.2 and revert to the 18.1 default arguments: `file/set-tui-version "18.1"`. Using this text command can help a TUI journal created in version 18.1 to work properly in version 18.2; it must be manually added within the 18.1 journal file (for example, as the top line), or invoked in the ANSYS Fluent 18.2 session prior to reading the 18.1 journal file. Note that TUI prompts that were removed for version 18.2 are not addressed by this text command, and must be addressed manually.

  Note that all journals created using version 18.2 will now automatically include this text command, specifying the current version (that is, `file/set-tui-version "18.2"`), to help backwards compatibility with future releases.

- When recording a TUI journal file, commands entered using paths from older versions of Fluent will be upgraded to their current path in the journal file.

- You can read multiple journal files into Fluent at the same time. It is also possible to create and run nested journal files. For additional information on reading multiple journal files and creating nested journal files, see Multiple Journal Files in the *Fluent User's Guide*.

### Adjoint Solver

- All of the ANSYS Fluent discretization schemes for the pressure equation are now available for computing adjoint solutions. This ensures that you can always have the adjoint pressure scheme correspond to that used in the flow calculation; while such correspondence is not necessary, it can provide greater accuracy. (Using the Adjoint Solution Methods Dialog Box)

- When defining a **surface-integral** observable, it is now possible to select isosurfaces and/or clipped surfaces. (General Observables and Editing Observable Definitions)

### Graphics, Postprocessing, and Reporting

- Selecting an animation sequence in the **Playback** dialog box automatically opens and/or moves the appropriate graphics window to the forefront.

- Disabling the **Use Stored View** option in the **Playback** dialog box allows you to manipulate the view for **HSF File** and **In Memory** animations.

- Particle tracks can now be included in scenes, allowing you to combine particle tracks with other graphical postprocessing objects such as contours and vectors. For information on creating scenes, see Displaying a Scene in the *Fluent User's Guide*.

- You can now move and resize the legends of saved particle track definitions.

- A generic force report is now available in addition to the drag, lift, and moment report definitions. See Monitoring and Reporting Solution Data in the *Fluent User's Guide* for information on creating report definitions.

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

# 1.2. Supported Platforms for ANSYS Fluent 18.2

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

# 1.3. New Limitations in ANSYS Fluent 18.2

The following is a list of new or recently discovered limitations known to exist in ANSYS Fluent 18.2. Where possible, suggested workarounds are provided.

- Models
  - For all multiphase models, the `solve/set/expert` option for not freeing temporary solver memory is incompatible with dynamic adaption, not only in parallel, but now in serial as well. (Workaround (p. 36))
  - The **Use DPM Domain** option of the hybrid parallel DPM tracking method cannot be used with the following:

    → the PDF Transport model

    → the Eulerian Wall Film model

    → cases that include a wall / shadow pair
  - If a user is not part of a video group, a session on a local Linux machine with SLES11, SP4, and OpenGL may terminate abnormally during an injection setup for the discrete phase model case. To avoid this issue, it is recommended that the user restarts the ANSYS Fluent session with the driver option `-x11` or that the user is added to the video group on that machine. **Workaround**: Deactivate the injection visualization during the case setup by using the following Scheme command:

    ```
    (rpsetvar 'dpm/visualize-injections? #f)
    ```
- Graphics, Reporting, and Postprocessing
  - When running the parallel version of ANSYS Fluent, the **Curve Length** function for solution XY plotting is only supported when the curvilinear surface is contained within a single partition.
  - Scene animations created using **Key Frames** in the **Animate** dialog box are not compatible with graphics displays on isosurfaces (contours, vectors, pathlines, or particle tracks). Pathlines are not compatible with scene animations, regardless of the selected surface(s).
- File / Data Import and Export
  - If you are accessing a file using a Universal Naming Convention (UNC) path, you must ensure that you have permission to access to all of the folders in the path or you will not be able to open the file.

- It is no longer possible to export volume data for RadTherm software from the serial version of ANSYS Fluent. (Workaround (p. 36))
- To properly view Fieldview Unstructured (`.fvuns`) results from a serial ANSYS Fluent simulation (as well as parallel), mesh files must be exported using the `fieldview-unstruct-grid` text command.

• User-Defined Functions (UDFs)

- For serial user-defined functions (UDFs), note the following:

  → The `DEFINE_RW_FILE` macro is no longer supported for serial UDFs in the following circumstances: in a compiled UDF on Windows 2012 or earlier; in an interpreted UDF on any platform. As a workaround for such unsupported combinations, you can parallelize the UDF (as described in Parallelizing Your Serial UDF in the *Fluent Customization Manual*).

  → The macro `PRINCIPAL_FACE_P` can only be used in compiled UDFs, not only in parallel, but now in serial as well.

  → Interpreted UDFs cannot be used with an Infiniband interconnect not only when running in parallel, but now in serial as well. The compiled UDF approach must be used instead.

  Note that new macro types that become available in version 19.0 and later are not guaranteed to be supported if they are not parallelized (see Parallelizing Your Serial UDF in the *Fluent Customization Manual*).

• Serial Processing

- If the network connection is lost during a serial (as well as a parallel) calculation, the ANSYS Fluent session may terminate abnormally.
- ANSYS Fluent uses several TCP/IP ports for communications and error handling. Port conflicts with other programs trying to use the same ports are handled by ANSYS Fluent and generate warnings similar to the following:

```
428: mpt_accept: warning: incorrect exercise message "GET /" from 10.1.0.188 on port 56564
```

  Long running large sessions are more prone to generating such warnings, but these are generally safe for you to ignore.

• For a list of ongoing Fluent limitations listed in previous ANSYS, Inc. Release Notes, refer to Known Limitations in ANSYS Fluent 18.2 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 18.2.

• Data Import and Export

- You can simultaneously export any number of variables to the AVS and/or PATRAN file formats.
- Reading and writing files using the hierarchical data format (HDF) is now available in the serial version of ANSYS Fluent.

- You can now read a parallel data (`.pdat`) file in the serial version of ANSYS Fluent. Note that you must be on the same platform as that used to write the file.
- When using the parallel version of ANSYS Fluent, you can now export custom field functions to all available formats (that is, you are no longer limited to just ANSYS CFD-Post, EnSight Case Gold, and Fieldview Unstructured).

- Models
  - In Release 18.0, the `F_STORAGE_R(f, t, SV_DPMS_EROSION)` macro for storing erosion rates at faces was replaced by `F_STORAGE_R_XV(f, t, SV_DPMS_EROSION, EROSION_UDF)`. The new argument `EROSION_UDF` is used internally and does not require user input. Starting with R18.0, journals or scripts that use the macro must be updated accordingly.
  - When running the parallel solver with the shell conduction model, junctions between coupled walls and periodic boundaries are now supported.
- Boundary Conditions
  - When you have a wall for which you have enabled one of the **High Roughness (Icing)** models in a case using the **Spalart-Allmaras** turbulence model, it is now possible to run the solver in serial.
- Mesh
  - Dynamic gradient adaption based on custom field functions is supported when running ANSYS Fluent in parallel (as well as serial).
  - If you are solving your sliding mesh model in several stages, whereby you run the calculation for some period of time, save case and data files, exit ANSYS Fluent, start a new ANSYS Fluent session, read the case and data files, continue the calculation for some time, and so on, it is no longer necessary to delete the mesh interface(s) before saving the case file.
- Field Variables
  - The **Boundary Volume Distance** field variable (in the **Adaption...** category) is no longer only available in the serial version of ANSYS Fluent, but can now be used in the parallel version as well. This applies to any functionality that uses field variables, such as displaying contours, creating custom field functions, creating surface report definitions, and so on.
- Graphics, Postprocessing, and Reporting
  - Any temperature units can be used for plotting and reporting minimums, maximums, standard deviations, and averaged quantities.
  - You can change the view during playback for 3D **HSF File** and **In Memory** animations after disabling **Use Stored View** in the **Playback** dialog box.
  - (Windows only) It is no longer necessary to disable the **Fade or slide menus into view** setting in Windows to avoid seeing context menu artifacts in the Fluent graphics window(s).
- User-Defined Functions (UDFs)
  - The `Get_Input_Parameter` macro can be called from either the host or the compute nodes when running Fluent in parallel.

## 1.5. Updates Affecting Code Behavior

This section contains a list of code changes implemented in ANSYS Fluent 18.2 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.

---

### Serial Processing

- The serial version of ANSYS Fluent has been revised in order to be more consistent with the parallel version, and will now interact with a host process and a single compute-node process. While such consistency has the benefit of adding new features to serial, it also introduces some new limitations and changes in code behavior, as described in the various sections of this document. **Workaround:** If you encounter unacceptable changes in serial that cannot be resolved by other means, you can revert to a version that is similar to serial from the previous release; note that this workaround will not be available indefinitely, but has been extended to this release in order to provide time to migrate to the new version. You can only revert when launching ANSYS Fluent: select the **Parallel** processing option in Fluent Launcher and enter `0` for the number of **Processors**; alternatively, include the argument `-t0` when launching from the command line.

### Solver-Numerics

- A correction has been made for cases that use a segregated pressure-velocity coupling algorithm (SIMPLE, SIMPLEC, or PISO) together with a fluid cell zone that has fixed values for the velocity components. This may improve the predicted pressure field in the vicinity of such a zone when not all of the velocity components are defined as fixed.

- The selective algebraic multigrid (SAMG) solver is not available as part of the advanced solution controls, not only in parallel, but now in serial as well; this means that all cases will use what was previously referred to as the aggregative AMG (AAMG) solver. This may affect convergence behavior compared to the previous release. (Workaround (p. 36))

### Solver-Meshing

- Improvements have been made in the handling of overset meshes, and so overset interfaces may change, and accuracy and robustness may improve compared to previous releases.

- For overset meshes, the least squares interpolation implementation has been optimized such that it is of the same computational speed as the inverse distance interpolation method.

- The graphics window now requires a refresh after you perform the scale operation.

### Heat Transfer

- When you are executing a transient simulation in serial ANSYS Fluent and you want to read the solar load data file from a serial run performed in a version prior to 18.2, you must now use the following text command: `define/models/radiation/solar-parameters/autoread-solar-data` (for details, see Automatically Reading Solar Data in the *Fluent User's Guide*).

### Reacting Flow

- When using the In-Situ Adaptive Tabulation (ISAT) method for chemistry solvers in a serial session, the node ID number `-0` will be added to the names of ISAT tables and (if **Verbosity** is enabled) monitor files.

### Discrete Phase Model

- The ability to export transient particle history data to separate FieldView files at each time step is now available. When you include the character string `%t` in the particle file name (in the **Automatic Particle History Data Export** dialog box), Fluent will replace it with the current time step at the time of export. The files written this way can be opened in FieldView 14 or newer.

- A new algorithm is used to compute the Lagrangian wall film mass, height, and temperature. The algorithm addresses issues of uneven and patchy wall film appearance, reduces the dependence of the film calculation accuracy on the number of wall film particles and the particle time step, and improves the consistency of the parallel runs. Consequently, the results of calculations involving the Lagrangian Wall Film model may differ from those obtained with earlier versions. You can revert to the old formulation by using the following Scheme commands:

  ```
  (rpsetvar 'dpm/enable-spotty-film-corrections? #f)
  ```

  ```
  (dpm-parameters-changed)
  ```

- In the serial version of ANSYS Fluent, **Shared Memory** is no longer the only available parallel processing method, and the default selection has changed from **Shared Memory** to **Hybrid** for new and old case files.

### Eulerian Multiphase Models

- For Mixture multiphase simulations run in double precision, if the volume fraction prediction falls below **Minimum Vol. Frac. for Matrix Solution** (specified in the **Solution Limits** dialog box), then ANSYS Fluent will now use the limiting value in the matrix solution for phase-specific equations such as Momentum, Energy, Turbulence, Species, UDS, and other applicable equations. As a result, you may see an improvement in solution robustness and convergence.

- The Equilibrium model is no longer supported for modeling species mass transfer in multiphase simulations. When reading a case with the Equilibrium model from earlier releases, ANSYS Fluent will change it to the Two Resistance model. This may result in more accurate and robust solutions.

### Field Variables

- In the serial version of ANSYS Fluent, there are changes in the **Cell Info...** field variable category: the **Cell Partition** variable is replaced with **Active Cell Partition** and **Stored Cell Partition**, and the **Cell Id** variable is now available. These changes affect any functionality that uses field variables, such as displaying contours, creating custom field functions, creating surface report definitions, and so on.

### Adjoint Solver

- Changes have been made so that the nodal sensitivities on a non-conformal interface are now calculated. This may have an effect on design results, such as for optimal displacement and morphing. You can use the following Scheme command to revert to the old implementation if you do not want to consider sensitivities on the sliding interface:

```
(rpsetvar 'adjoint/exclude-sliding-boundary-sensitivity? #t)
```

- Implementations of the following are improved in the adjoint solver, and so cases involving these may yield more accurate results:
  - second-order energy equation implementation

    This scheme is used when the second-order momentum scheme is chosen in the adjoint solver.
  - k-ω model
  - k-ε model (all near-wall treatments except for user-defined wall functions are now supported)

**User-Defined Functions (UDFs)**

- When compiling a UDF source file through the text user interface (TUI) using the serial version of ANSYS Fluent, note the following changes:
  - When setting up the folder structure, you must now create *two* build folders in the architecture folder for each version of the solver. For example, instead of only creating a `3d` folder, you must build a `3d_node` folder and a `3d_host` folder. You must then copy `user.udf` (for Linux) or `user_nt.udf` and `makefile_nt.udf` (for Windows) to both version folders. For details, see Set Up the Directory Structure in the *Fluent Customization Manual*.
  - When building the UDF library, you must now edit both copies of `user_nt.udf` (for Windows) or `user.udf` (for Linux), that is, the one in the host folder and the one in the node folder. You will need to specify the appropriate `CSOURCES`, `HSOURCES`, and so on. Note that for Windows, you must set `PARALLEL_NODE = none` for the host version and one of the other options (that is, `ibmmpi`, `intel`, or `msmpi`) for the node version, and use the Visual Studio command prompt window to go to each version folder and type `nmake`. For details, see Build the UDF Library in the *Fluent Customization Manual*.
  - When building a shared library for precompiled object files that are derived from external sources, you must now copy the precompiled object files (for example, `myobject1.obj myobject2.obj` for Windows) to all of the architecture/version folders (for example, `3d_node` and `3d_host`), and then edit the `user_nt.udf` (for Windows) or `user.udf` (for Linux) in each of the architecture / version folders. For details, see Link Precompiled Object Files From Non-ANSYS Fluent Sources in the *Fluent Customization Manual*.

**Fuel Cell and Electrolysis Add-On Module**

- A fix was introduced to the Fuel Cell and Electrolysis module. As a result, the current density calculation may show differences when modeling membrane as a solid zone.

**User Interface**

- The user interface for mass-flow outlets has been corrected to no longer display the **Acoustic Wave Model** group box, as such settings have never been supported. For case files set up in the previous release of ANSYS Fluent, such settings will be ignored (and therefore may result in more accurate results).

# Chapter 2: CFX Release Notes

The following sections contain release information for Release 18.2 of ANSYS CFX.

**Note**

CFX-RIF flamelet library generation has been discontinued as of the current release. CFX-RIF flamelet libraries generated from previous releases are supported in the current release.

2.1. Supported Platforms
2.2. New Features and Enhancements
2.3. Incompatibilities
2.4. Updates Affecting Code Behavior

## 2.1. Supported Platforms

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

## 2.2. New Features and Enhancements

This section lists features and enhancements that are new in Release 18.2 of ANSYS CFX.

- There is a modeling option for high wall roughness, especially suited for icing applications. For details, see Wall Roughness in the *CFX-Solver Modeling Guide*.
- For new Transient Blade Row cases involving mesh deformation, the mesh stiffness option `Blended Distance and Small Volumes` is selected by default. For details on this option, see Blended Distance and Small Volumes in the *CFX-Solver Modeling Guide*.
- Stage (Mixing-Plane) interfaces with the `Constant Total Pressure` option can now be used with gas mixtures.

## 2.3. Incompatibilities

This section describes the operational changes, the procedural changes (actions that have to be done differently in this release to get an outcome available in Release 18.1), and the support changes (functionality that is no longer supported) in Release 18.2 of ANSYS CFX.

- The `Virtual Mass Force` is no longer written to the results file by default. If you want to postprocess this variable, add it to the Extra Output Variables List. For details, see Extra Output Variables List in the *CFX-Pre User's Guide*.

## 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 18.2 and Release 18.1.

- For cases involving mesh deformation, specified displacement boundaries now show exactly the prescribed value in CFD-Post. In the previous release, the observed displacement at a stationary boundary (for example) could have non-zero values due to numerical round-off.

# Chapter 3: TurboGrid Release Notes

The following sections contain release information for Release 18.2 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 18.2 of ANSYS TurboGrid.

- The **Line of rotation on hub and shroud** settings moved from `Geometry` to `Topology`. For details, see Split Mesh Regions in the *TurboGrid User's Guide*.

# Chapter 4: ANSYS BladeModeler Release Notes

The following sections contain release information for Release 18.2 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 18.2 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 18.2 of ANSYS CFD-Post.

- Added support for FENSAP-ICE file formats (grids and solutions).
- Reports can now be written in `.arz` format, a file format supporting 3-D interactive views and readable by the ANSYS Viewer.

# Chapter 6: Polyflow Release Notes

The following sections contain release information for ANSYS Polyflow 18.2.

6.1. New Features
6.2. Supported Platforms
6.3. New Limitations in ANSYS Polyflow 18.2
6.4. Past Versions of ANSYS Polyflow Release Notes

## 6.1. New Features

The new features in ANSYS Polyflow 18.2 are as follows:

- The viscosity ratio description has been enhanced in the POLYFLOW User's Guide and in the POLYMAT User's Guide.
- A new Workbench template is available (`Bird_Carreau_Viscosity_Asymptotic_Slip`) and is designed to fit the material parameters of the Bird-Carreau law and the asymptotic slip law on the basis of experimental extrusion data. See Choosing a Polyflow Project Template for details.
- Scalability of the solver has been improved for Generalized Newtonian flows. On large 3D cases, the gain on elapsed time ranges from 10 to 40%.

## 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 18.2

There are no new limitations to note for ANSYS Polyflow 18.2. For limitations that are present in ANSYS Polyflow 18.2 but that were discovered during previous releases, see Known Limitations in ANSYS Polyflow 18.2 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\v182\polyflow\polyflow18.2.x\help\index.htm`

- For Linux:

  *path* `\ansys_inc\v182\polyflow\polyflow18.2.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 `polyflow18.2.0`).

# Chapter 7: Forte Release Notes

The following sections contain release information for Release 18.2 of ANSYS Forte.


7.1. New Features and Enhancements
7.2. Resolved Issues since Forte 18.1
7.3. Supported Platforms


## 7.1. New Features and Enhancements

This section lists new features and enhancements in Release 18.2 of ANSYS Forte CFD, organized by topic.

### Simulation Interface

- Added ability to specify a material point for a region that moves with a specified moving boundary. This can be useful in case there is zero or very limited squish region in an engine configuration, for example.
- New Valve Lift Profile Utility adjusts a valve profile to account for the valve opening/closing threshold.
- To avoid performance impacts of opening and closing very large files during simulation, changed the default file output option for Spatially Resolved data to be one solution per file. This is the recommended option for optimal performance.
- For initializing spray injection, allow the specification of a log-normal droplet-size distribution, in addition to the existing uniform and Rosin-Rammler distribution options.
- Changed the default setting for the Visibility tree, such that one tree on the left is shown by default. A user preference can be set to revert to the previous default of split tree view.
- Changed the default setting for the Reference Time Frame to be visible by default. A user preference can be set to hide the Reference Time Frame from the project tree, if desired.

### Job Submission, Monitoring, and Running Options

- Automatically monitor the instantaneous mass flow rates across the interface between a port and the engine cylinder, reporting the mass flow rates and accumulated mass flow as a function of crank angle or time for an engine.

### Engineering Models and Computation

- For spark-ignited engines, significantly enhanced the Flame Speed Library, which provides on-the-fly blending of flame-speeds for fuel blends and accurate flame speed values for a wide range of fuel components. The number of fuels available was significantly increased, adding: ethylene, acetylene, propene, allene, propyne, 1,3-butadiene, *n*-nonane, *iso*-dodecane, cyclohexane, *n*-butylbenzene. In addition, the range of applicability was increased, with more coverage for low temperatures and low pressures and more resolution of the tables overall. The accuracy of data retrieval from the Library was also significantly improved.

- EGR estimation by ANSYS Forte when computing the mixture state for laminar-flame-speed table look-up is now consistent with that used by the ANSYS Chemkin-Pro flame-speed simulator.

## 7.2. Resolved Issues since Forte 18.1

### Simulation Interface

- The Preview Mesh in the Simulation Interface is now consistent with the on-the-fly automatic mesh generation (for fixed mesh refinements).
- Fixed a problem encountered when editing the table entries in an Injection profile. [DE147616]

  Fixed a problem encountered when trying to View/Edit project data as text. [DE147617]

  Corrected version-number text in the Sector Mesh Generator interface. [DE148406]

  Prevent selection of multiple boundaries for boundary-condition-based probes, since the probe requires a single boundary condition for each definition. [DE148971]

  Allow the surface checker to be rerun after removing the report of problem vertices. [DE149404]

  Fixed an issue that caused the Seat the Valve utility to fail to find a seated position. [DE150673]

  Added missing parameter-study option for Spray parcel # initialization control options. [DE151964]

### Job Submission, Monitoring, and Running Options

- Corrected the behavior of Monitor Probes, such that the monitor plots respond to a user's specified units preference. [DE148550]
- Improved the documentation and consistency of the command-line interface (CLI). [DE149174, DE149178]
- Allow units to be specified for profiles when using the command-line interface (CLI). [DE149592]
- Removed the buffering behavior when writing spatially averaged data to the monitored `*.csv` files and `*.ftavg` file. [DE149593]

### Engineering Models and Computation

- Addressed issues that resulted in failures while running with large numbers of cores over several nodes with MPI. [DE147617, DE149047, DE149265, DE149336, DE149342, DE149477, DE150973, DE153384]
- Addressed issues found in running arbitrary sector sizes with automatic mesh generation. [DE148949, DE149109, DE149247, DE150742]

## 7.3. Supported Platforms

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

# Chapter 8: Chemkin-Pro Release Notes

The following sections contain release information for Release 18.2 of Chemkin-Pro:

8.1. New Features and Enhancements
8.2. Resolved Issues since 18.1
8.3. Supported Platforms

## 8.1. New Features and Enhancements

### Chemkin-Pro

- Enable the inclusion of real-gas compression effects for all Chemkin-Pro reactor models, expanding on the initial capability available in 18.1.
- Performance improvement, especially for smaller reaction-mechanism sizes.
- Allow reaction-rate sensitivity analysis for surface chemistry in transient CVD reactor simulations.
- Allow user-specification of the inlet cross-section area to determine mass fluxes for the Planar Shear-flow Reactor model.

### Reaction Workbench

- New option for the methods available for mechanism reduction, which is based on path-flux analysis (PFA).
- New utility to generate critical properties for a species or a set of species. The data can be output in a format compatible with the thermodynamic data input requirements for Chemkin-Pro's real-gas simulations.

## 8.2. Resolved Issues since 18.1

This section lists issues and limitations that existed in previous releases, but that were resolved and removed in Release 18.2.

### Chemkin-Pro

- Fixed a bug that caused the solution variable count to be incorrect when the SI engine model is used but the gas heat release equation is NOT turned on. The work-around was to turn on the "Integrate Heat Release by Gas-Phase Reactions" option on the "Output Control -> Heat Release" panel of the SI engine projectfile.
- Fixed an error that occurred when a warning is triggered and reported during pre-processing of the chemistry-set data for the last reaction, when real-gas data are present in the chemistry set. [DE148639]
- Fixed an issue that caused Continuations to be unavailable for certain reactor model options.

- Fixed an issue that prevented the display of the Species panel for the Partially Stirred Reactor model. [DE150424]
- Removed an error occurrence from processing a pressure fall-off reaction when the name of the third-body species, that is, the (+species), contains a parenthesis. [DE153361]

## 8.3. Supported Platforms

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

# Chapter 9: FENSAP-ICE Release Notes

The following sections contain release information for ANSYS FENSAP-ICE 18.2.


9.1. New Features and Enhancements in ANSYS FENSAP-ICE
9.2. Beta Features


## 9.1. New Features and Enhancements in ANSYS FENSAP-ICE

### FENSAP

- Adiabatic-wall air flow for icing calculations
  - For turbomachinery icing and other internal icing problems, specifying stagnation +10 K temperature on the walls heats the flow artificially and affects the accuracy of results on downstream components.
  - To remedy this issue, icing can now be calculated with flows using adiabatic walls instead of isothermal walls.
  - EID feature must be enabled within FENSAP which post-processes the adiabatic solution to extract the heat transfer coefficients needed by ICE3D to calculate the energy source terms in the icing model, in addition to calculating EID.
  - For adiabatic flow solutions imported from CFX or Fluent, an EID-only FENSAP run must be executed and provided as the input to subsequent DROP3D and ICE3D runs.
- EID Speed-up and other changes
  - EID calculations now take about 20 iterations to complete, and are capped at a maximum of 100 iterations in case the 1e-12 residual criterion is not met. Previously EID would take hundreds of iterations, and sometimes complete the total number of iterations the user specified for the primary air flow.
  - The solver panel settings for EID has been removed, CFL and number of iterations are now automatically set within FENSAP.
  - EID runs now require that the source flow solution is calculated using all adiabatic walls, in contrast to walls with temperature specified in the prior versions. An error message will ensue if EID is executed with a flow solution containing non-adiabatic walls.
  - EID is not appended to the FENSAP solution file “soln” instead of “surface.dat” file, which removes the surface.dat file from the workflow entirely.
- Energy equation solver speed up
  - Energy equation solution algorithm upgraded to require less linear solver iterations, reducing default GMRES iterations from 160 to 30 to save calculation time per time step.
- Energy equation sub-iterations for air flow

- Possibility to do energy equation sub-iterations within a time step to increase convergence rate and to enhance convergence of difficult cases.
- Performing two energy equation iterations within a time step increases the convergence rate up to 25%.
- Doing more than two iterations has no significant advantage.

- Weak form outlet BC
  - The possibility to impose the exit back pressures as surface integrals (weak form) rather than nodal values (strong form) is provided to let the pressure values locally float about the specified value.
  - The weak form outlet BC allows pressure variations in the wakes to pass more freely through the exit planes and improves convergence.
- Wall distance
  - Wall distance calculations for some grids were unintentionally slowed down in the previous version due to an unnecessary increase in the resolution of the method of computation. The issue has been addressed and the intended speed-up of wall distance in R18.1 is now complete.
- Miscellaneous
  - A final non-numbered solution file is written alongside the numbered files for FENSAP, DROP3D, and C3D when the "Do not overwrite" mode is enabled.

ICE3D

- Enabled writing of "pitchdata" files in Simple Reinjection mode.
- ICE3D-TURBO runs can now use restart files.

GENERAL

- Automatic specification of node periodicity for 2D grids.
  - FENSAP grid format specification for 2D calculations is a 1-element wide grid where one side is declared symmetric and periodic to the other side.
  - This is done to keep the flow assembly algorithms identical to 3D while only solve a single set of nodes as expected from a 2D solution process.
  - The extra step of having to set the nodal periodicity is no longer required at the grid generation or grid conversion level, instead this is done automatically within the code when the two sides of a 1-element wide grid contain the same symmetry BC family ID.
  - When the periodicity was not set, 2D calculations would take twice the time they normally should.
- Support for centerline grids where hexas are placed on the centerline in a diamond-like configuration, where an edge of a single hexa makes up the centerline instead of collapsed facets of hexas placed side-by-side.
- ANSYS Product Improvement Program (APIP) - Now supported by FENSAP-ICE. A new settings panel permits to opt-in/out. If not yet defined for R18.2, the settings panel will display upon first launch.

CFD-POST

- "View with CFD-Post" is now using a native FENSAP file reader, instead of CGNS.

WORKBENCH

- ICEM CFD mesher can be connected directly to FENSAP-ICE systems.
- CFD-Post connection now uses a native reader, multiple systems can be connected to a single CFD-Post to perform data comparisons.

## 9.2. Beta Features

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

- Icing with vapor transport
  - A new vapor transport module is added to DROP3D that calculates the local vapor pressure and relative humidity in the domain to better inform ICE3D of local evaporative conditions.
    - → When the vapor solution file is available, ICE3D uses the local vapor pressure to determine the evaporative heat fluxes instead of using the free stream vapor pressure specified in the form of RH% setting in the ICE3D settings.
    - → This increases the fidelity of icing calculations especially in turbomachines where relative humidity progressively changes through engine stages.
    - → The air flow must be obtained using adiabatic walls in order to calculate local RH correctly near the walls - this also implies that EID has to be enabled.
    - → “Wet walls” can be specified in the vapor transport model where the humidity is maintained at 100% and evaporation is possible.
- Fogging/defogging model
  - A fogging/defogging model added to ICE3D that can simulate basic condensation, frost formation, and defogging/defrosting scenarios.
    - → In the airflow solution, a “cold” temperature must be specified on walls where fogging is to be calculated (i.e. windshields, internal passage walls, etc.).
    - → DROP3D must be executed in vapor-only mode to calculate vapor concentration and condensation rates within the domain.
    - → ICE3D’s fogging model keeps the wall temperatures constant and calculates the theoretical conductive heat flux that would occur as condensation releases heat onto the walls.
    - → Fogging model currently cannot be combined with in-flight icing calculations.

# Part III: ANSYS Electronics Products

Release notes are available for the following ANSYS Electronics products:

Icepak (p. 59)

# Chapter 1: Icepak Release Notes

Release 18.2 of the ANSYS Icepak application offers most of the capabilities from previous releases plus many new features and enhancements.


- Introduction (p. 59)
- New and Modified Features in ANSYS Icepak 18.2 (p. 59)
- Resolved Issues and Limitations in ANSYS Icepak 18.2 (p. 59)


## 1.1. Introduction

ANSYS Icepak 18.2 is a release of ANSYS Icepak that has new features and resolved issues and limitations.

## 1.2. New and Modified Features in ANSYS Icepak 18.2

- Icepak Objects
  - Added capabilities to the network object, including polyline support for links, a context-aware right-click menu, and quick alignment and morphing of network face to an object.
- Meshing
  - Added per-assembly mesh settings to simplify global versus local mesh settings.
  - Added capability to create a mesh size region for 3D cut cells to control size of interior mesh.

## 1.3. Resolved Issues and Limitations in ANSYS Icepak 18.2

- Import/Export
  - For a specific model, slot-shaped holes are unable to be imported using an ANF file. (148783)
- Meshing
  - Meshing fails for a model with 2D hanging node mesh and uniform mesh param if the assembly boundary intersects adjacent hollow objects. (149139)
  - Hexa Unstructure mesher failed in an assembly if the bounding box of adjacent polygonal shape intersects the assembly boundary. (150761)
- Model Building
  - When rotating a prism about a centroid multiple times, small shifts of the object geometry that causes an error in the calculation. (147509)
  - When using the Copy from tool on a cylindrical block, the inner radius is not copied. (148197)

- Using F9 to change the view does not work when using Align tools. (149271)
- Object Parameters
  - K-Omega SST is not avilable for openings. (148298)
- Post-processing
  - For a heat exchanger object, Icepak does not display object face contours (facet based). (148356)
  - When viewing an object face (facet-based), the color legend cannot be moved. (148697)
- Solution
  - Heat exchanger case file generation not working when Domain shape is "None." (148287)
  - An error occurs when using Remote Solve Manager on a windows machine to a windows server. (148347)
  - Trials names with only numeric symbols produce a Fluent error. (148827)
  - State-space optimization produces an error upon completion. (150180)

# 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. 73)
ICEM CFD (p. 75)
Fluent Meshing (p. 77)

# Chapter 1: Geometry Release Notes

This section summarizes the new features in DesignModeler Release 18.2. Topics include:

**Support for Solid/Shell Structure Configurations in Shared Topology**

This release provides support for multibody part configurations in which external shells may be preserved by the Share Topology operation, such that solids and shells may coexist in the same part when portions of the shells lie outside of a solid volume. Use the new **Allow Solid/Shell Parts** property in such configurations to control whether portions of surface bodies outside a solid volume are preserved or trimmed by the Share Topology operation.

For more information, see Shared Topology and Multibody Part Configurations and Shared Topology Properties in the DesignModeler User's Guide.

# Chapter 2: SpaceClaim

For detailed information specific to SpaceClaim 2017.2, see the SpaceClaim 2017.2 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 18.2.

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:

- AutoCAD 2018 (Plug-in)
- CATIA V5 V5-6R2017 (Reader)
- Creo Parametric 4.0 (Reader)
- Inventor 2018 (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

Release 18.2 of the Meshing application contains many new features and enhancements. Areas where you will find changes and new capabilities include the following:


4.1. Meshing Application Advisories
4.2. Resuming Databases from Previous Releases
4.3. Sizing Enhancements and Changes
4.4. Contact Enhancements
4.5. Selective Meshing Enhancements
4.6. Documentation


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

## 4.1. Meshing Application Advisories

Be aware of the following advisories related to this release:

- The simplified mesh sizing user interface, which is being introduced in Release 18.2 as an advanced option, is intended to replace the current options for mesh sizing. The simplified user interface eliminates (deprecates) older options and behaviors that are no longer useful or are no longer best practices as a result of more recent improvements to the Meshing application. This new option aligns more closely with best practices.

- The following options will be deprecated as part of the simplified mesh sizing user interface and have been moved to the new Deprecated Sizing subcategory on the Options dialog box. These options will be removed from the product soon:

  – **Relevance**

  – **Number of Retries**

  – **Size Function**

- Some options no longer appear in the Details view when the simplified mesh sizing user interface is enabled.

## 4.2. Resuming Databases from Previous Releases

If you resume a database from a previous release into Release 18.2, you must check all size settings in the resumed database carefully. This is necessary because beginning in Release 18.2, the default defeature size and default minimum size are factors of the element size and react to dynamic sizing. If you specified user-defined values for these sizes before saving the database in a previous release, your user-defined values will be preserved when you resume the database into Release 18.2. However, if you retained the sizing defaults before saving the database, the retained default values of the **Min Size**, **Proximity Min Size**, and/or **Defeature Size** options may be different than the default values calculated in Release 18.2. In such cases, the original (retained) value will be set for that option in the resumed database. The value will no longer behave like a default and will not respond to dynamic sizing.

## 4.3. Sizing Enhancements and Changes

This release includes the following enhancements and changes related to sizing:

- If you make a change to the global element size, other default global mesh sizes (minimum size, maximum size, defeature size, and so on), will update dynamically. Similarly, default local mesh sizes will update dynamically if you change the global element size *and* the **Type** of the local sizing control is **Factor of Global Size**. In either case, if you have specified user-defined values for any of the sizes, they will not update dynamically. User-defined size values are always retained even when you make changes to other size values.
- If you set the new Simplified Mesh Sizing UI option to **Yes**, the mesh sizing user interface will be reconfigured to provide a simplified user experience.
- In the Details view, the controls in the Sizing group have been reordered.
- **Automatic Mesh Based Defeaturing** has been renamed Mesh Defeaturing and its **On** and **Off** options have been changed to **Yes** and **No**.
- The default global **Defeature Size** is now a factor of element size; in previous versions, it was a factor of minimum size. For local sizing, the default **Defeature Size** is still a factor of the local minimum size, unless the **Type** of the local sizing control is **Factor of Global Size**.
- The Uniform size function no longer supports a **Min Size** option.
- Assembly meshing no longer has an option to use the Uniform size function. This is because assembly meshing is driven by minimum and maximum sizes, making the option invalid.
- The presence or absence of sheets in a model no longer has an effect on how Min Size is calculated.
- New validity rules must be satisfied to generate a mesh:
  - The defeature size must be smaller than or equal to the minimum size (curvature minimum size or proximity minimum size, whichever is smaller) when the curvature, proximity, or curvature and proximity size function is being used.
  - The defeature size must be smaller than or equal to the element size when the uniform size function is being used.
  - The minimum size must be smaller than or equal to the element size.
- The Bounding Box Diagonal field has been added to provide a read-only indication of the length of the assembly diagonal.
- You can now control local mesh sizing by specifying Factor of Global Size as the **Type** of a local mesh control and entering a value for **Element Size Factor**. Depending on the entities and other options you select when you define the control, you can then use the new **Defeature Size Scale**, **Curvature Min Size Scale**, and **Proximity Min Size Scale** local size control options to control the default local mesh sizing values.
- You can use the new **Mechanical Min Size Factor (Default: 0.01)**, **CFD Min Size Factor (Default: 0.01)**, and **Defeature Size Factor (Default: 0.005)** options to set your preferences for the scale factors that will be used to calculate the corresponding default sizes. Essentially, the scale factors control the default values for global minimum size and global defeature size, as well as the default sizes used by local mesh sizing controls when **Type** is set to Factor of Global Size.

- In anticipation of their removal from the product at Release 19.0, the **Relevance**, **Number of Retries**, and **Size Function** options that appear on the **Options** dialog box have been moved to the new Deprecated Sizing subcategory.

## 4.4. Contact Enhancements

This release includes the following enhancements related to contact:

- You can select the entire **Contacts** folder or an individual **Contact Region** in the Tree and use the new **Create** > **Contact Sizing** context menu command to create Contact Sizing controls for the selected contact regions automatically.
- You can select two bodies and use the new **Go To** > Contact Sizing Common to Selected Bodies option to identify any contact sizing controls that exist between the two bodies. This feature provides an easy way for you to delete the common controls.

## 4.5. Selective Meshing Enhancements

This release includes the following enhancements related to selective meshing:

- Mixed Order Meshing is now supported by Selective Meshing. When selectively meshing bodies with different order, any existing mesh has precedence at an interface.

## 4.6. Documentation

In order to maintain consistency across ANSYS, Inc. products, the **Tutorials** section in the Meshing User's Guide has been removed and will be published as a separate document at release 19.0.

You may refer to the **Tutorials** section in the Meshing User's Guide in the Release 18.0 Help, if needed.

# Chapter 5: IC Engine Release Notes

Release 18.2 has no new features or enhancements.

# Chapter 6: ICEM CFD Release Notes

Release 18.2 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 18.2 includes new features and improvements in the following areas:

6.1. Multizone Block Editing improvements
6.2. Usability Improvements

## 6.1. Multizone Block Editing improvements

The following enhancements were made to improve block editing:

- **Blocking** → **Split Block** → **Imprint Face** now supports mapped faces as targets for imprinting. This enhancement also applies to **Split Free Block** → **By Imprint**.
- **Blocking** → **Edit Block** → **Merge Blocks** was extended to allow the selection of more than two 2D, free blocks for a single merge operation.

## 6.2. Usability Improvements

The following enhancements were made to improve usability:

- Data integrated ICEM CFD now supports direct connection from an ICEM CFD task into a FENSAP-ICE task, within the Workbench environment.
- Block faces, edges, and vertices may be grouped into **NAMED_SELECTION** subsets. All information relevant to any such subset is saved with the Blocking (`*.blk`) file.

# Chapter 7: Fluent Meshing Release Notes

The following sections contain release information for ANSYS Fluent Meshing Release 18.2:

7.1. Changes in Product Behavior from Previous Releases
7.2. New Features
7.3. Known Issues and Limitations

## 7.1. Changes in Product Behavior from Previous Releases

**Serial Processing**

The serial version of ANSYS Fluent Meshing has been revised in order to be more consistent with the parallel version, and will now interact with a host process and a single compute-node process. While such consistency has the benefit of adding new features to the serial version, it also introduces some new limitations.

**Workaround**: If you encounter unacceptable changes in the serial version that cannot be resolved by other means, you can revert to a version similar to serial from the previous release. Note that this workaround has been extended to this release to provide time to migrate to the new version and may be discontinued in future releases. You can only revert when launching ANSYS Fluent Meshing: select the **Parallel** processing option in Fluent Launcher and enter `0` for number of **Meshing Processes**. Alternatively, include the argument `-tm0 -t0` when launching from the command line (for example, `fluent 3d -tm0 -t0 -meshing`).

**CAD Import**

- When importing a CAD file, the **Length Unit** is now set to **mm**, by default.
- New commands are available for importing CAD geometry and setting up import parameters.

> **Note**
>
> Commands from previous versions may be discontinued in future releases.

## 7.2. New Features

The new features available in ANSYS Fluent Meshing include enhancements to existing features, and improved robustness through defect fixes.

**CAD Import**

The following enhancements have been made:

- The options in the **Import CAD Geometry** and **CAD Options** dialog boxes have been rearranged for easier access.
- The **Tessellation** options are now available in the **Import CAD Geometry** dialog box.

- For import using the **CFD Surface Mesh** option, you can specify proximity size functions scoped to faces or faces and edges.
- Enabling the **Auto-Create Scoped Sizing** option will automatically, during import, create scoped sizing controls based on the defined parameters.

**Scripting Improvements**

New API functions are available for the following:

- Managing labels for face, cell, and edge zones, including unreferenced zones.
- Reporting the count of marked faces for face zones.
- Identifying overlapping zone pairs based on join angle and tolerance.
- Setting the number of threads to use for algorithms like mesh check and quality computation.

**Data Import and Export**

The following improvements have been made:

- A new text user interface (TUI) command allows you to hide any new TUI prompts that were added for version 18.2 and revert to the 18.1 default arguments: `/file/set-tui-version "18.1"`. Using this text command can help a TUI journal created in version 18.1 to work properly in version 18.2; it must be manually added within the 18.1 journal file (for example, as the top line), or invoked in the ANSYS Fluent Meshing 18.2 session prior to reading the 18.1 journal file. Note that TUI prompts that were removed for version 18.2 are not addressed by this text command, and must be addressed manually.

  Note that all journals created using version 18.2 will now automatically include this text command, specifying the current version (that is, `/file/set-tui-version "18.2"`), to help backwards compatibility with future releases.
- When recording a TUI journal file, commands entered using paths from older versions of Fluent will be upgraded to their current path in the journal file.
- You can read multiple journal files into Fluent at the same time. It is also possible to create and run nested journal files.

**Miscellaneous Improvements**

The following improvements have been made:

- When setting up scoped prisms, you can choose to grow prisms on both sides of baffles (default) or grow prisms on a single side.
- The poly meshing process has been improved, resulting in peak memory reduction by 5-10%.
- The retriangulation process has been improved, resulting in meshing speedup (up to 45% in some cases).
- There is a significant peak memory reduction (up to 40%) for writing the mesh file in distributed parallel mode.

## 7.3. Known Issues and Limitations

The following is a list of known issues and limitations in ANSYS Fluent Meshing Release 18.2:

- Interrupting or canceling an operation may cause the application to hang or crash. A Workaround (p. 77) is available for cases that can be run in serial.
- The ability to export to the STL format (through the `/file/export/stl` text command) is not available in the serial version of ANSYS Fluent Meshing. See Workaround (p. 77).

# Part V: ANSYS Simulation Products

Release notes are available for the following ANSYS Simulation products:


Workbench (p. 83)
ACT (p. 87)
RSM (p. 93)
EKM (p. 95)
DesignXplorer (p. 97)
ANSYS Viewer (p. 99)

# 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. Fluid Flow (CFX)
1.3. External Connection Add-In
1.4. Engineering Data Workspace
1.5. External Data
1.6. External Model
1.7. Enhancement to Mechanical Model Cells
1.8. FE Modeler
1.9. System Coupling
1.10. TurboSystem Release Notes


## 1.1. ANSYS Workbench

Enhancements were made to the following areas for Release 18.2:


1.1.1. Design Point Update Enhancements
1.1.2. Units
1.1.3. Project Toolbox
1.1.4. Mechanical APDL Enhancements
1.1.5. ANSYS Workbench-Remote Solve Manager Enhancements
1.1.6. ANSYS Workbench-EKM Enhancements


### 1.1.1. Design Point Update Enhancements

#### *Using Partially Updated Results for Output Parameters When Updating Design Points in Workbench and AIM's Design Points Dashboard*

As of Release 18.2, if the update of a design point is interrupted, the partially updated icon may appear beside the values of the output parameters for the interrupted component. The next time that this design point is selected for update, a dialog box opens. To continue, one of the following buttons must be clicked:

- **Use Partially Updated**: Accept output parameters that are only partially updated as up-to-date, using the existing results to update design points.

- **Update All**: Resume or restart the interrupted update, recalculating results for all output parameters that are partially updated and then updating all design points. The update resumes if the data is retained and the analysis system supports continuing the update. Otherwise, the update restarts. The dialog box notes that this can be a lengthy process.

**Note**

In cells for design exploration systems, ANSYS DesignXplorer always updates partially updated design points to completion and publishes an informational message. Consequently, before updating, such design points display the update required icon rather than the partially updated icon. For more information, refer to the help for the ANSYS product that you are using.

### 1.1.2. Units

#### *Improved Conversion Factors for Units*

ANSYS Workbench now uses more precise conversion factors for the units "psi," "slug," and "slinch." You may see small changes compared to old results when working with these units.

### 1.1.3. Project Toolbox

The Project Toolbox will no longer show non-ANSYS systems by default. You can choose to show these systems using the **View All / Customize** button at the bottom of the Toolbox and then selecting the systems you want to view.

### 1.1.4. Mechanical APDL Enhancements

No enhancements were made in Workbench with regard to Mechanical APDL.

### 1.1.5. ANSYS Workbench-Remote Solve Manager Enhancements

Remote Solve Manager now correctly monitors the cluster status after the client application (such as Mechanical or Fluent) is closed. RSM will resume monitoring and return the correct status when the client application is restarted so that you can continue working on it.

RSM continues job monitoring when you close Mechanical and will continue to correctly update the job status.

### 1.1.6. ANSYS Workbench-EKM Enhancements

No enhancements were made in Workbench with regard to EKM.

## 1.2. Fluid Flow (CFX)

If your CFX run fails and a backup file is available, you can resume a run from the latest standard or essential backup, or equivalent transient results file. This capability is especially useful for cases with very long runtimes to avoid restarting from initial conditions or, if the solver failed due to an error, reviewing backed up results to diagnose and correct the issue.

For details on resuming a CFX run, see *Resuming a Failed Run* in Fluid Flow (CFX).

## 1.3. External Connection Add-In

In the Workbench External Connection Add-In guide, images showing the ANSYS version were updated to 18.2. Additionally, all four appendices were updated, with a new "Harmonic Acoustics" table added to both Appendix A. ANSYS Workbench Component Inputs and Outputs and Appendix B. ANSYS Workbench Internally Defined System Template and Component Names.

---

**Note**

ANSYS ACT has superseded the SDK and External Connection Add-In as the best-in-class tool set for customizing ANSYS products. We encourage all customers and partners engaged in Workbench customization to transition to ACT. ANSYS 18.2 includes the final public distribution of the ANSYS Workbench Software Development Kit (SDK) and the final release of the External Connection Add-In. Existing SDK-based content providers seeking specific end-of-life support can contact their local ANSYS account manager or established support representatives. For more information, see the release notes for ACT (p. 87).

---

## 1.4. Engineering Data Workspace

For Release 18.2, the following enhancements have been made to the Engineering Data Workspace:

- The Menetrey-Willam material model is now supported for Static Structural and Transient Structural analyses. It is useful for modeling geomechanical materials such as concrete.
- The "**Specific Heat**" property has been renamed to "**Specific Heat, $C_p$**".

## 1.5. External Data

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

## 1.6. External Model

For Release 18.2, 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.

## 1.7. Enhancement to Mechanical Model Cells

For Release 18.2, the Mechanical Model cell has no new features or enhancements.

## 1.8. FE Modeler

Release 18.2 for FE Modeler has no new features or enhancements.

Newer ANSYS technologies, such as External Model, have made the FE Modeler application obsolete. Therefore, this is notice that FE Modeler will be undocumented at Release 19.1, then fully removed at a future release.

## 1.9. System Coupling

In a coupled analyses with a sliding mesh that rotates in Fluent, the mesh in a coupled participant (such as Mechanical) should not rotate, which is standard practice for structural analysis. In previous releases, the rotation of the mesh in the coupled participant had to match Fluent's mesh rotation. This no longer true. To ensure that the mesh does not rotate, you can now use a Rotational Velocity on the coupled participant side when it is paired with a sliding mesh zone in Fluent. Cases from pre-18.2 releases will fail without modification. To run these cases in 18.2 and later, you must update the Mechanical settings to specify a Rotational Velocity condition rather than applying rotational motion to the part via a Remote Displacement or other similar mechanism.

## 1.10. 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.

---

### 1.10.1. 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 18.2. All referenced topics are in the *ANSYS ACT Developer's Guide*.

> **Note**
>
> For beta features, documentation is available 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 links for ACT beta documents.

## ANSYS Customization Suite

ANSYS ACT has superseded the SDK and External Connection Add-In as the best-in-class tool set for customizing ANSYS products. We encourage all customers and partners engaged in Workbench customization to transition to ACT. ANSYS 18.2 includes the final public distribution of the ANSYS Workbench Software Development Kit (SDK) and the final release of the External Connection Add-In. Existing SDK-based content providers seeking specific end-of-life support can contact their local ANSYS account manager or established support representatives.

ACT delivers a simple yet powerful approach to Workbench customization with an emphasis on ease of use and consistency. ACT workflows specifically accommodate the SDK user base by improving on the external application data integration opportunities offered by the SDK.

ACT workflows continue to benefit from the full-featured host platform of ANSYS Workbench. Traditional coverage includes Project Schematic exposure, ANSYS product data transfer, design exploration through parameterization, and remote execution management. Users will also discover new and time-saving features made possible only through the simplicity and product portfolio coverage of ACT.

Additionally, past SDK prerequisites of separate ANSYS installation packages, Integrated Development Environments, and code compilation routines no longer apply to ACT application creation. In-product developer tools provide automated workflow construction, essential API discovery, application verification, and deployment preparation.

Learn more by accessing the *ANSYS ACT Developer's Guide* from the ANSYS Help or logging on to the ACT Resources page from the ANSYS Customer Portal.

## Mechanical APIs

The **`ExtAPI.Graphics.ExportScreenToImage`** command provides for exporting the current graphical view in Mechanical to a PNG file. When you use this command, the exported view retains the same orientation, zoom factor, and other graphics rendering properties as the view in ANSYS Mechanical, which is essential when you want to compare views to see changes in results. For more information, see Graphical Views.

## Custom Help for ACT Wizard Extension

ACT wizard extensions for AIM are called *custom templates*. You no longer have to place custom help files for a custom template in a **help** folder within the ACT extension. You can store custom help files in any folder within the extension as long as the **<step>** blocks in the XML definition file reference the relative paths for the HTML files to display in help panels.

Additionally, for both AIM and non-AIM wizard extensions, you can now choose from two options for defining custom help for properties. You can either create HTML files or define help text directly in **<property>** blocks in the extension's XML file.

For more information, see Custom Help for Wizards.

## General Data Transfer for External Solver Systems

ACT workflows provide generic typeless connectivity to enabled custom tasks and targeted pre-installed tasks. In addition to supporting the pre-installed task groups and tasks listed in Appendix E, general data transfer now supports ACT-integrated external solver systems. For more information, see Defining Task-Level General Data Transfer.

## Encapsulation of UDF Libraries with ACT Extensions for ANSYS Fluent

An ACT extension for ANSYS Fluent can now encapsulate one or more UDF libraries. Using the new **<udf>** tag in the **<simdata>** block, you declare where the UDF folder containing these libraries is located. Because the UDF libraries are packaged with the extension, when the extension is installed, ACT can load these libraries. For more information, see Capabilities for ANSYS Fluent.

## ACT Extension Execution for SpaceClaim

The graphics window in SpaceClaim is now stable when running an ACT wizard extension for SpaceClaim. To achieve this stability and also to enhance performance, the graphics window is not updated until the callbacks for a step have been executed.

## ACT Documentation

The introduction of the *ANSYS ACT Developer's Guide* now includes a summary of all referenced extension examples for your convenience. For more information, see Summary: Extension Examples.

## ACT Debugger (Beta)

The ACT Debugger is a standalone utility for debugging ACT extensions. Introduced in 18.2 as a beta feature, you can use the ACT Debugger to observe the run-time behaviour of your app step by step, quickly locating logic errors in your code. For more information, refer to the *ACT Debugger (Beta) 18.2* document on the ANSYS Customer Portal.

## ACT Workflow Designer (Beta)

The ACT Workflow Designer makes creating custom workflows easy. While you can still manually create ACT custom workflows and extensions, you can now choose to use the Workflow Designer instead. This 18.2 beta feature automates workflow setup, relieving the burden of creating an ACT extension from scratch. Using the Workflow Designer, creation of a task group and tasks is a quick and interactive process. For more information, refer to the *ACT Workflow Designer (Beta) 18.2* document on the ANSYS Customer Portal.

## ACT App Builder (Beta)

The ACT App Builder is a standalone utility for creating ACT extensions in a visual environment. Introduced as a beta feature in 18.0 and further enhanced in both 18.1 and 18.2, 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. For information on the new resource manager, refer to the *ACT App Builder (Beta) 18.2* document on the ANSYS Customer Portal.

## 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:

```
# 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:

```
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:

```
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:

```
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)

>elemIds = [1, 2, …]
>s = [ S11X, S11Y, S12X, S12Y, S13X, S13Y, S21X, S21Y, …]
```

In this example, each element has three nodes. The stress values are located at the nodes of the elements. Thus, **`SijX`** 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:

```
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 18.2:


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


## 3.1. New Features and Enhancements

Remote Solve Manager now correctly monitors the cluster status after the client application (such as Mechanical or Fluent) is closed. RSM will resume monitoring and return the correct status when the client application is restarted so that you can continue working on it.

RSM continues job monitoring when you close Mechanical and will continue to correctly update the job status.

## 3.2. Issues Resolved in this Release

- RSM password caching from the Mechanical UI to RSM now works on Linux.

## 3.3. 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) 18.2 consists of the EKM server product and its companion web application. The following sections provide an overview of new features and enhancements in ANSYS EKM 18.2:


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


## 4.1. New Features and Enhancements

There are no new features in EKM Release 18.2.

## 4.2. Issues Resolved in this Release

Below are the major issues that were resolved since the release of EKM18.1, including issues that were addressed in the EKM 18.1.1 Service Pack.

- Compatibility issues between the current version of EKM and older versions of Workbench have been resolved.
- When uploading a folder to EKM using the File Transfer Client, files that you have excluded from the upload are no longer erroneously uploaded.
- When using the File Transfer Client to upload files/folders to EKM, uploading a Workbench project file without its associated `_files` folder no longer causes the upload to fail.
- Workbench project files are now properly cached when Workbench projects are uploaded to EKM via a cache server. This applies when using the **Save to Repository** action in Workbench, and when uploading a Workbench (`.wbpj`) file and its `_files` folder using the File Transfer Client in EKM. It also applies when Workbench projects are uploaded to the Master workspace from a Slave workspace in a Master-Slave configuration.
- When downloading files via a cache server using the File Transfer Client, the **Transfer Details** dialog box now reports the **Percentage cached** value correctly.
- When saving to a repository that contains a large number of sub-directories, you can now navigate these sub-directories with reasonable speed, and sort them alphabetically.
- When using the EKM Client API to interact with an EKM repository, users are now successfully authenticated with the server file transfer service.
- When running jobs in a multi-node Linux cluster, attempting to cancel a job that is running on a non-primary node no longer results in an exception.
- When submitting distributed CFX jobs to an HPC cluster via EKM, and the solver version is 18.0 or higher, using certain `.def` files no longer causes jobs to fail.

- When updating a Workbench project that has a space in its name, the project is now properly updated in EKM once the job is complete.

## 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 18.2. All referenced topics are in the *ANSYS DesignXplorer User's Guide*.

## New Operation for Using Existing Design Points After Non-Parametric Changes

Previously, making a non-parametric change to a model required a **Refresh** operation, which invalidated design point results and the cache of design points. An **Update** operation for generating new design points was then required. To avoid the time and costs associated with generating new design points, when you know that your non-parametric changes do not necessitate recalculating design points, you can run the new **Approve Generated Data** operation instead.

While this operation clears the cache of design points, it does not invalidate design point results. Instead, design points are treated as if they are up-to-date. To remind you that results are based on user-approved data that may include non-parametric changes, icons and cautionary notes display within the DesignXplorer interface, on standard reports, and in exported CSV files.

The **Approve Generated Data** operation applies only to design points in DesignXplorer systems. The design points in the **Parameter Set** bar remain out-of-date after a non-parametric change. For more information, see Working with Parameters.

## Ability to View Design Point Images for Solved Design Points

If a Workbench project report has been generated, you can display image files for solved design points in DesignXplorer tables and from DesignXplorer charts. When you select the root node in the **Outline** view for a DesignXplorer component that uses design points, the **Properties** view displays **Design Point Report > Report Image**. From the list of available PNG files for this property, you select the image to display. For more information, see Viewing Design Point Images in Tables and Charts.

## Design Point Update Enhancements

In cells for design exploration systems, ANSYS DesignXplorer always updates partially updated design points to completion and publishes an informational message. However, when updating design points in Workbench and AIM's Design Points Dashboard, you can now choose either to use partially updated design points or to update them. For more information, see Design Point Update Enhancements (p. 83) in the ANSYS Workbench release notes.

## Consideration of Failed Design Points When Refining a Kriging Response Surface

When refining a Genetic Aggregation response surface, failed design points are considered. In 18.2, when refining a Kriging response surface, failed design points are also now considered. The processing of failed design points is the same for both Genetic Aggregation and Kriging response surfaces. During the refinement process, if one or more design points fail, auto-refinement takes into account the failed points by avoiding the areas close to the failed design points when generating the next refinement points. The **Crowding Distance Separation Percentage** property specifies the minimum allowable distance between new refinement points, providing a radius around failed design points that serves as a constraint for refinement points. Applicable topics in the *ANSYS DesignXplorer User's Guide* are updated accordingly.

## DesignXplorer Extension Compatibility Updates

The following extensions were updated for compatibility with the 18.1 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 18.2 release are in progress and will be released promptly upon completion.

To access DesignXplorer extensions, go to the ANSYS App Store. To filter the apps available, type `DesignXplorer` in the **Search Apps** field and then click the search button. To further filter the results, make a selection in the **Product Version** field to the left.

# Chapter 6: ANSYS Viewer

The following sections contain release information for ANSYS Viewer 18.2:

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:

- You can now open ANSYS Report `.arz` files that have been published in CFD-Post.

## 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 18.2 (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 18.0 and Release 18.1 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 18.2

The following enhancements were made to ANSYS AIM Release 18.2.

- The ability to model, visualize and animate time-dependent flow, including tabular or expression-based time-dependent physics conditions, fluid-solid heat transfer, and transient startup behavior.
- Enhanced save/resume and results processing performance for fluids models with a large number of geometric faces and/or bodies through optional merging of faces within conditions and/or bodies within regions.
- Enhanced region interfaces for fluid-fluid and fluid-solid interfaces that include the specification of thermal contact conductance to model thermal interface materials.
- The ability to simulate the transport of particulates in fluid flow such as solid particles in a gas or liquid, or liquid droplets in a gas.
- Templates for fluid-solid heat transfer (conjugate heat transfer) and fluid-structure interaction have been enhanced to automate the setup of the required physics regions, material assignments and region interfaces
- The ability to model polymer co-extrusion with different generalized Newtonian fluids.
- The ability to perform topology optimization to maximize the strength or minimize the response to free vibration for structural components and assemblies.
- The ability to include a point mass including the mass and mass moments of inertia in either a static structural or modal analysis.
  - The ability to define the formulation (rigid, deformable or coupled) of how the point mass is connected to the model.
- Enhanced specification of structural boundary conditions that include conditional expressions to allow the specification of more complex loading conditions such as bearing loads.
- The ability to define the formulation (rigid, deformable or coupled) of remote force and remote displacement.
- Extended range of solution performance tuning and access to advanced solver controls for electromagnetic simulations.
- Enhanced workflow for specifying solid and stranded conductors for magnetostatic and magnetic frequency response simulation.
- The ability to display contour or vector magnetic results on a geometric edge.
- The ability to use a 3D space navigation device to pan, zoom and rotate the model display.
- The ability to graphically display the locations of minimum and maximum values for results.

- Automatic display of result values at the cursor location for contour plots on faces, planes and bodies.
- Inclusion of fluid regions in model transfer from AIM to Fluent.
- Inclusion of structural connections in model transfer from AIM to Mechanical.
- Addition of material appearance settings for the majority of materials in the material library.
- Realistic material rendering is now the default graphics option.

# Chapter 3: 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.

- 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 4: 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.

- 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 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.