



ANSYS, Inc. Release Notes



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

Release 13.0
November 2010
002913

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

Copyright and Trademark Information

© 2010 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

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

Disclaimer Notice

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

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

U.S. Government Rights

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

Third-Party Software

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

Published in the U.S.A.

Table of Contents

1. Global	1
1.1. Advisories	1
1.2. Installation	1
1.3. Licensing	1
2. Workbench	3
2.1. ANSYS Workbench 13.0	3
2.1.1. Expanded Support for RSM	3
2.1.2. ANSYS CFX in ANSYS Workbench	3
2.1.3. Improved Design Point Behavior	4
2.1.4. Archived Project Format	4
2.1.5. ANSYS SpaceClaim Direct Modeler	4
2.1.6. Using Excel with ANSYS Workbench Products	4
2.1.7. Software Development Kit	4
2.1.8. Localization on Linux	5
2.1.9. Documentation Enhancements	5
2.1.10. Incompatibilities	5
2.2. DesignModeler Release Notes	5
2.3. TurboSystem Release Notes	8
2.3.1. ANSYS BladeModeler	9
2.3.1.1. BladeGen	9
2.3.1.1.1. BladeGen New Features and Enhancements	9
2.3.1.2. BladeEditor	9
2.3.1.2.1. BladeEditor New Features and Enhancements	9
2.3.2. Vista TF	9
2.3.2.1. Vista TF New Features and Enhancements	9
2.3.3. Vista RTD	10
2.3.3.1. Vista RTD New Features and Enhancements	10
2.3.3.2. Vista RTD Incompatibilities	10
2.3.4. Vista CCD	10
2.3.4.1. Vista CCD New Features and Enhancements	10
2.3.4.2. Vista CCD Incompatibilities	10
2.4. CFX-Mesh Release Notes	10
2.5. Meshing Application Release Notes	11
2.6. Mechanical Application Release Notes	16
2.7. FE Modeler Release Notes	21
2.8. DesignXplorer Release Notes	21
2.9. Engineering Data Workspace Release Notes	22
2.10. EKM Desktop	23
2.10.1. Improved User Interface	23
2.10.2. Supports Multiple Connections to EKM Repositories	24
2.10.3. Improved Workbench Integration	24
2.10.4. Improved File Capabilities	24
2.10.5. Monitoring Transfers	24
2.10.6. Improved Search Capabilities	24
2.10.7. EKM Studio Improvements	24
3. Mechanical APDL	25
3.1. Structural	25
3.1.1. Contact	25
3.1.1.1. Surface-Projection-Based Contact	26
3.1.1.2. Geometry Correction for 3-D Contact and Target Surfaces	26

3.1.1.3. Multiple Load Step Interference Fit	26
3.1.1.4. Modeling Contact Offset (CNOF) as a Function of Location	26
3.1.1.5. Coefficient of Restitution	26
3.1.1.6. New Contact Element Output Quantities	27
3.1.2. Elements and Nonlinear Technology	27
3.1.2.1. New 2-D Reinforcing Element	27
3.1.2.2. General Axisymmetric Surface Element	27
3.1.2.3. Hydrostatic Fluid Elements	27
3.1.2.4. Preintegrated Composite Beam Sections	28
3.1.2.5. Enhanced Failure Criteria Support	28
3.1.2.6. Layer and Temperature Limits Lifted	28
3.1.2.7. Transverse-Shear Strain Formulation	28
3.1.2.8. Manual Rezoning Enhancements	28
3.1.2.9. Volumetric Force Density	28
3.1.2.10. Enhanced Ocean Loading	29
3.1.3. Linear Dynamics	29
3.1.3.1. Reusing Eigenmodes	29
3.1.3.2. Spectrum Reaction Forces and Multipoint Response Spectrum Enhancements	29
3.1.3.3. Enforced Motion Method for Mode Superposition Harmonic/Transient Analysis	30
3.1.3.4. Unsymmetric and Damped Extraction Methods	30
3.1.3.5. Solution Accuracy Improvement for Brake Squeal Analysis (QRDAMP Solver)	30
3.1.3.6. Harmonic Response Analysis	30
3.1.3.7. Spin Softening	30
3.1.4. Materials and Fracture	30
3.1.4.1. Virtual Crack Closure Technique (VCCT)	31
3.1.4.2. Response Function Hyperelastic Material Option	31
3.1.4.3. Extended Tube Material Model	31
3.1.4.4. Gurson Plasticity with Isotropic/Chaboche Kinematic Hardening	31
3.1.4.5. Creep Enhancement	32
3.1.4.6. Cap Creep Model	32
3.2. Coupled-Field	32
3.2.1. Structural Material Nonlinearities	32
3.3. Low-Frequency Electromagnetics	32
3.3.1. Stranded Coil Analysis	32
3.4. Acoustics	33
3.4.1. New Acoustic Fluid Elements	33
3.4.2. Perfectly Matched Layers (PML)	33
3.5. Thermal	33
3.5.1. New Thermal Solid Elements	33
3.5.2. Thermal Element Enhancement	33
3.5.3. Convection Analysis	33
3.6. Solvers	34
3.6.1. Distributed ANSYS Enhancements	34
3.6.2. GPU Accelerator Capability	34
3.6.3. Miscellaneous Solver Changes and Enhancements	34
3.7. Linear Perturbation	35
3.8. APDL Math	35
3.9. Commands	35
3.9.1. New Commands	35
3.9.2. Modified Commands	36
3.9.3. Other Command Enhancements	39
3.9.4. Undocumented Commands	39

3.9.5. Archived Commands	41
3.10. Elements	41
3.10.1. New Elements	42
3.10.2. Modified Elements	42
3.10.3. Undocumented Elements	43
3.10.4. Archived Elements	44
3.11. Other Enhancements	45
3.11.1. Postprocessing	45
3.11.2. Documentation	45
3.11.2.1. <i>Technology Demonstration Guide</i>	45
3.11.2.2. <i>Feature Archive</i>	46
3.11.2.3. Documentation Updates for Programmers	46
3.11.2.3.1. Routines and Functions Updated	46
3.11.2.3.2. /UPF Command for Linking UPFs	46
3.11.2.3.3. New Routines for Ocean Loading	46
3.12. Known Incompatibilities	46
3.12.1. Surface Elements	47
3.12.2. Change in Default Byte-Swapping Behavior for Binary Files	47
3.12.3. Results File Format Change	47
3.12.4. Spin Softening Default	47
3.12.5. Ocean Environment Definition	47
3.12.6. Rate-Dependent Plastic (Viscoplastic) Material Model Option	48
3.12.7. Lumped Matrix Formulation with Beam, Pipe, or Shell Elements	48
3.12.8. Contacting Area for Contact Elements	48
3.13. The ANSYS Customer Portal	48
4. AUTODYN	49
4.1. Euler Solver Enhancements	49
4.2. Interaction Enhancements	49
4.2.1. Automatic Coupling Set-Up	49
4.2.2. Efficient Treatment of Fully-Constrained Rigid Parts with Full Coupling	49
4.3. Analytical Blast Boundary	49
4.4. Remote Points and Displacements	49
4.5. Parallel Processing	50
4.5.1. HP-MPI Message Passing Protocol	50
4.5.2. Automatic Decomposition of Euler parts	50
4.6. Shells with Variable Thickness	50
5. ICEM CFD	51
5.1. Highlights of ANSYS ICEM CFD 13.0	51
5.2. Key New Features/Improvements	51
5.2.1. Workbench Integration	51
5.2.2. Geometry	51
5.2.3. Hexa	51
5.2.4. Mesh Editing	52
5.2.5. Tetra/Prism	52
5.2.6. General	52
5.3. Documentation	52
5.3.1. Tutorials	52
6. TurboGrid	53
7. FLUENT	55
7.1. Introduction	55
7.2. New Features in ANSYS FLUENT 13.0	55
7.3. Supported Platforms for ANSYS FLUENT 13.0	58

7.4. Known Limitations in ANSYS FLUENT 13.0	59
7.5. Limitations That No Longer Apply in ANSYS FLUENT 13.0	61
7.6. Updates Affecting Code Behavior	61
8. CFX	65
8.1. New Features and Enhancements	65
8.1.1. ANSYS CFX in ANSYS Workbench	65
8.1.2. ANSYS CFX in General	65
8.1.3. ANSYS CFX Documentation	65
8.1.4. ANSYS CFX-Pre	66
8.1.4.1. Efficient Handling of Large Numbers of Renderable Objects	66
8.1.4.2. Stereo Viewer Capabilities	66
8.1.4.3. Automatic Domain Interfaces	66
8.1.4.4. Additional ANSYS Element Type Support	66
8.1.4.5. License Server Checking Improvements	67
8.1.4.6. Full User Interface Support for New Solver Models	67
8.1.5. ANSYS CFX-Solver Manager	67
8.1.5.1. Automatic Display of Electro-Magnetism Plots	67
8.1.5.2. License Server Checking Improvements	67
8.1.6. ANSYS CFX-Solver	67
8.1.6.1. CFX-Solver	67
8.1.6.1.1. Turbulence	67
8.1.6.1.2. Particle Tracking	67
8.1.7. ANSYS CFD-Post	67
8.2. Incompatibilities	68
8.2.1. CFX-Pre	68
8.2.2. CFX-Solver	68
8.2.3. CFX-Solver Manager	70
8.2.4. CFD-Post	70
9. POLYFLOW	73
9.1. Introduction	73
9.2. New Features	73
9.3. Defect Fixes	74
9.4. Known Limitations	75
10. Icepak	77
10.1. Introduction	77
10.2. New and Modified Features in ANSYS Icepak 13	77
11. CFD-Post	79
11.1. New Features and Enhancements	79
11.2. Incompatibilities	80
12. AQWA	81
12.1. ANSYS AQWA	81
13. ASAS	83
13.1. ANSYS ASAS	83
13.2. ANSYS BEAMCHECK	83
13.3. ANSYS FATJACK	83
13.4. FEMGV	83

Chapter 1: Global

The information shown below apply to all ANSYS, Inc. products at the 13.0 release. Be sure to read the Release Notes for your individual product(s) for additional installation and licensing changes specific to your product(s).

1.1. 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 for information about the documentation errata, 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.

1.2. Installation

- ANSYS, Inc. products now support Windows 7.
- ANSYS, Inc. has discontinued support for the HP-UX 64 PA-RISC and the Sun SPARC 64 platforms for all products. The ANSYS, Inc. License Manager will continue to support Sun SPARC 64.
- On Windows systems, the unified installation process now automatically checks for the necessary prerequisites on your system and will install any prerequisites that are missing. You no longer have to choose to install the prerequisites as a separate step.
- The installation and product configuration utilities have been improved.
- Silent mode operations have been extended to include installs, uninstalls, product configuration, and product unconfiguration on all platforms.

1.3. Licensing

- In order to run ANSYS Release 13.0 products, you must upgrade to the Release 13.0 License Manager. The Release 13.0 License Manager will continue to support ANSYS licensing from prior ANSYS releases.
- At ANSYS Release 13.0, the license manager daemons (`lmgrd` and `ansyslmd`) have been upgraded to FLEXlm 11.8 (FLEXnet 11.8). We strongly recommend that you upgrade to this version of the license manager, regardless of whether you are upgrading to ANSYS Release 13.0. This version of the license manager supports our current licenses as well as provides support for FLEXlm Tamper Resistant Licensing (TRL) licenses. When you receive a license that contains TRL, you must be using this version of the license manager or you will not be able to run ANSYS, Inc. products.
- ANSYS, Inc. no longer requires you to choose between commercial and academic licenses when setting your license preferences, giving customers with both academic and commercial licenses greater flexibility in managing their licenses.
- The ANSYS, Inc. License Manager can now be installed silently using the `-silent` command line option. See [Silent License Manager Installation Instructions](#) for detailed information on running a silent license manager installation.
- The ANSYS Licensing Interconnect now supports the use of IP addresses in the FLEXlm options file for those settings that allow their use, such as EXCLUDE and INCLUDE.

- The **ANSLIC_ADMIN** utility now includes a queued license tracking capability. Use the **Display Queued Licenses** option under **View Status/Diagnostic Options** to see a list of capabilities that are queued and awaiting availability, and the applicable licenses that are being used.
- We have enhanced many licensing messages to include more detailed information to assist you in resolving errors. We have also added more diagnostic information, such as enhancements to the **ANSLIC_ADMIN**'s **Gather Diagnostic Information** option under **View Status/Diagnostic Options**.

Chapter 2: Workbench

2.1. ANSYS Workbench 13.0

2.1.1. Expanded Support for RSM

The Remote Solve Manager (RSM) is a job queuing system that enables computationally-intensive jobs to be queued for execution locally or on remote machines. In previous releases, RSM supported the update of the Solution cell only for Mechanical systems. ANSYS 13.0 extends support for RSM to include Solution cell update for Mechanical APDL, CFX, FLUENT and POLYFLOW systems (in addition to Mechanical). Note that CFX supports execution on remote machines only for serial and shared-memory parallel jobs that do not rely on external files (for example, profile boundary condition files, Flamelet model files, etc.) and FLUENT supports RSM update only on the local machine. For details, see [Submitting Solutions for Local, Background, and Remote Solve Manager \(RSM\) Processes](#).

Support has also been added for the update of design points via RSM, which allows the update of any parametric system or set of systems to be executed on a remote machine. At ANSYS 13.0, each design point update operation (update of a single design point, a set of selected design points, or the update of all design points) is packaged up and run on the remote machine as a single job. Simultaneous execution of design points in parallel will not be supported at ANSYS 13.0, but is planned for a future release.

2.1.2. ANSYS CFX in ANSYS Workbench

Volumetric temperature data can now be transferred from an ANSYS CFX solution to ANSYS Mechanical for one-way FSI calculations.

CFX can now make use of the Remote Solve Manager (RSM) capability. See *Expanded Support for RSM in ANSYS Workbench 13.0* (p. 3) for more details.

Most files imported into CFX-Pre (such as boundary profile files and flamelet libraries) are now registered with the project and can be archived with the project. References to these files will be automatically updated to refer to the new locations when the archive is restored.

A new option has been added to allow old solution data files to be removed without clearing the current solution data, in order to reduce the disk space used by a project. This can be accessed by right-clicking on the Solution cell and choosing Clear Old Solution Data.

CFX now provides more options for handling cases where Execution Control is specified both within CFX-Pre and within the Solution cell, which could previously lead to conflicts and hence unexpected settings being used.

The disk space used in the temporary directory has been reduced for cases where the CFX-Solver Manager continues to display monitors after a run has completed.

2.1.3. Improved Design Point Behavior

Only design points affected by a change to the project will be marked as out of date. Any change that is not relevant to the parametric study, such as adding a standalone system or making a change downstream of the parametric study, will not cause design points to go out of date. Likewise, only the out-of-date components and systems will be updated during a design point update operation. The improved behavior often reduces the amount of time and computer resources necessary for a design point update.

You can now specify whether design points will be updated beginning from the current design point (DP0) or starting from the previous design point. In some situations, it may be more efficient to update design points starting from parameter values from the previous design point, rather than starting from DP0 each time.

Output parameter values are now displayed in the Table of Design Points and Details views as they are calculated. In previous releases, no updated values were shown until the entire design point update was complete. This capability allows design points that are only partially updated to show up-to-date parameter values for those parameters which were updated successfully.

2.1.4. Archived Project Format

Workbench projects can now be archived to a Workbench-specific archive, with a `.wbpz` file extension. On Windows systems, you can double-click the `.wbpz` file to open the project. In this manner, you can work directly in an archived project and save changes back to the archive.

2.1.5. ANSYS SpaceClaim Direct Modeler

The Geometry component system now allows you to select from two different type of editors: ANSYS DesignModeler and ANSYS SpaceClaim Direct Modeler (SCDM). SpaceClaim Engineer is referred to as ANSYS SpaceClaim Direct Modeler (SCDM) in ANSYS Workbench. Unlike ANSYS DesignModeler which is a history-based parametric application, SCDM is a direct modeling application. Access and use of SCDM requires you have an existing SCDM license. For more information, see [SpaceClaim Related to CAD Integration](#) in the CAD Integration section of the ANSYS Workbench help.

2.1.6. Using Excel with ANSYS Workbench Products

Leveraging the calculation capabilities of Microsoft Office Excel, you can now perform parametric analyses to create design points and design exploration studies via the Microsoft Office Excel option in the Component Systems toolbox of ANSYS Workbench. For detailed usage information, see the [Microsoft Office Excel](#) section of the ANSYS Workbench help.

2.1.7. Software Development Kit

A Software Development Kit (SDK) for the integration of third party applications was developed for ANSYS 13.0. This SDK provides APIs, tools, and documentation that enable a software developer to write custom code to integrate external applications so that they can participate in the Workbench workflow at the project schematic level. The SDK is available as an independent installation and can be downloaded from the ANSYS Customer Portal (actual availability date may follow the release of ANSYS 13.0).

2.1.8. Localization on Linux

At release 13.0, ANSYS Workbench, including the Mechanical application, DesignModeler, EKM, FE Modeler, Design Exploration, Meshing, and Engineering Data, will be available in multiple languages on both Windows and Linux platforms. Available languages include English, French, German, and Japanese.

2.1.9. Documentation Enhancements

CAD-centric information related in general to ANSYS Workbench and specific to the component systems is now centrally located in the [CAD Integration](#) section of the ANSYS Workbench help. The section is accessible from the ANSYS Help Viewer and the CAD sections within the Mechanical and DesignModeler sections of the application help. The sectional topics include:

- [Overview](#)
- [Geometry Interface Support for Linux and Windows](#)
- [Project Schematic Presence](#)
- [Mixed import Resolution](#)
- [CAD Configuration Manager](#)
- [Named Selection Manager](#)
- [Caveats and Known Issues](#)
- [Installation and Licensing](#)
- [File Format Support](#) (with information specific to the ANSYS DesignModeler application)
- [ANSYS Teamcenter Engineering Connection](#)
- [SpaceClaim Related to CAD Integration](#)
- [Frequently Asked Questions](#)
- [Troubleshooting](#)
- [Glossary](#)
- [Updates](#)

A [Troubleshooting](#) section has been added to the Workbench documentation.

A section on [Working with Views and Workspaces](#) has been added.

Documentation on [Getting Started](#), [Design Points](#), and other discussions has been enhanced.

2.1.10. Incompatibilities

File Migration If a Mechanical (.dsdb) file from release 11.0 (or earlier) containing a Vector Principal Stress result is imported into Mechanical release 13.0 then the Vector Principal Stress display is incorrect. If you clean the solution, re-solve, and click Vector Principal Stress result, then the display will be correct.

2.2. DesignModeler Release Notes

Release 13.0 provides significant improvements in many different areas including Beam and Shell modeling, model preparation, usability and performance, and custom tools for specific analysis. Other focus areas include improved SpaceClaim integration, new CAD readers to support additional file formats and workflows, and geometry interface improvements for performance and new versions support.

The following general enhancements have been made at release 13.0:

Automatic Surface Extension

Surface extension is much more automated now. Within the Details View, a new automatic option for extending surfaces is available. When you select yes, the extension groups based on the gap value you specify are displayed. The initial value of the gap is pre-filled and a visual feedback is provided for adjusting it further. Options are available to preview the selections along with proposed solutions. You can edit the list or accept default solution to perform extensions. For detailed information, see the [Surface Extension Selection Methods](#) description in the DesignModeler section of the ANSYS Workbench help.

Visualization Tool for Connectivity

A new option is available to display edge color based on its connectivity or number of faces shared by the edge. In addition, edge thickness can also be displayed based on its connectivity to help identify connectivity issues in the model. For detailed information, see the [Edge Coloring](#) description in the DesignModeler section of the ANSYS Workbench help.

Improved Joints Handling

Joints feature now accepts both surface and line bodies as inputs. This can be used to easily form joints between line and surface bodies to ensure proper topology sharing among beams and shells. For detailed information, see the [Joint](#) description in the DesignModeler section of the ANSYS Workbench help.

Support for Custom Face Thickness

Mid-surfacing improvements include, support for custom face thickness. You can specify different thickness at face level within a body. You can also assign thickness for multiple faces in a single step.

Instancing Support

Instancing is now supported at the part level including instances imported from external CAD packages and instances defined in ANSYS DesignModeler. As a result, performance is improved and the mesh is identical on similar parts. For detailed information, see the [Instancing Support](#) description in the DesignModeler section of the ANSYS Workbench help.

Face Connect

Accessible via the Tools menu, the Connect feature now allows you to select faces as a connection type in the Details View. Face connect can be used to ensure proper topology sharing among bodies. For detailed information, see the [Connect](#) description in the DesignModeler section of the ANSYS Workbench help.

GAMBIT Reader for ANSYS Workbench

ANSYS DesignModeler allows you to import legacy GAMBIT databases into ANSYS Workbench for editing. The Real (ACIS) geometry is read from the database excluding the meshing information. In addition, if no editing is needed, the model can be transferred directly into Meshing. The GAMBIT reader is enabled via the Geometry Interface for SAT product. Note that faceted and virtual geometry cannot be read from GAMBIT databases. For detailed information, see the [Import External Geometry File](#) description in the DesignModeler section of the ANSYS Workbench help.

Decomposition Tools

Several new tools are available for greater flexibility and automation.

Edge Split by Location: Edge split by location is available to split an edge by screen locations.

Face Split by Points and Edges: Face split by points and edges is available to split a face by points, edges or a combination of both. For detailed information, see the [Face Split](#) description in the DesignModeler section of the ANSYS Workbench help.

Face Split by Locations: Via the Face Split feature in the Tools menu, you can now split a face by screen locations. For detailed information, see the [Face Split](#) description in the DesignModeler section of the ANSYS Workbench help.

Slice by Edge Loops: Via the Slice feature in the Create menu, you can now slice a solid by selecting edge loops. For detailed information, see the [Slice by Edge Loop](#) description in the DesignModeler section of the ANSYS Workbench help.

Usability Enhancements

This release contains several usability enhancements to improve the workflow and performance.

Both Side Highlighting: A new option to highlight both sides while selecting a face.

Hide Points: Option to hide points

More intuitive Fluid/Solid Type Assignment: Fluid/Solid type of multiple bodies/parts can be assigned in a single step.

Ns Propagation: Option to propagate named selection on resulting entities as a result of split/Boolean/merge/copy operation to improve automation and persistence

Spline Control Points Edits: Option is available to edit control points of a spline

Electronics Support

The Electronics menu contains a set of custom tools to automate model preparation for electronics simulation using ANSYS Icepak. It offers four different levels of simplifications for converting a complex CAD geometry into a simplified representation that can be used to perform a thermal modeling analysis using ANSYS Icepak. For detailed information, see the [Electronics](#) description in the DesignModeler section of the ANSYS Workbench help.

Geometry Interfaces: New CAD Readers

Geometry Interfaces are further expanded by introducing three new readers:

- **GAMBIT** (enabled by the Geometry Interface for SAT product)
- **JT Open** (a separate license key is required for use)
 - To import geometry represented in the lightweight JT format into Workbench for modeling, meshing and analysis by ANSYS applications.
- **Pro/ENGINEER** (a separate license key is required for use)
 - To read native Pro/Engineer models into ANSYS Workbench without requiring a Pro/Engineer installation or license.

In addition, all existing interfaces have been updated to support newer CAD releases and improved attributes processing.

For detailed CAD-related information specific to the ANSYS DesignModeler application and ANSYS Workbench, see the [CAD Integration](#) section of the ANSYS Workbench help.

Geometry Interfaces: Teamcenter Engineering Interface Improvements

You can now check-in and check-out an entire ANSYS project into Teamcenter. The ANSYS Teamcenter Engineering Connection supports project with multiple configurations and databases. In addition, you can use any CAD system/analysis system configuration available with ANSYS Workbench. For detailed information, see the [Teamcenter Engineering Connection](#) description in the CAD Integration section of the ANSYS Workbench help.

Geometry Interfaces: Smarter and Faster Update

Compare geometry on update option is available to update geometry and mesh of only those entities which have changed. This results in targeted and faster update of geometry and meshing.

Geometry Interfaces: Line/Curves Import from NX

NX interface now supports import of line bodies and curves from NX into ANSYS Workbench.

Geometry Interfaces: SpaceClaim Integration

SpaceClaim Direct Modeler is integrated with the project page and can be accessed from the geometry cell.

2.3. TurboSystem Release Notes

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

ANSYS TurboGrid is a meshing tool for turbomachinery blade rows. The release notes for ANSYS TurboGrid are given at "ANSYS, Inc. Release Notes > ["TurboGrid Release Notes"](#)".

CFX-Pre, a CFD preprocessor, and CFD-Post, a CFD postprocessor, are part of the ANSYS CFX product. Both of these products have Turbomachinery-specific features. The release notes for CFX-Pre are given at "ANSYS, Inc. Release Notes > ["CFX Release Notes"](#)". The release notes for CFD-Post are given at "ANSYS, Inc. Release Notes > ["CFD-Post Release Notes"](#)".

Release notes for the remaining TurboSystem applications are provided in the following sections:

- [BladeGen](#) (p. 9)
- [BladeEditor](#) (p. 9)
- [Vista TF](#) (p. 9)
- [Vista RTD](#) (p. 10)
- [Vista CCD](#) (p. 10)

Note

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

2.3.1. ANSYS BladeModeler

2.3.1.1. BladeGen

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

2.3.1.1.1. BladeGen New Features and Enhancements

- Vista RTD and Vista CCD are no longer available from BladeGen. For details, see [Vista RTD \(p. 10\)](#) and [Vista CCD \(p. 10\)](#).
- When you perform the **Create New Blade CFD Mesh** command (available by right-clicking the Blade Design cell of a BladeGen system), ANSYS Meshing is used to create a mesh. Formerly, ANSYS CFX-Mesh was used for this purpose. CFX-Mesh is no longer supported in ANSYS Workbench 13.0. Because the two meshing applications are different, the meshes generated by ANSYS Meshing will differ from those previously created by CFX-Mesh in earlier versions of Workbench. For more information, see [Tips on using Automated Meshing in the TurboSystem](#).

2.3.1.2. BladeEditor

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

2.3.1.2.1. BladeEditor New Features and Enhancements

- Airfoil Design Mode

Airfoil Design Mode enables you to define a blade using profiles rather than camberline and thickness definitions. In this release, this feature is primarily intended for axial turbomachinery. For details on Airfoil Design Mode, see [Blades made using Blade Section \(Airfoil Design Mode\) Sub-features in the TurboSystem](#).
- Auxiliary view (no longer a Beta feature)

This feature includes a blade-to-blade view and a blade lean angle graph.
- Blade design parameterization (no longer a Beta feature)

This feature enables you to assign an input parameter to any numeric BladeEditor feature property.

2.3.2. Vista TF

Vista TF is a tool for performing rapid throughflow analyses of rotating machinery for preliminary design purposes.

Installation note: Vista TF is always installed, but CFD-Post is required to post-process Vista TF results. Without CFD-Post, the Results cell of the Vista TF system will not be visible.

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

2.3.2.1. Vista TF New Features and Enhancements

Vista TF now has support for real gases.

2.3.3. Vista RTD

Vista RTD is a program for the preliminary design of radial inflow turbines. It can be used to rapidly generate an optimized 1D turbine design before moving to a full 3D geometry model and CFD analysis. See ["TurboSystem: Vista RTD"](#) for details on using this new version of Vista RTD.

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

2.3.3.1. Vista RTD New Features and Enhancements

Vista RTD has been improved to work for a wider range of operating conditions.

Enhancements to Vista RTD:

- Vista RTD has moved from BladeGen to ANSYS Workbench.
- Input data may be declared as parameters via the properties view.

2.3.3.2. Vista RTD Incompatibilities

The new version of Vista RTD is not backwards compatible with earlier versions. You must use the earlier versions if you want to view the Vista data for previous BladeGen models.

2.3.4. Vista CCD

Vista CCD is a program for the preliminary design of centrifugal compressors. See ["TurboSystem: Vista CCD"](#) for details on using this new version of Vista CCD.

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

2.3.4.1. Vista CCD New Features and Enhancements

Vista CCD has been improved to work for a wider range of operating conditions.

Enhancements to Vista CCD:

- Vista CCD has moved from BladeGen to ANSYS Workbench.
- Input data may be declared as parameters via the properties view.
- Vista CCD now supports real gas properties using the Redlich Kwong equation of state model. RGP files are no longer supported in this version.

2.3.4.2. Vista CCD Incompatibilities

The new version of Vista CCD is not backwards compatible with earlier versions. You must use the earlier versions if you want to view the Vista data for previous BladeGen models.

2.4. CFX-Mesh Release Notes

CFX-Mesh is no longer supported in ANSYS Workbench 13.0. In many cases, you can use ANSYS Meshing as a substitute.

Users of BladeGen should read the release note about the **Create New Blade CFD Mesh** command in [BladeGen New Features and Enhancements](#) (p. 9).

2.5. Meshing Application Release Notes

This release of the Meshing application contains many new features and enhancements, including completion of ANSYS CFX-Mesh user migration and evolutionary improvements to help GAMBIT, TGrid, and ANSYS ICEM CFD user migration. Areas where you will find changes and new capabilities include the following:

Resuming Databases from Previous Releases

Note the following when resuming databases from previous releases:

- ANSYS Workbench no longer supports the **CFX-Mesh** method. Upon import of a legacy model into release 13.0, any **CFX-Mesh** method controls will be made inoperable, and you must either delete the method manually or change it to a valid method type. If importing a *.cldb file from Release 10.0 that contains **CFX-Mesh** data, you must first take the model into Release 11.0 and save it to convert it to a supported format for use in 13.0. In either case the geometry will be maintained, but the mesh method must be replaced.

New CutCell Cartesian Mesh Method

The **CutCell** Cartesian mesh method has been added at release 13.0. **CutCell** meshing is a general purpose meshing method designed for ANSYS FLUENT. The **CutCell** meshing algorithm is suitable for a large range of applications, and due to the large fraction of hex cells in the mesh, often produces better results than tetrahedral methods. The **CutCell** method uses a **patch independent** volume meshing approach (surface mesh automatically created from boundary of volume mesh) without the need for manual geometry cleanup or decomposition, thereby reducing the turnaround time required for meshing.

The **CutCell** method is useful for meshing fluid bodies in single body parts and multibody parts; at release 13.0 it cannot be used to mesh assemblies of parts, nor a collection of loosely closed surface patches. **CutCell** is supported in the Meshing application only; it is not supported in the Mechanical application.

Orthogonal Quality Mesh Metric

When **Physics Preference** is set to **CFD** and **CutCell** meshing is being used, a shape checking algorithm based on **orthogonal quality** is used. Orthogonal quality, which is new at release 13.0, is the recommended quality criterion for CFD simulations and can be used for all types of meshes including **CutCell** and polyhedral.

Process Improvements

Process improvements such as direct meshing, mesh method interoperability, and improved failure handling have been made at release 13.0:

- Using **direct meshing**, you can selectively pick bodies and mesh them incrementally, allowing you some control over meshing order. The **Generate Mesh, Preview Surface Mesh, Preview Source and Target Mesh**, and **Preview Inflation** ease of use features all support direct meshing. In the Tree Outline, the Meshed **status icon** will now appear for a meshed body within the **Geometry** folder, or for a multibody part whose child bodies are all meshed. If you make changes after meshing that invalidate the mesh for an individual body (such as adding sizing to the body), you will need to re-mesh that body only. This is in contrast to previous releases in which the entire part would need to be re-meshed. Direct meshing is supported for the following mesh methods: **Patch Conforming Tetra, Patch Independent Tetra, MultiZone, Sweep, Hex Dominant, Quad Dominant, All Triangles, Uniform Quad/Tri**, and **Uniform Quad**. Direct meshing is enabled by default, but you can use the **Allow Direct Meshing** option to disable it.

- You can mix and match mesh methods on the individual bodies in a multibody part, and the bodies will be meshed with conformal mesh. Through this flexible approach, you can better realize the value of the various methods on the individual bodies. Refer to [Conformal Meshing Between Parts](#) for information about conformal meshing and mesh method interoperability. Also see [Interactions Between Mesh Methods](#) for information about how inflation is handled when more than one mesh method is being used.
- When you mix mesh methods in multibody parts, the manner in which topology shared by multiple bodies is protected depends on whether adjacent bodies are being meshed with [Patch Independent](#) methods and/or [Patch Conforming](#) methods. Refer to [Meshing by Algorithm](#) and [Direct Meshing](#) for information about protected topology.
- The new [Verbose Messages from Meshing](#) option controls the verbosity of messages returned to you. Depending on the setting, before meshing a message reports the subset of bodies that is going to be meshed and/or after meshing a message reports the subset of bodies that failed to mesh. In either case, you can right click on the message to view the bodies.
- The new [Extra Retries For Assembly](#) option specifies whether the mesher should perform extra retries when meshing an assembly if meshing would otherwise fail due to poor mesh quality. These retries are in addition to the number specified by the [Number of Retries](#) option. [Extra Retries For Assembly](#) is available in the [Advanced](#) group under the Details view as well as in the [Options](#) dialog box.
- For the [Number of Retries](#) option, there are new behaviors to be aware of, most of which are due to the introduction of [direct meshing](#) and [mesh method interoperability](#):
 - Retries will not occur if you are using the [Patch Independent Tetra](#) or [MultiZone](#) mesh method in combination with any other solid mesh method to mesh bodies contained in the same part, or if you are using [Uniform Quad/Tri](#) or [Uniform Quad](#) in combination with any other surface mesh method to mesh bodies contained in the same part.
 - If [Number of Retries](#) is set to **0** and a single body in a multibody part fails during [direct meshing](#), the mesher returns as much of the mesh as possible. Bodies with valid meshes will have a meshed state while bodies with invalid or partial meshes will have an unmeshed state. You can display and examine the partially meshed bodies and then apply more mesh controls to correct any problems you find. If [Number of Retries](#) is set to a value greater than **0** and any body fails to mesh, the mesher returns nothing for the given part. The return of a partial mesh is applicable to the [Quad Dominant](#), [All Triangles](#), [Uniform Quad/Tri](#), [Uniform Quad](#), [Patch Independent Tetra](#), and [MultiZone](#) methods only. The [Patch Conforming Tetra](#), [Sweep](#) (general or thin), and [Hex Dominant](#) methods cannot return partial meshes.
 - If you are performing [direct meshing](#) and at least one body of a particular part has been meshed successfully, no additional retries will occur if the mesh of a subsequent body within that same part fails.
 - For shell models, if the Advanced Size Function is on, the default values of [Min Size](#), [Max Face Size](#), and [Defeaturing Tolerance](#) are reduced automatically with each subsequent retry.

MultiZone Mesh Method

The following [MultiZone](#) mesh method enhancements have been made at release 13.0:

- Source face automation has been improved.
- [Baffle meshing](#) is supported by the [MultiZone](#) mesh method for free meshing. The body with a baffle must be meshed with a free mesh of tetrahedral elements. For this reason, you must set the [Free Mesh Type](#) to [Tetra](#) for bodies with baffles.
- [Program Controlled inflation](#) is supported by the [MultiZone](#) mesh method.

- The **MultiZone** mesh method now supports the **Smooth Transition** option for the **Inflation Option** control, along with the previously-supported **Total Thickness** and **First Layer Thickness** options. **Smooth Transition** is the default for **MultiZone**.
- Improvements in imprinting include improved side face and body handling, as well as new support for models that contain **multiple connected internal loops**.
- While mixing **Sweep** and **MultiZone** mesh methods, pre-meshed faces may be used in these ways:
 - Mapped faces can be supported as side faces when **MultiZone** or **Sweep** is used to mesh subsequent bodies. The pre-meshed faces may have been generated using either **General Sweep** or **MultiZone**. There are limitations on how the face is mapped. Simple mapped faces (that is, 4-sided) are supported; however, more complicated sub-mapped cases may cause problems.
 - Mapped faces can be supported as source faces.
 - Free faces (where mesh does not have a quad mapped pattern) can be supported as source faces only.

Patch Independent Tetra Mesh Method

The following **Patch Independent Tetra** mesh method enhancements have been made at release 13.0:

- Improvements have been made in quadratic memory handling.
- The **body sizing** control is supported by the **Patch Independent Tetra** mesh method.
- You can use the **Smooth Transition** and **Growth Rate** controls to further define the **Patch Independent Tetra** mesh method. The value of **Smooth Transition** determines whether the Octree volume mesh generated from the **Patch Independent Tetra** mesh method should be kept or whether it should be replaced with a Delaunay volume mesh starting from the Patch Independent surface mesh. When **Smooth Transition** is on, the volume mesh will be a Delaunay mesh. When **Smooth Transition** is off, the volume mesh will be an Octree mesh. The **Growth Rate** value represents the increase in element edge length with each succeeding layer of elements. Its **Default** value is affected by the settings of the **Use Advanced Size Function** and **Smooth Transition** controls.
- The new **Feature Angle** control specifies the minimum angle at which geometry features will be captured when using the **Patch Independent Tetra** mesh method. If the angle between two faces is less than the specified **Feature Angle**, the edge between the faces will be ignored, and the nodes will be placed without respect to that edge. If the angle between two faces is greater than the **Feature Angle**, the edge should be retained and mesh aligned and associated with it (note the edge could be ignored due to defeaturing, etc.).

Uniform Quad/Tri and Uniform Quad Mesh Methods

The following **Uniform Quad/Tri** and **Uniform Quad** mesh method enhancements have been made at release 13.0:

- If you select the **Uniform Quad/Tri** or **Uniform Quad** mesh method to mesh a multibody part that contains a mix of line bodies and surface bodies, all surface bodies and all line bodies that share edges with surface bodies will be meshed with the selected method. Any remaining line bodies (where only vertices are shared with surface bodies) will be meshed with the **Quad Dominant** mesh method.
- The **Element Midside Nodes** option is now supported for the **Uniform Quad/Tri** and **Uniform Quad** mesh methods, allowing you to choose between a quadratic or linear mesh.
- The **Uniform Quad/Tri** and **Uniform Quad** mesh methods are available for 2D models.

Defeaturing Controls

The following enhancements to defeaturing have been made at release 13.0:

- The **Pinch** group of global mesh controls has been replaced by the new **Defeaturing** group. The **Pinch** controls are now located under the **Defeaturing** group, along with the new global **defeaturing tolerance** controls and **loop removal** controls. Turning on the new **Automatic Mesh Based Defeaturing** option exposes the **Defeaturing Tolerance** option, where you can specify a global tolerance for defeaturing. Using the loop removal controls—which apply only to sheet models—you can instruct the Meshing application to remove loops automatically according to the criteria you specify. Prior to meshing, you can use the **Show Removable Loops** feature to preview the loops that will be removed according to the current settings. The user interface controls for defeaturing tolerance and loop removal (where applicable) are now consistent across the **Patch Conforming Tetra**, **Patch Independent Tetra**, **MultiZone**, **Sweep**, **Hex Dominant**, **Quad Dominant**, **All Triangles**, **Uniform Quad/Tri**, and **Uniform Quad** mesh methods.
- The **Pinch** feature has been extended to include support for face-edge and face-vertex pinch controls. In addition, overall usability of the **Pinch** feature has been improved with the introduction of the **Set As Pinch Master/Slave** and **Add To Pinch Master/Slave** context menu options and the **Snap to Boundary** control.
- The **Pinch** feature now supports the use of the **same master** in more than one manual pinch control. This is true for all types of manual pinch controls: edge-edge, edge-vertex, vertex-vertex, face-edge, and face-vertex. When multiple pinch controls use the same master, the aggregate of the pinch controls is used to determine the pinch.

Sheet Model Defaults

The following enhancements to sheet model default handling have been made at release 13.0:

- Better defaults for sheet models have been implemented. When **Use Advanced Size Function** is on, the default **Defeaturing Tolerance** for sheets is 75% of the value of **Min Size**. (For solids, it is 50% of the value of **Min Size**.)
- When **Physics Preference** is set to **Mechanical** or **Explicit**, **Use Advanced Size Function** is set to **On: Curvature** for sheet models by default.
- The **Max Size** option (which was known as the **Max Tet Size** option in previous releases) is hidden if no solids are present in a model.

Extended Meshing

At release 13.0, the **Write ICEM CFD Files** control has been moved from the **Options** dialog box (**Tools > Options**) to the Details view of the **Uniform Quad**, **Uniform Quad/Tri**, **Patch Independent Tetra**, and **MultiZone** mesh methods. The control has been extended to include options for running ANSYS ICEM CFD interactively or in batch mode from an ANSYS ICEM CFD Replay file.

Large Scale Meshing

The following enhancements related to large scale meshing have been made at release 13.0:

- Speed improvements have been made across the board, but especially for surface and hex meshing.
- Better memory management for **Patch Conforming Tetra** has been implemented.
- The **Number of CPUs** option has been added to the **Options dialog box** in support of same machine parallel (SMP) meshing (for multiple cores; not supported for clusters). Using this option you can specify

a number of processors from 0 to 256. Specifying multiple processors will enhance the performance of the **Uniform Quad**, **Uniform Quad/Tri**, **Patch Independent Tetra**, and **MultiZone** mesh methods. This option has no effect when other mesh methods are being used.

Parameter Handling

In release 13.0, you can **parameterize** global and local mesh controls for use in the ANSYS Workbench Parameter Workspace.

Virtual Topology

New at release 13.0, you can use the **Virtual Split Edge** feature to split one edge into two virtual edges. You can define the location of the split either by picking the location in the **Geometry** window or by specifying a numerical value in the Details View. Using the F4 key, you can interactively adjust previously defined virtual edge splits.

Mixed Order Meshing

At release 13.0, **mixed order meshing** is now supported for the **Patch Independent Tetra**, **MultiZone**, **Uniform Quad/Tri**, and **Uniform Quad** mesh methods. To use mixed order meshing with these methods, all of the bodies in the part must be meshed with the same mesh method [that is, either all Patch Independent Tetra, all MultiZone, or all Uniform Quad(/Tri)].

Inflation Controls

The following inflation control enhancements have been made at release 13.0:

- Improved pre-inflation smoothing has led to less stair stepping and better quality during layer compression.
- For pre-inflation, new options are available for the **Inflation Option** control. The new options, called **First Aspect Ratio** and **Last Aspect Ratio**, allow you to control the heights of the inflation layers by defining the aspect ratio of the elements that are extruded from the inflation base.
- To simplify inflation control specification, you can define **inflation boundaries** via Named Selections (one or more for the faces in 3D models, or one or more for the edges in 2D or **Sweep** models).

Ease of Use Features

The following enhancements to ease of use features have been made at release 13.0:

- In the area of **ease of use features**, **Show** and **Preview** RMB menu options have been reorganized under new **Show** and **Preview** flyout menus. In addition, a new **Parts** flyout menu has been created to organize the **Generate Mesh**, **Preview Surface Mesh**, and **Clear Generated Data** menu options when they are selected via RMB on the **Geometry** window.
- A **Graphics Options toolbar** has been added to help diagnose potential problems with a geometry's topology and connectivity.
- To assist you in mapped face meshing, you can use the new **Show Mappable Faces** feature to highlight mappable faces prior to defining mapped face meshing controls, which provide more guidance to the mesher.
- You can use the new **Show Missing Tessellations** feature to highlight geometry with missing facets prior to generating the mesh. This feature is available only for the **CutCell** and **Patch Independent Tetra** mesh methods.

- You can **Clear Generated Data** on a selected part or body.

Size Function Improvements

The following improvements to size functions have been made at release 13.0:

- **Body of influence** now behaves as a soft setting instead of a hard setting, making it more useful for external aerodynamics problems.
- The **proximity size function** is not applied between faces that share an edge or between edges that share a vertex, reducing unnecessary refinement in **Patch Conforming Tetra** meshing.
- Transitioning between **swept** and **tetra** meshes with inflation has been improved.

Miscellaneous Changes and Behaviors

The following changes and behaviors are new at release 13.0:

- The **Max Tet Size** option, which was available in previous releases, has been renamed the **Max Size** option.
- You cannot apply a **match control** to topology on which a face-edge **pinch**, **mesh connection**, or **symmetry** control has been applied. In cases involving a match control and a pinch control, the match control will be suppressed and the reason (Overridden) will be reported in the **Active** read-only field in the Details view. In cases involving match with either mesh connection or symmetry, an error message will be issued.

2.6. 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:

Incompatibilities and Changes in Product Behavior from Previous Releases

Release 13.0 includes several new features and enhancements that result in product behaviors that differ from previous releases. These behavior changes are presented below.

- The effects of pressure load stiffness from a pre-stress analysis are evaluated at the analysis time at which the eigen analysis (**modal** or **buckling**) occurs. In previous releases, the effects of pressure load stiffness were based on the loading at time = 0.
- If a Display Time for a result is specified that is greater than the final time recorded in the result file, then Mechanical will post an error message and the result will not be evaluated. In previous releases, Mechanical would evaluate the result at the final time in the result file.

Similarly, if a set number is specified that is greater than the number of sets in the result file, then Mechanical will post an error message and the result will not be evaluated.

- The Update Stiffness contact region setting now includes the "Each Equilibrium Iteration, Aggressive" option and no longer includes the "Each Substep" option.
- A face-to-face contact using the MPC formulation will become underdefined if the contact is a solid body, the target is a surface body, and the behavior is set to Asymmetric.

Resuming Databases from Previous Releases

Note the following when resuming databases from previous releases:

- Pre-stress eigen analyses (either pre-stress modal or linear buckling) have been changed to use the Mechanical APDL technique of **linear perturbation**, as opposed to the PSTR command used in previous releases. Since the procedure and solution files are different, when a legacy project is opened that contains a solved static analysis with pre-stress effects, that analysis will need to be re-solved (in order to generate the new requisite files) if any additional eigen analyses are to be performed.
- Databases from previous releases that include a Sweep Mesh Control using gasket elements will no longer be supported. When resuming these databases, they will be marked as invalid and the user will need to create a new Gasket Mesh Control by changing the Stiffness Behavior of the body and recreating the geometry selections for the sweep direction.
- Connection objects from databases of previous releases that include contact regions or joints will be grouped based on their respective type and migrated into Connection Group folders (see **Connection Group** below under Connection Enhancements).

General Enhancements

The following general enhancements have been made at release 13.0:

- **Cyclic Symmetry.** Cyclic symmetry simulations are now available in static structural analyses, pre-stress modal analyses, and thermal analyses. Support for the following is available at release 13.0:
 - Cyclic controls.
 - Cyclic symmetry loads: [static structural analyses](#), [pre-stress modal analyses](#), [thermal analyses](#).
 - Cyclic symmetry results: [static structural analyses](#), [pre-stress modal analyses](#) (including a complete range of modes or a combination of degenerate modes), [thermal analyses](#).
 - [Cyclic Symmetry Modal Result Animations](#): Traveling waves and standing waves can be displayed as result animations.
- **Named Selections Based on Criteria.** Named Selections can now be based on criteria (size, location, geometry type) in addition to geometry selection. The ability to convert geometry types based on topology changes (for example, converting vertices “up” to edges, or bodies “down” to faces) is also available, as is the ability to create named selections that use criteria based on pre-selected geometry.

Analysis Enhancements

The following analysis enhancements have been made at release 13.0:

- **Harmonic Response Analysis Using Linked Modal Analysis System.** A harmonic response analysis using the Mode Superposition method can now be accomplished by linking a **Harmonic Response** analysis system to an existing **Modal** analysis system on the **Project Schematic**. In this way, multiple harmonic analyses with different loading conditions could effectively reuse the eigenvectors obtained in the modal analysis. Additional analysis settings have also been added to the [Output Controls](#) category to provide flexibility in controlling the downstream use of the modal superposition expansion.
- **Enhanced Pre-Stress Eigen Analysis.** The Mechanical APDL linear perturbation technique is now used for all pre-stress eigenvalue analyses. Support is available in this area for large deflection, cyclic analyses, true contact status, and multiple step static analyses.
- **Fluid Structure Interaction in Explicit Analysis.** In an Explicit Dynamics analysis that involves fluid structure interaction, there is now an option to represent volume bodies in an Eulerian reference frame. Bodies with reference frame set to Eulerian (Virtual) are mapped into a single structured hexahedral Eulerian mesh which by default encloses all bodies in the model. The Eulerian reference frame should be used when modelling fluids, gases or solids which may experience very large deformation during the simulation. Materials represented in the Euler domain will be automatically coupled to any Lagrangian

shell or solid bodies. This feature enables 2-way fluid structure interaction type problems to be solved in an Explicit Dynamics system. Note that the size and resolution of the Euler domain can be controlled in the analysis settings an Explicit Dynamics system. There is an optional graphic representation of the Euler domain for the geometry.

- **Explicit Dynamics Support for Linux.** Explicit Dynamics analyses are now supported on Linux platforms.
- **Design Assessment Analysis System.** The Design Assessment Analysis system enables the selection and combination of upstream results from Static Structural and Transient Structural analyses and the ability to optionally further assess results with customizable scripts (BEAMCHECK and FATJACK scripts are provided for Windows installations) and display the modified results. Furthermore, it enables the user to associate attributes, which may be geometry linked but not necessarily a property of the geometry, to the analysis via customizable items that can be added in the tree. Design Assessment is currently only released on Windows platforms; please contact ANSYS Technical Support if you wish to use it on Linux platforms.
- **Thermal-Stress Analysis Between Dissimilar Models.** Thermal-stress analyses can now be performed between thermal and structural analysis systems that use different meshes by mapping the temperatures across the two meshes.
- **Contact in Rigid Dynamics.** Contact can now be simulated in rigid dynamic analyses. Collisions between rigid bodies are detectable, even when the time step is large. You can also simulate sliding frictionless contact. Typical applications include cams and rollers.
- **Command Reference for Rigid Dynamics Systems.** Python command snippets are now available and they can be used in many ways. Examples include creating constraint equations between joint degrees of freedom, specifying a nonlinear stiffness for a spring, and using screw joints.

Geometry Enhancements

The following geometry enhancements have been made at release 13.0:

- **Specifying Variable Thickness on Surface Bodies.** You can now specify the thickness of selected faces on a surface body. Variable thickness can be specified through tabular or function input.
- **Part Associativity Maintained from DesignModeler Updates.** When geometry from DesignModeler is updated, any associativity applied in the Mechanical or Meshing application prior to the update is maintained, despite any part groupings that may have changed in DesignModeler.
- **Parts Compared in Update.** When geometry is updated, if no changes to the body are detected, the update can be configured such that a re-mesh of the body is not required.
- **Remote Points in Explicit Dynamics.** Remote Points can now be used in Explicit Dynamics analyses. Only rigid behavior is supported.

Graphics Enhancements

- **Enhanced Edge Visualization.** Options have been added to improve your ability to distinguish the edge connectivity in a surface body by inspecting geometry and meshes.
- **Highlighted Vertices.** A toolbar button is now available to highlight all vertices on a model.
- **Thicker Line Display for Annotations.** A toolbar button is now available to display thicker lines associated with annotations to make them more easily identifiable.
- **Interactive Editing of Virtual Edge Splits.** This feature allows an edge to be split into virtual edges as an aid in preparing geometry for meshing.

Connection Enhancements

The following connection enhancements have been made at release 13.0:

- **Mesh Connection.** The mesh connection feature allows manual or automatic joining of meshes of neighboring surface bodies in a multibody part that may not share topology.
- **Connection Group.** A new Connection Group tree object folder has been added to allow groupings of like connections and allows you to auto generate contact regions, mesh connections, or joints for a group of bodies in a model using a tolerance value that is unique to that group.
- **Stiffness and Damping Added to General Joint.** Worksheet entries for Stiffness and Damping coefficients are now available for general joints.
- **Line Body End Releases.** Edge interactions on line bodies can now have some degrees of freedom released between a vertex and an associated edge.
- **Tension/Compression Only Springs for Rigid Dynamics.** Springs can now be configured as tension-only or compression-only, in rigid dynamics analyses.
- **Result Tracker for Contact Area.** Contact area has been added as an output type available in the Contact result tracker.

Loads/Supports Enhancements

The following loads/supports enhancements have been made at release 13.0:

- **External Load Import.** Point-cloud data of temperature, pressure and convection coefficient from external files can be imported as loads in a static structural, transient structural, steady state thermal, transient thermal or thermal electric analysis.
- **Bolt Pretension for Line Bodies.** Bolt Pretension loads can now be applied to line bodies.
- **FSI - Volumetric Temperature Transfer.** This feature allows you to transfer domain temperatures from a CFD analysis and apply them as body loads in a structural analysis.
- **Ansoft - Mechanical Data Transfer.** Transfer of results between Ansoft applications (HFSS, Maxwell, or Q3D Extractor) and Mechanical can now be enabled by linking the systems in the project schematic.
- **Maxwell - Mechanical Stress Coupling .** Surface and body force density results from the Maxwell application can be imported and applied as loads in a structural analysis.
- **Detonation Point in Explicit Dynamics.** A Detonation Point load is now available in Explicit Dynamics analyses. This load generates a spherical detonation wave (shockfront) travelling radially outwards from the specified location and initiation time. The load will only affect materials containing the JWL equation of state property.
- **Remote Displacement in Explicit Dynamics.** Remote Displacements can now be used in Explicit Dynamics analyses.

Solution Enhancements

The following solution enhancements have been made at release 13.0:

- **Solution Restart.** Restart analysis and restart controls are now included in the analysis settings for Static Structural and Transient Structural analyses, that allow the analysis to be restarted under a variety of conditions. In addition to generating restart points, they can be managed in the Timeline and Tabular Data windows. Jobs can also be interrupted and restarted for local, RSM, and distributed solutions.
- **Gaskets.** Gasket simulations can now be performed in a static structural analysis.

- **Creep.** Analysis settings and results for simulating creep are now available.
- **Stabilization.** Stabilization controls and [Stabilization Energy](#) results are now available.
- **Multiple Restart Points in Static Structural Analyses for Pre-Stress Modal and Linear Buckling Analyses.** If a parent static structural analysis has multiple restart points at load steps/sub steps, the pre-stress modal or linear buckling analysis can start from any restart point available in the static structural analysis.
- **GPU Acceleration.** The Graphics Processing Unit (GPU) acceleration capability offered by Mechanical APDL is accessible in the Mechanical Application with support for NVIDIA acceleration cards.
- **Improvements for Explicit Dynamics Point Scoped Result Trackers.** The location of point scoped result trackers is now easier to define and they can be imported from a file.
- **Euler Body Result Trackers.** Most result trackers are available for Eulerian Bodies in Explicit Dynamics analyses.
- **Distributed Explicit Dynamics Solution.** Distributed solutions are now enabled for Explicit Dynamics (ANSYS) analyses.
- **Extended Support for RSM at Project Schematic.** RSM solutions can now be configured and initiated from the Project Schematic. At prior releases, only the default solution handler could be invoked from an Update action.

Results Enhancements

The following results enhancements have been made at release 13.0:

- **Results Scoped to Named Selections.** [Contour results](#) and [user defined results](#) can be scoped to named selections.
- **Beam (Line Body) Results.** Results in terms of axial force, bending moment, torsional moment, and shear force can now be applied to line bodies.
- **Shear-Moment Diagrams.** Diagrams are available for simultaneously illustrating line body results as the distribution of shear forces, bending moments and displacements, displayed as a function of arc length along a path consisting of line bodies. The path can be any contiguous line body edges.
- **Paths Scoped to Line Bodies.** Paths can now be scoped to line bodies as long as the path is defined by edge.
- **Peak Composite Results.** Result contours can now be displayed over an independent variable such as time in a static or transient structural analysis, or frequency/phase in a harmonic analysis, or cyclic phase in a [cyclic modal](#) analysis.
- **Worksheet View.** Custom variables are now available for Euler bodies to allow display of results associated with any single body, or all bodies, defined with an Eulerian reference frame.
- **Probe Result to Nearest Corner Node.** When picking a specific x, y, z location, a probe result can be applied directly to the closest corner node by using a new “Snap to mesh nodes” feature. The identification number of the closest corner node is also displayed as the **Node ID** in the Details view of the probe.
- **Defining a Path from Probe Labels.** When reviewing results, a path can be defined automatically from two probe labels.

Ease of Use Enhancements

The following ease of use enhancements have been made at release 13.0:

- **Additional “Go To” Options.** Options have been added to identify parts without contact in the tree as well as bodies with one element in at least two directions (through the thickness).

2.7. FE Modeler Release Notes

There are no changes in this release.

2.8. DesignXplorer Release Notes

The following general enhancements have been made at release 13.0:

Outline View

The Outline view has been enhanced. There are now state icons that show the status for each object in the outline. There are contextual menu entries, such as insert and delete, to manage objects. Parameters are now shown as a tree organized by each system on the desk top. This makes it easier to see where each parameter came from as well as which systems must be updated for each design point.

New DOE Types

Several New DOE types have been added. These include Box-Behnken, Custom + Sampling (which uses an OSF algorithm to enhance a custom DOE) and Sparse Grid, an adaptive DOE/Response surface method that features automatic adaptive refinement.

Sparse Grid

Sparse Grid requires a specific DOE as a starting point, this is labeled as Sparse Grid Initialization in the GUI and uses a Clenshaw Curtis formulation. While calculating the response surface, Sparse Grid adds design points to the DOE in order to resolve gradients where necessary. It is more efficient than other methods, particularly for large numbers of parameters.

Goodness of Fit

Goodness of fit has been reworked and extended. It is now an independent object with its own table of metrics and predicted vs observed chart. Verification points have also been added to check the goodness of Fit. Design candidates can easily be turned into verification points so that the “real solve” can be compared with the response point to determine the accuracy of the response surface. There is a new option to plot the design points on the response charts, these provide visual feedback and another way to check the goodness of fit.

Efficiency

Efficiency has been enhanced. An internal cache of design points are shared by all the DX systems in a workbench project in order to avoid recalculation between systems. These design points are stored as they are created and calculated design points can be recovered so that the series of simulations can resume after a power shutdown or hard crash.

Data Export

Data from any table or chart can be exported in CSV (formatted ascii text) file format for use outside of Workbench. It is also possible to import a DOE (in CSV format), with or without results. This could be used to import a particular DOE formulated elsewhere, or to use DX to post process experimental data.

Excel Interoperability

An Excel system is also available within the Workbench Component systems and can exchange parameters with DX and the Parameter set bar. Parameters can be flagged in Excel using the “Name a Range” method. In this way, Excel can be used as a solver within the workbench project. Users may want to optimize based on Excel calculated parameters such as cost, or perhaps the Excel system could be a reduce order model (ROM) coupled with parameters from other systems on the project page.

Correlation Samples

Correlation samples can now be previewed without a full update, similar to the DOE tables. Once started the correlation update can be stopped, partial results reviewed and then the correlation can be continued.

Parameters

It is possible to edit the initial value of a disabled parameter.

DX Help

DX Help has been enhanced and can be accessed with F1.

2.9. Engineering Data Workspace Release Notes

Material Enhancements

The following material models are now available for an implicit analysis:

- Gasket Model
- Gent hyperelastic model
- Blatz-Ko hyperelastic model
- Arruda-Boyce hyperelastic model
- Anand Viscoplasticity
- Chaboche Kinematic Hardening
- Bilinear Isotropic Hardening with temperature dependency
- Bilinear Kinematic Hardening with temperature dependency
- Response Function

The following material models are now available for an explicit analysis:

- Ideal Gas EOS
- JWL EOS

The following are now supported for temperature dependency:

- Hyperelastic models
- Experimental data
- Curve Fitting

The following creep models can be defined for use in a creep analysis:

- Strain Hardening
- Time Hardening
- Generalized Exponential
- Generalized Graham
- Generalized Blackburn
- Modified Time Hardening
- Modified Strain Hardening
- Generalized Garofalo
- Exponential Form
- Norton
- Combined Time Hardening
- Rational Polynomial
- Generalized Time Hardening

Additional sample material data has been included in the following libraries:

- Explicit materials
- Thermal materials

Ease of Use Enhancements

The **Outline Filter View** has been removed. To access **Favorites** and **Libraries** use the **Engineering Data Sources** button on the toolbar (set of books), to toggle the view on and off. You may also access this mode from the right mouse button menu.

2.10. EKM Desktop

EKM Desktop is new software for Release 13.0 and derives many of its features from two existing sources: the ANSYS EKM File Transfer Client (FTC) that was released with EKM 2.0, and ANSYS EKM Desktop 12.0 that was an add-in application to ANSYS Workbench 12.0 and 12.1 releases. Like EKM Desktop 12.0, you can use EKM Desktop 13.0 to manage and search personal simulation files on your local machine or a shared drive. Like FTC, you can use it to transfer files to and from a remote EKM server. EKM Desktop can be launched in a variety of ways: from ANSYS Workbench, a remote EKM server web user interface, Windows Explorer, and from a desktop shortcut.

2.10.1. Improved User Interface

The EKM Desktop user interface has been completely redesigned to improve usability and provide additional functionality. Changes to the user interface include the following:

- Closely matches the EKM Web user interface including access to most actions provided by EKM 13.0.
- Configurable views that allow parts of the user interface to be positioned, hidden, or shown in a way that makes the most sense to the user.
- Repositories tree view to simplify navigation of EKM repositories.
- Support for drag and drop to simplify transfers of files and folders to EKM repositories.

2.10.2. Supports Multiple Connections to EKM Repositories

EKM Desktop provides the capability to connect to, transfer, and manage data that are stored in a local EKM repository and in any number of connected repositories that are on EKM servers. For instance, you can upload files to a remote EKM repository in a workspace that is shared by your workgroup or enterprise, and download files from the shared repository to your local file system for your use or reuse. You can also create a local repository that resides on your local file system if you do not have a connection or access to a remote EKM server.

2.10.3. Improved Workbench Integration

Integration with ANSYS Workbench has been improved in Release 13.0. You can now easily transfer Workbench projects to and from a remote connected EKM server using new upload and download menus that are available in Workbench. Similarly, get and send changes features enable you to easily transfer only those parts of a Workbench project that have changed.

2.10.4. Improved File Capabilities

You can download and open a file in an associated application from an EKM server using EKM Desktop by simply double-clicking on the file (supported only on Windows operating systems).

2.10.5. Monitoring Transfers

EKM Desktop 13.0 allows you to easily view the status of all transfers to/from an EKM repository and the EKM cache server if your environment is configured to use it. Each transfer, whether it is queued, failed, or successful, is displayed in its own tab in the action status view. The same is true for each transfer made using the cache server. Additionally, you can create log files for failed and successful transfers.

2.10.6. Improved Search Capabilities

You can now search for data in multiple EKM repositories using improved search capabilities in EKM Desktop. As an added benefit, search results are shown in a separate view so that you can continue viewing other data in the repository while still having access to the search results.

2.10.7. EKM Studio Improvements

EKM Studio has been enhanced to provide improved usability for editing and viewing workflows. This includes the ability to edit and save workflow files to a remote EKM repository. Studio now also supports the ability to graphically edit EKM lifecycles.

Chapter 3: Mechanical APDL

This release of the Mechanical APDL 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:

- [Structural](#) (p. 25)
- [Low-Frequency Electromagnetics](#) (p. 32)
- [Acoustics](#) (p. 33)
- [Thermal](#) (p. 33)
- [Solvers](#) (p. 34)
- [Linear Perturbation](#) (p. 35)
- [APDL Math](#) (p. 35)
- [Other Enhancements](#) (p. 45)
- [Commands](#) (p. 35)
- [Elements](#) (p. 41)

Also see [Known Incompatibilities](#) (p. 46) and [The ANSYS Customer Portal](#) (p. 48) for important information about this release.

For information about changes to the ANSYS Workbench Products, see the ANSYS Workbench Products Release Notes.

3.1. Structural

Release 13.0 includes the following new features and enhancements for structural analyses:

- 3.1.1. Contact
- 3.1.2. Elements and Nonlinear Technology
- 3.1.3. Linear Dynamics
- 3.1.4. Materials and Fracture

3.1.1. Contact

Release 13.0 includes the following enhancements for structural analyses involving contact:

- 3.1.1.1. Surface-Projection-Based Contact
- 3.1.1.2. Geometry Correction for 3-D Contact and Target Surfaces
- 3.1.1.3. Multiple Load Step Interference Fit
- 3.1.1.4. Modeling Contact Offset (CNOF) as a Function of Location
- 3.1.1.5. Coefficient of Restitution
- 3.1.1.6. New Contact Element Output Quantities

3.1.1.1. Surface-Projection-Based Contact

Surface-projection-based contact is available for the 3-D surface-to-surface contact elements, [CONTA173](#) and [CONTA174](#). Surface projection based contact enforces contact constraints on an overlapping region of contact and target surfaces rather than on individual contact nodes or Gauss points. The advantages of using this technique are:

- It provides more accurate contact tractions and stresses of underlying elements.
- The designation of contact and target surfaces is less sensitive.
- It satisfies moment equilibrium and does not introduce artificial rotation energy, even when an offset in contact normal direction exists between contact and target surfaces with friction.
- Contact forces do not jump when contact nodes slide off the edge of target surfaces.

This contact method is implemented by setting `KEYOPT(4) = 3` on the [CONTA173](#) and [CONTA174](#) elements. For more information, see [Using the Surface Projection Based Contact Method \(KEYOPT\(4\) = 3\)](#) in the *Contact Technology Guide*.

3.1.1.2. Geometry Correction for 3-D Contact and Target Surfaces

In some contact applications, using a faceted surface in place of the true curved geometry can significantly affect the accuracy of contact stresses. An optional geometric correction feature has been introduced for nearly spherical and revolute (cylindrical) contact surfaces. Geometry correction, which is defined via `SECTYPE` and `SECDATA` section commands, is available for 3-D surface-to-surface contact elements: [TARGE170](#), [CONTA173](#), and [CONTA174](#). Applying geometry correction reduces the discretization error associated with faceted surfaces and can greatly improve the accuracy of contact stresses for certain types of curved contact/target surfaces. For more information, see [Geometry Correction for Contact and Target Surfaces](#) in the *Contact Technology Guide*.

3.1.1.3. Multiple Load Step Interference Fit

Previously, the ramping option for initial contact penetration (`KEYOPT(9) = 2` or `4`) was active only within the first load step. A new contact element real constant, `STRM`, allows you to define the load step number in which the ramping option will take place for a given contact pair. This real constant is useful for modeling multiple interference fits that occur sequentially (that is, the interference present in each contact pair can be resolved in different load steps). `STRM` is available for contact elements [CONTA171](#) through [CONTA177](#).

3.1.1.4. Modeling Contact Offset (CNOF) as a Function of Location

Contact offset `CNOF` can now be defined as a function of time and/or x , y , z location (in global or local coordinates) by using tabular input, allowing more flexibility for accurately modeling contact behavior. Consider the case of a CAD geometry based on nominal values. The geometry may lack a slight curvature variation that is important for analysis purposes. Moving nodes to the actual positions can be a tedious process, yet using the original geometry and neglecting the slight variation in curvature will result in a different contacting area. Consequently, inputting `CNOF` as a function of location allows you to easily include curvature that varies with location without having to modify the original CAD geometry. For more information, see [Adjusting Initial Contact Conditions](#) in the *Contact Technology Guide*.

3.1.1.5. Coefficient of Restitution

When using impact constraints to model impact between rigid bodies, a coefficient of restitution can now be input via the real constant `COR` to model loss of energy during impact. The coefficient of restitution defines the ratio of relative velocity of rigid bodies after impact to relative velocity of rigid bodies before

impact; its value varies between 0 and 1. A value of 0 indicates that the rigid bodies stick to each other after impact, while a value of 1 indicates that the rigid bodies rebound after impact with the magnitude of relative velocity after impact being the same as before impact. The new COR real constant is available for contact elements [CONTA171](#) through [CONTA178](#).

3.1.1.6. New Contact Element Output Quantities

The following new output quantities are available (via the **ETABLE** command) for contact elements [CONTA171](#) through [CONTA177](#): slip rate (VREL); fluid penetration starting time (FSTART); and true geometric gap/penetration at current converged substep (GGAP). In addition, a pair-based contacting area (summation of areas where contact is closed) can be reported through the **NLHIST** and **NLDIAG** commands.

3.1.2. Elements and Nonlinear Technology

Release 13.0 includes the following enhancements to elements and nonlinear technology:

- 3.1.2.1. New 2-D Reinforcing Element
- 3.1.2.2. General Axisymmetric Surface Element
- 3.1.2.3. Hydrostatic Fluid Elements
- 3.1.2.4. Preintegrated Composite Beam Sections
- 3.1.2.5. Enhanced Failure Criteria Support
- 3.1.2.6. Layer and Temperature Limits Lifted
- 3.1.2.7. Transverse-Shear Strain Formulation
- 3.1.2.8. Manual Rezoning Enhancements
- 3.1.2.9. Volumetric Force Density
- 3.1.2.10. Enhanced Ocean Loading

3.1.2.1. New 2-D Reinforcing Element

Use the new [REINF263](#) element with a standard 2-D solid or shell element (referred to as the *base element*) to provide extra reinforcing to that element. [REINF263](#) uses a smeared approach and is suitable for modeling evenly spaced reinforcing fibers that appear in layered form. Each reinforcing layer contains a cluster of fibers with unique orientation, material, and cross-section area, and is simplified as a homogenous membrane having unidirectional stiffness. You can specify multiple layers of reinforcing in one [REINF263](#) element. The nodal locations, degrees of freedom, and connectivity of the element are identical to those of the base element.

3.1.2.2. General Axisymmetric Surface Element

Use the new [SURF159](#) element to model axisymmetric solid surface loads acting on general axisymmetric solid ([SOLID272](#) or [SOLID273](#)) elements. The element has linear or quadratic displacement behavior on the **master plane** and is well suited to modeling irregular meshes on the master plane. It is defined by two or three nodes on the master plane, and nodes created automatically in the circumferential direction based on the master plane nodes. The total number of nodes depends on the number of nodal planes. Each node has three degrees of freedom: translations in the nodal x, y, and z directions. Various loads and surface effects can exist simultaneously.

3.1.2.3. Hydrostatic Fluid Elements

The new hydrostatic fluid elements, [HSFLD241](#) and [HSFLD242](#), allow you to model fluids that are fully enclosed by 2-D/axisymmetric solids, 3-D solids, or 3-D shells. The elements are well suited for calculating fluid volume and pressure for coupled problems involving fluid-solid interaction. The pressure in the fluid volume is assumed to be uniform (no pressure gradients), so sloshing effects cannot be included. Temperature effects and compressibility may be included, but fluid viscosity cannot be included. The elements have linear and

quadratic displacement behavior for nodes shared with the enclosing solid. A single pressure node with a hydrostatic pressure degree of freedom is shared by all the hydrostatic fluid elements defining a fluid volume. The elements can be used in static and transient dynamic analyses with various loads and boundary conditions.

3.1.2.4. Preintegrated Composite Beam Sections

A [preintegrated composite beam section](#) is an abstract cross-section type that allows you to define a fully populated but symmetrical cross-section stiffness and mass matrix directly. You can use preintegrated composite beam sections when using [BEAM188](#) or [BEAM189](#) elements, provided that linear elastic material behavior is acceptable. Four new commands ([CBMX](#), [CBMD](#), [CBTMP](#), and [CBTE](#)) are available for specifying the individual component quantities necessary for defining a preintegrated composite beam section. For more information, see [Using Preintegrated Composite Beam Sections](#) in the *Structural Analysis Guide*.

3.1.2.5. Enhanced Failure Criteria Support

Support has been added for Puck and Hashin fiber and matrix failure criteria. The new [FCTYP](#) command specifies which failure criteria are active for postprocessing and includes support for Puck and Hashin failure criteria. Output commands ([PxxSOL](#)), along with the [ETABLE](#) postprocessing command, have also been enhanced to support the new failure criteria.

A new data table ([TB,FCLI](#)) is available for defining material strength limits used to calculate failure criteria. For more information, see [Material Strength Limits \(TB,FCLI\)](#) in the *Element Reference*.

3.1.2.6. Layer and Temperature Limits Lifted

When layers are defined via cross sections, the number layers for [current-technology](#) shell, solid and elbow elements is no longer restricted to 250. Likewise, the number of temperatures that can be used with those elements is no longer limited to 1024. The changes apply to the following elements: [SHELL181](#), [SHELL281](#), [SHELL208](#), [SHELL209](#), [ELBOW290](#), [SOLID185](#), [SOLID186](#), [SOLSH190](#).

3.1.2.7. Transverse-Shear Strain Formulation

An enhanced transverse-shear strain formulation has been implemented in solid-shell element [SOLSH190](#). Previously, [SOLSH190](#) could predict only constant transverse-shear strains through the thickness. With the new formulation, [SOLSH190](#) is capable of parabolic transverse-shear distribution in the thickness direction, leading to significant improvement in particularly thick shell models.

3.1.2.8. Manual Rezoning Enhancements

Manual rezoning is now available for 3-D analyses. The remeshing method uses a generic new mesh (`.cdb` file) imported via a separate meshing step ([REMESH,READ](#)). Rezoning also supports additional solid and contact elements. For more information, see [Manual Rezoning](#) in the *Advanced Analysis Techniques Guide*.

3.1.2.9. Volumetric Force Density

Force density is now supported as a vector ([BFE,Element,FORCE](#)). The vector is interpreted in the global Cartesian coordinate system. Only constant values are valid (and not tabular loads).

The force density is distributed to elements nodes via the shape functions. Density values and directions remain unchanged as the element deforms; therefore, the total force varies as the element volume changes.

Force-density support is available in the following elements: [PLANE182](#), [PLANE183](#), [SOLID185](#), [SOLID186](#), [SOLID187](#), [SOLSH190](#), [PLANE223](#), [SOLID226](#), [SOLID227](#), and [SOLID285](#).

3.1.2.10. Enhanced Ocean Loading

Three new wave types (specified via *KWAVE* on the **OCDATA** command) are available for ocean loading: irregular wave, Shell new wave, and constrained new wave.

The irregular wave is created by adding the parameters (wave height, velocity, and acceleration) of a number of regular airy waves (wave components) with random phases and amplitudes corresponding to the required spectrum. The spectrum is divided into a number of equal energy strips based on the number of wave components specified, and each of the strips is a wave component. The frequency of the wave component is the frequency at the centroid of the strip. The amplitude of a wave component is given by the square root of twice the area of the strip.

The Shell new wave model is similar to irregular wave. It uses a statistically based linear superposition of linear wave components to define the wave profile and associated kinematics representing the most likely maximum condition of a real sea.

A constrained new wave embeds a Shell new wave into an irregular wave so that the maximum crest amplitude as given by the new wave occurs at a specific time and position while the statistical nature of the random sea is preserved.

For more information, see [Hydrodynamic Loads on Line Elements](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

3.1.3. Linear Dynamics

Release 13.0 includes the following enhancements in the area of linear dynamics:

- 3.1.3.1. Reusing Eigenmodes
- 3.1.3.2. Spectrum Reaction Forces and Multipoint Response Spectrum Enhancements
- 3.1.3.3. Enforced Motion Method for Mode Superposition Harmonic/Transient Analysis
- 3.1.3.4. Unsymmetric and Damped Extraction Methods
- 3.1.3.5. Solution Accuracy Improvement for Brake Squeal Analysis (QRDAMP Solver)
- 3.1.3.6. Harmonic Response Analysis
- 3.1.3.7. Spin Softening

3.1.3.1. Reusing Eigenmodes

Most computational resources in a modal analysis are dedicated to mode extraction. By reusing these results, fewer resources are required. All downstream analysis involving eigenmodes can now reuse the modes from the `Jobname.MODE` file in a subsequent spectrum analysis, modal transient or harmonic analysis, or QR damped complex-mode-extraction modal-based methods.

For more information, see [Reusing Eigenmodes](#) in the *Structural Analysis Guide*.

3.1.3.2. Spectrum Reaction Forces and Multipoint Response Spectrum Enhancements

The summation of the element nodal forces in a spectrum analysis now takes into account the forces signs. Applicable postprocessing commands are: **NFORCE**, **FSUM**, **PRRFOR**, and **PRNLD**.

To define the input spectrum in a multipoint response spectrum (MPRS) analysis, use the following new commands: **SPDAMP**, **SPFREQ**, **SPUNIT**, and **SPVAL**. The **SPDAMP** command allows you to input a damping ratio for each spectrum curve.

To display the input spectrum, issue the new **SPGRAPH** command.

In MPRS and power spectral density (PSD) analyses, you can now define the excitation direction in a global coordinate system via the **SED** command.

Spectrum analysis enhancements have also been made to [other commands](#).

3.1.3.3. Enforced Motion Method for Mode Superposition Harmonic/Transient Analysis

In mode superposition harmonic or transient analysis, the enforced motion method can be used when the excitations are caused by imposed motions (such as acceleration or displacement). For more information, see [Enforced Motion Method for Mode Superposition Transient and Harmonic Analysis](#) in the *Structural Analysis Guide* and [Enforced Motion in Structural Analysis](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

3.1.3.4. Unsymmetric and Damped Extraction Methods

The unsymmetric extraction method (**MODOPT,UNSYM**) is now applicable to non-damped models when the system matrices are unsymmetric, allowing a larger number of eigenvalues to be extracted in less time using an automated frequency shift strategy.

The unsymmetric and damped (**MODOPT,DAMP**) extraction methods are now supported using distributed memory parallelism in Distributed ANSYS.

For more information, see [Unsymmetric Method, Comparing Mode-Extraction Methods](#), and the **MODOPT** command documentation.

3.1.3.5. Solution Accuracy Improvement for Brake Squeal Analysis (QRDAMP Solver)

The [linear perturbation](#) process supports modal solutions for Block Lanczos, UNSYM, DAMP, and QRDAMP eigensolvers (**MODOPT**). The solution accuracy of QRDAMP for brake squeal analysis has been greatly improved when QRDAMP is used in conjunction with linear perturbation and the **CMROTATE** command.

3.1.3.6. Harmonic Response Analysis

The **HROPT** command now selects the most efficient method to solve an acoustic harmonic analysis by defaulting to AUTO (**HROPT,AUTO**). Depending on the model, either the full method or the Variational Technology (VT) method is selected. Using the VT method can reduce the time for an analysis by up to a factor of 10, especially if the number of harmonic solutions (specified with **NSUBST** command) is large.

For more information, see the **HROPT** command documentation, [Harmonic Response Analysis](#), and [Harmonic Sweep Using VT Accelerator](#).

This capability is also available for harmonic cyclic symmetry analyses. For more information, see the **HROPT** command documentation and [Cyclic Symmetry Analysis](#).

3.1.3.7. Spin Softening

Spin softening is now activated by default in prestressed modal analyses when any rotational velocities are specified. The *KSPIN* option on the **OMEGA** and **CMOMEGA** commands is no longer documented.

3.1.4. Materials and Fracture

Release 13.0 includes the following enhancements to materials and fracture technology:

3.1.4.1. Virtual Crack Closure Technique (VCCT)

3.1.4.2. Response Function Hyperelastic Material Option

3.1.4.3. Extended Tube Material Model

3.1.4.4. Gurson Plasticity with Isotropic/Chaboche Kinematic Hardening

3.1.4.5. Creep Enhancement

3.1.4.6. Cap Creep Model

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

3.1.4.1. Virtual Crack Closure Technique (VCCT)

Energy-release rates can now be calculated using VCCT technology for two-dimensional continuum elements, such as [PLANE182](#), and the 3-D continuum element [SOLID185](#). To specify the VCCT calculation type, use the enhanced **CINT** command. For more information, see [Fracture Mechanics Parameters](#) in the *Structural Analysis Guide*.

3.1.4.2. Response Function Hyperelastic Material Option

The new response function option for hyperelastic material constants (**TB,HYPER,,,,RESPONSE**) uses experimental data (**TB,EXPE**) to determine the constitutive response functions. The response functions (first derivatives of the hyperelastic potential) are used to determine the hyperelastic constitutive behavior of the material.

The response function hyperelastic option can include experimental data from uniaxial tension, uniaxial compression, equibiaxial, and/or planar shear tests. Additionally, the volumetric response can be specified via experimental pressure-volume data or a polynomial volumetric potential function.

For more information, see [Response Function Hyperelastic Option](#) in the *Structural Analysis Guide*, [Response Function Hyperelastic Material Constants](#) in the *Element Reference*, and [Experimental Response Functions](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

3.1.4.3. Extended Tube Material Model

The new extended tube model is available as a hyperelastic material option (**TB,HYPER,,,,ETUBE**). The model simulates filler-reinforced elastomers and other rubber-like materials, supports material curve-fitting, and is available in all [current-technology](#) continuum, shell, and pipe elements.

For more information, see the documentation for the **TB** command, [Extended Tube Material Constants](#) in the *Element Reference*, and [Extended Tube Model](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

3.1.4.4. Gurson Plasticity with Isotropic/Chaboche Kinematic Hardening

The new Gurson-Chaboche material model option is an extension of the [Gurson plasticity model](#). The option is used for modeling porous metal materials, combining both isotropic and kinematic hardening effects. It accounts for microscopic material behaviors, such as void dilatancy, void nucleation, and void coalescence into macroscopic plasticity models. Compared to the Gurson option with isotropic hardening only, the new option can provide more realistic deformation results for cyclic loading.

The Gurson-Chaboche model first requires the input parameters for Gurson plasticity with isotropic hardening (**TB,GURSON**), followed by additional input parameters for Chaboche kinematic hardening (**TB,CHABOCHE**).

For more information, see [Gurson-Chaboche Material Model](#) in the *Structural Analysis Guide*, and [Gurson Plasticity with Isotropic/Chaboche Kinematic Hardening](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

3.1.4.5. Creep Enhancement

When specifying [Newton-Raphson options](#) in an applicable static creep analysis, you now have the option of using modified Newton-Raphson with a creep-ratio limit in order to reduce the solution time. For more information, see the **NROPT** command description.

3.1.4.6. Cap Creep Model

The cap creep model is an extension of the [cap \(rate-independent plasticity\) model](#). The extension is based on creep theory similar to that of the [extended Drucker-Prager \(EDP\) creep model](#).

Unlike EDP which requires only one creep test measurement, a cap creep model requires *two* independent creep test measurements to account for both shear-dominated creep and compaction-dominated creep behaviors. The new **TBEO** command allows you to define both types of creep data separately.

For more information, see [Cap Creep Model](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

3.2. Coupled-Field

Release 13.0 includes the following enhancement in the area of coupled-field analysis:

3.2.1. Structural Material Nonlinearities

The coupled-field elements [PLANE223](#), [SOLID226](#), and [SOLID227](#) now support structural material nonlinearities. The following plasticity, viscoelasticity, and viscoplasticity/creep material properties are available with structural-thermo-electric analyses.

- Plasticity: PLASTIC, BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL, SMA, CAST, EDP, and GURSON
- Viscoelastic: PRONY and SHIFT
- Viscoplasticity/creep: CREEP and RATE

For more information, see [PLANE223](#), [SOLID226](#), and [SOLID227](#) in the *Element Reference*, the **TB** command in the *Command Reference* and [Structural-Thermal-Electric Analyses](#) in the *Coupled-Field Analysis Guide*.

3.3. Low-Frequency Electromagnetics

Release 13.0 includes the following enhancement in the area of low-frequency electromagnetics:

3.3.1. Stranded Coil Analysis

You can now use current-technology electromagnetic elements [PLANE233](#), [SOLID236](#), and [SOLID237](#) to perform stranded coil analyses. The elements are applicable to 2-D and 3-D static, time-harmonic, and time-transient electromagnetic analyses. The stranded coil analysis option is suitable for modeling a stranded winding with a prescribed current flow direction vector. The stranded coil may be voltage- or current-driven, as well as circuit-fed. The new formulation uses edge-flux (AZ), voltage drop across the coil (VOLT) and electromotive force (EMF) degrees of freedom. The elements have an option to perform a stranded coil analysis with time-integrated voltage drop or time-integrated electromotive force. For more information,

see the *Low-Frequency Electromagnetic Analysis Guide* and PLANE233, SOLID236, and SOLID237 in the *Element Reference*.

3.4. Acoustics

The following enhancements to acoustic analysis are available in this release:

- 3.4.1. New Acoustic Fluid Elements
- 3.4.2. Perfectly Matched Layers (PML)

3.4.1. New Acoustic Fluid Elements

The new 3-D 20-node acoustic fluid element, FLUID220, models the fluid medium and the interface in fluid/structure interaction problems. The element is well suited for modeling sound wave propagation and submerged structure dynamics. Also available is a 3-D 10-node acoustic fluid element, FLUID221.

3.4.2. Perfectly Matched Layers (PML)

PML support is now available in the FLUID30, FLUID220, and FLUID221 elements. PML can be specified (via KEYOPT(4)) to absorb the outgoing sound wave.

3.5. Thermal

The following enhancements to thermal analysis are available in this release:

- 3.5.1. New Thermal Solid Elements
- 3.5.2. Thermal Element Enhancement
- 3.5.3. Convection Analysis

3.5.1. New Thermal Solid Elements

Two new thermal elements, SOLID278 and SOLID279, are now available for 3-D steady state and transient analyses. SOLID278 is a 8-node brick element with 3-D thermal conduction capability. SOLID279 is a 20-node brick element that exhibits quadratic thermal conduction behavior. Two forms are available for these new elements: homogeneous (nonlayered) thermal solid and layered thermal solid. The layered solid form can model heat conduction in layered thick shells or solids. For thermal-structural analysis, SOLID278 and SOLID279 are designed to be companion elements for structural solid elements SOLID185 and SOLID186, respectively. For more information, see SOLID278 and SOLID279 in the *Element Reference*.

3.5.2. Thermal Element Enhancement

The 8-node thermal element PLANE77 now has a plane thickness option (KEYOPT(3)).

3.5.3. Convection Analysis

A two-extra-node option is now available for 2-D thermal surface effect element SURF151 (KEYOPT(5) = 2). This new option offers greater accuracy than the one-extra-node option. FLUID116 nodes can be mapped to SURF151 elements using the **MSTOLE** command.

The SURF151 and SURF152 thermal surface effect elements can now be used to define film effectiveness on a convection surface, providing more accurate simulations of film cooling.

For more information, see *Using the Surface Effect Elements* in the *Thermal Analysis Guide*.

3.6. Solvers

Release 13.0 includes the following new enhancements that improve solution procedures and features.

3.6.1. Distributed ANSYS Enhancements

3.6.2. GPU Accelerator Capability

3.6.3. Miscellaneous Solver Changes and Enhancements

3.6.1. Distributed ANSYS Enhancements

The following enhancements are available for Distributed ANSYS:

- Enhanced scalability is available in these areas:
 - Parallel equation ordering scheme is now default for the distributed sparse solver.
 - Improved scalability of assembly code for unsymmetric matrices.
 - Improved scalability during creation of the PCG preconditioner.
- The following features are now supported and use distributed memory parallelism within Distributed ANSYS:
 - **SURF151** and **SURF153** surface elements that use the element behavior of the underlying solid element (**KEYOPT(3) = 10**).
 - Modal analyses using the unsymmetric or damped solution (**MODOPT,UNSYM** or **MODOPT,DAMP**, respectively).
 - Multiple load vectors and enforced motion in a modal analysis (**MODCONT**).
 - Cyclic symmetry full harmonic analyses.
 - Tracking nodal and element solution data (**NLHIST,NSOL** and **NLHIST,ESOL**).
- Many additional features now work in Distributed ANSYS, but do not use distributed memory parallelism. For more information, see [Supported Analysis Types](#) and [Supported Features](#) in the *Distributed ANSYS Guide*.

3.6.2. GPU Accelerator Capability

It is becoming increasingly common to use the graphics processing unit (GPU) on certain high-end graphics cards to perform general-purpose computations. This capability, now available in Mechanical APDL, can accelerate portions of a simulation. Solution performance is especially improved, as the acceleration benefit applies primarily to the equation solvers, which use highly parallel, heavy number-crunching algorithms ideal for off-loading from the CPU onto the GPU.

GPU acceleration is supported on the Windows 64-bit and Linux x64 platforms. Currently, only a single GPU accelerator device can be used by the Mechanical APDL program during a solution. Only the NVIDIA Tesla GPU cards are supported, with the Tesla 20-series cards being recommended. The **ACCOPTION** command controls GPU acceleration options. GPU acceleration is not supported in Distributed ANSYS.

For more information, see "GPU Accelerator Capability" in the *Advanced Analysis Techniques Guide*.

3.6.3. Miscellaneous Solver Changes and Enhancements

The following are solver-related changes and enhancements:

- A major reduction in the amount of I/O for PCG Lanczos runs should improve performance.

- Improved initial shift logic for the Block Lanczos eigensolver when used with a buckling analysis.

3.7. Linear Perturbation

In many engineering applications, the linear behavior of a structure based on a prior linear or nonlinear preloaded status is of interest. You can use the new [linear perturbation analysis](#) procedure to solve a linear problem from this preloaded case for modal analyses. The preloaded case can have any nonlinear materials and geometric and contact nonlinearities. You can also use the linear perturbation procedure to perform a buckling analysis if the preloaded case is linear or in the case of linear with bonded contact.

To perform a modal linear perturbation after a static or full transient analysis, restart the analysis at the load point of interest and issue the commands **PERTURB**,**MODAL** and **SOLVE**. If a downstream analysis is desired using a modal load vector, you can define or modify the perturbation load for the downstream analysis. The perturbed load is calculated and stored in the `.FULL` and `.MODE` files for subsequent mode-superposition, PSD, or other type of modal-based linear dynamic analysis.

For more information, see [Linear Perturbation Analysis](#) in the *Structural Analysis Guide*, the **PERTURB** command documentation, and [Linear Perturbation Analysis Theory](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

This feature supersedes the partial-solution-based prestressed modal procedures (using the **PSOLVE** command).

3.8. APDL Math

The new APDL Math feature extends the APDL scripting environment of Mechanical APDL to give you access to the powerful matrix manipulation routines in the Mechanical APDL product, including its fast and efficient solvers.

APDL Math provides access to matrices and vectors from the `.FULL`, `.EMAT`, `.MODE` and `.SUB` files, as well as other sources, so that you can read them in, manipulate them, and write them back out or solve them directly. The new functionality augments the vector and matrix operations in the standard APDL scripting environment. Both dense matrices and sparse matrices can be manipulated using APDL Math.

For more information, see "APDL Math" in the *ANSYS Parametric Design Language Guide*.

3.9. Commands

This section describes changes to commands at Release 13.0.

Some commands are not accessible from menus. The documentation for each command indicates whether or not a menu path is available for that command operation. For a list of commands not available from within the GUI, see [Menu-Inaccessible Commands](#) in the *Command Reference*.

[3.9.1. New Commands](#)

[3.9.2. Modified Commands](#)

[3.9.3. Other Command Enhancements](#)

[3.9.4. Undocumented Commands](#)

[3.9.5. Archived Commands](#)

3.9.1. New Commands

The following new commands are available in this release:

- **ACCOPTION** -- Specifies GPU accelerator capability options.
- ***AXPY** -- Performs the matrix operation $M2 = v * M1 + w * M2$ (an [APDL Math](#) operation).
- **CBMD** -- Specifies the preintegrated mass-density matrix for [composite-beam sections](#).
- **CBMX** -- Specifies preintegrated cross-section stiffness for composite beam sections.
- **CBTE** -- Specifies a thermal expansion coefficient for a composite beam section.
- **CBTMP** -- Specifies a temperature for the composite beam matrix.
- **CNKMOD** -- Modifies contact element key options.
- ***COMP** -- Compresses the columns of a matrix using a specified algorithm (an [APDL Math](#) operation).
- ***DMAT** -- Creates a dense matrix (an [APDL Math](#) operation).
- ***EIGEN** -- Performs a modal solution with unsymmetric or damping matrices (an [APDL Math](#) operation).
- ***EXPORT** -- Exports a matrix to a file in the specified format (an [APDL Math](#) operation).
- **FCTYP** -- Activates or removes failure-criteria types for postprocessing.
- ***FREE** -- Deletes a matrix or a solver object and frees its memory allocation (an [APDL Math](#) operation).
- ***ITENGINE** -- Performs a solution using an iterative solver (an [APDL Math](#) operation).
- ***LSBAC** -- Performs the solve (forward/backward substitution) of a factorized linear system (an [APDL Math](#) operation).
- ***LSENGINE** -- Creates a linear solver engine (an [APDL Math](#) operation).
- ***LSFACTOR** -- Performs the numerical factorization of a linear solver system (an [APDL Math](#) operation).
- ***MULT** -- Performs the matrix multiplication $M3 = M1(T1) * M2(T2)$ (an [APDL Math](#) operation).
- ***NRM** -- Computes the norm of the specified matrix or vector (an [APDL Math](#) operation).
- **PERTURB** -- Sets [linear perturbation](#) analysis options.
- ***PRINT** -- Prints the matrix values to a file (an [APDL Math](#) operation).
- **QRDOPT** -- Specifies additional QRDAMP modal analysis option.
- ***SMAT** -- Creates a sparse matrix (an [APDL Math](#) operation).
- **SPDAMP** -- Defines input spectrum damping in a multipoint response spectrum analysis.
- **SPFREQ** -- Defines the frequency points for the input spectrum tables **SPVAL** vs. **SPFREQ** for multipoint spectrum analysis.
- **SPGRAPH** -- Displays input spectrum curves for MPRS analysis.
- **SPVAL** -- Defines multipoint response spectrum values.
- **SPUNIT** -- Defines the type of multipoint response spectrum.
- **TBEO** -- Sets special options or parameters for material data tables. For example, a [cap creep model](#) requires two independent creep test measurements to account for both shear-dominated creep and compaction-dominated creep behaviors. The new command allows you to define both types of creep data separately.
- ***VEC** -- Creates a vector (an [APDL Math](#) operation).

3.9.2. Modified Commands

The following commands have been enhanced in this release:

- **ANTYPE** -- Specifies the analysis type and restart status. For modal analysis, the REST option allows you to reuse the existing modes extracted from a previous modal analysis. In addition, the new PERTURB option specifies that a [linear perturbation](#) analysis should be performed at restart.
- **BF** -- Defines an element body force load. The command has a new structural label (*Lab* = FORC) for body-force density in a momentum equation.
- **CMOMEGA** -- Specifies the rotational velocity of an element component about a user-defined rotational axis. Spin softening is now activated by default in prestressed modal analyses when any rotational velocities are specified. The *KSPIN* option is no longer documented.
- **D** -- Defines degree-of-freedom constraints at nodes. This command can now be used to prescribe an HDSP degree-of-freedom constraint at the pressure node of hydrostatic fluid elements, [HSFLD241](#) and [HSFLD242](#).
- **EQSLV** -- Specifies the type of equation solver. The automatic iterative solver option (*Lab* = ITER) is no longer supported. In addition, the DSPARSE solver option has been replaced with the SPARSE solver option. When the SPARSE solver option is specified, the distributed sparse solver is used in Distributed ANSYS, and the non-distributed sparse solver is used in SMP ANSYS.
- **ESURF** -- Generates elements overlaid on the free faces of existing selected elements. The *Shape* = TRI option has been removed. The command no longer generates triangular facet target elements using [TARGE170](#).

In addition, the **ESURF** command can now be used to generate hydrostatic fluid elements, [HSFLD241](#) and [HSFLD242](#). The command generates triangular (2-D) or pyramid-shaped (3-D) elements with bases that are overlaid on the faces of selected 2-D or 3-D solid or shell elements.

- **ETABLE** -- Fills a table of element values for further processing. This postprocessing command now supports failure criteria. Also, the command now has the new label SEDN for strain energy density
- **ETCHG** -- Changes element types to their corresponding types. For thermal-structural analyses, this command can change the new thermal elements [SOLID278](#) and [SOLID279](#) to their companion structural elements [SOLID185](#) and [SOLID186](#), respectively.
- **F** -- Specifies force loads at nodes. The DVOL force label has been added to specify fluid mass flow rate (with units of mass/time) at the pressure node of hydrostatic fluid elements, [HSFLD241](#) and [HSFLD242](#).
- ***GET** -- Retrieves a value and stores it as a scalar parameter or part of an array parameter. The command can now retrieve [composite beam-section](#) data (*Entity* = CMPB).
- **HROPT** -- Specifies harmonic analysis options. The command now has a *Method* = AUTO option, which automatically select the most efficient method. This is the new default method.
- **IC** -- Specifies initial conditions at nodes. The HDSP (hydrostatic pressure) degree-of-freedom has been added as a new structural label to allow specification of initial hydrostatic pressure at the pressure node of hydrostatic fluid elements, [HSFLD241](#) and [HSFLD242](#).
- **IOPTN** -- Controls options relating to importing a model. For IGES import, the FACETED (RV53) option is no longer available.
- **LCOPER** -- Performs load case operations. Use the new *Oper2* = CPXMAX to calculate the maximum of derived stresses of complex results.
- **MODOPT** -- Specifies modal analysis options. This command now features the *QrdReuse* field, which provides the option to reuse the block Lanczos eigenvectors from the first load step.
- ***MOPER** -- Performs matrix operations on array parameter matrices. Additional options have been added for sorting arrays by their columns.
- **MSTOLE** -- Adds two extra nodes from [FLUID116](#) elements to [SURF151](#) or [SURF152](#) elements for convection analyses. This command now supports [SURF151](#) elements.

- **NLDIAG** -- Sets nonlinear diagnostics functionality. This command now reports contacting area in the `Jobname.CND` file when contact information is requested.
- **NLHIST** -- Specify result items to track during solution. Contacting area can now be requested as an item to track in the `Jobname.NLH` file.
- **NROPT** -- Specifies Newton-Raphson options in a static or full transient analysis. When applicable in a static creep analysis, the command now supports the modified Newton-Raphson option with a creep-ratio limit.
- **OCTYPE** -- Defines an ocean environment using non-table data. When the specified ocean data type is "wave" (**OCTYPE**,,WAVE), valid wave-theory values for specifying the API (for computing particle velocities and accelerations due to waves and current) have been changed from 10+ to 101+. For further information, see *Known Incompatibilities* (p. 46).
- **OMEGA** -- Specifies the rotational velocity of the structure. Spin softening is now activated by default in prestressed modal analyses when any rotational velocities are specified. The *KSPIN* option is no longer documented.
- **PLESOL, PLNSOL, PRESOL, PRNSOL** -- These output commands have new item and component labels for Puck and Hashin failure criteria.
- **PSDFRQ** -- Defines the frequency points for the input spectrum tables PSDVAL vs. PSDFRQ for PSD analyses. No longer applies to multipoint spectrum analyses. (See **SPFREQ**.)
- **PSDUNIT** -- Defines the type of input PSD. No longer defines the type of multipoint response spectrum. See **SPUNIT**.
- **PSDVAL** -- Defines PSD spectrum values. No longer defines multipoint response spectrum values. See **SPVAL**.
- **PREENERGY** -- Lists the energies of the entire model or the energies of the specified components. Formerly, the command printed the total energies of a model.
- **RESCONTROL** -- Controls file writing for multiframe restarts. The new **LINEAR** option on the *Action* field specifies writing of restart files during a linear static analysis. This option is needed if a subsequent **linear perturbation** analysis is anticipated.
- **RSTMAC** -- Calculates modal assurance criterion (MAC) and matches nodal solutions from two results files. You can now specify *TolerN* = -1 to map the nodes of `File2` into the elements of `File1`. The MAC is then based on the interpolated solutions from `File1`. This procedure is particularly useful when comparing the modes of cyclic symmetric structures.
- **SECDATA** -- Describes the geometry of a section. This command now supports the definition of contact sections for spherical or revolution surfaces associated with certain contact/target elements.
- **SECTYPE** -- Associates section type information with a section ID number. The command now has support for **composite (temperature-dependent) beam sections** and contact sections.
- **SED** -- Defines the excitation direction for response spectrum and PSD analyses. This command now contains the *Cname* field, which stores the component name corresponding to the group of excited nodes. This new field allows the definition of the excitation in the global coordinate system for MPRS or PSD analyses.
- **SFE** -- Specifies surface loads on elements. For the **SURF151** and **SURF152** extra node option (**KEYOPT**(5) = 1), film effectiveness and free stream temperatures may now be input for convection surface loads.
- **SFELIST** -- Lists the surface loads for elements. Film effectiveness and free stream temperatures specified by the **SFE** command can now be listed by this command.

- **SOLVE** -- Starts a solution. A new *Action* field is available for use in [linear perturbation](#) analyses. Setting *Action* = ELFORM causes all appropriate element matrices to be reformed in the first phase of a [linear perturbation](#) analysis.
- **SPOPT** -- Selects the spectrum type and other spectrum options. This command now contains the *modReuseKey* field. When running multiple spectrum analyses, this new key specifies that the existing MODE file is updated for reuse.
- **SRSS** -- Specifies the square root of sum of squares mode combination method. This command now contains the *AbsSumKey* value, which activates the Absolute Sum combination method. This method first combines the modes for each excitation direction. It is supported in MPRS analysis.
- **TB** -- Activates a data table for nonlinear material properties or special element input.

A new response function option for hyperelastic material constants (**TB,HYPER,,,,RESPONSE**) has been added. The option uses experimental data (input via the new **TB,EXPE** command) to determine the constitutive response functions. The response functions are used to determine the hyperelastic constitutive behavior of the material.

A new fluid material model (**TB,FLUID**) has been added for hydrostatic fluid elements, [HSFLD241](#) and [HSFLD242](#). Additional input allows you to define the material as a liquid, a gas, or a fluid represented by pressure-volume data.

The coupled-field elements [PLANE223](#), [SOLID226](#), and [SOLID227](#) can now use the **TB** table to specify nonlinear material properties. Plasticity and viscoplasticity/creep properties are now available.

- **TBFT** -- Performs material curve-fitting operations. The plot functionality (**TBFT,PLOT**) has been removed. Plotting for curve-fitting operations should be performed via the graphical user interface (GUI).
- ***VOPER** -- Operates on two array parameters. Options have been added for performing vector or tensor (such as stress) transformations to and from the global Cartesian coordinate system to a local coordinate system.

3.9.3. Other Command Enhancements

The absolute values of real data entered in a command can now be between $1.0E^{-200}$ and $1.0E^{+200}$. In prior releases, the absolute value was limited to an exponent of +/-60.

The environment variable ANSYS_MACROLIB, which specifies the directories to search for user-supplied macros, has been extended from 255 characters to 2480 characters.

3.9.4. Undocumented Commands

The following features have been undocumented at this release:

- The faceted geometry option for IGES import (**IOPTN,FACET**) and its defeaturing commands
- The p-method
- The run statistics processor /RUNSTAT

The following legacy commands have therefore been undocumented:

- **ALPFILL**
- **ARCOLLAPSE**
- **ARDETACH**

- **ARFILL**
- **ARMERGE**
- **ARSPLIT**
- **GAPFINISH**
- **GAPLIST**
- **GAPMERGE**
- **GAPOPT**
- **GAPPLOT**
- **LNCOLLAPSE**
- **LNDETACH**
- **LNFill**
- **LNMERGE**
- **LNSPLIT**
- **PCONV**
- **PLCONV**
- **PEMOPTS**
- **PEXCLUDE**
- **PINCLUDE**
- **PMETH**
- **/PMETH**
- **PMOPTS**
- **P PLOT**
- **PPRANGE**
- **PRCONV**
- **PRECISION**
- **RALL**
- **RFILSZ**
- **RITER**
- **RMEMRY**
- **RSPEED**
- **RSTAT**
- **RTIMST**
- **/RUNST**
- **RWFRNT**
- **SAR PLOT**
- **SHSD**
- **SLP PLOT**

- **SLSPLOT**
- **VCVFILL**

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

3.9.5. Archived Commands

The piping module is now archived. The following legacy commands have therefore been moved to the *Feature Archive*:

- **BELLOW**
- **BEND**
- **BRANCH**
- **FLANGE**
- **MITER**
- **PCORRO**
- **PDRAG**
- **PFLUID**
- **PGAP**
- **PINSUL**
- **PIPE**
- **POPT**
- **PPRES**
- **PSPEC**
- **PSPRNG**
- **PTEMP**
- **PUNIT**
- **REDUCE**
- **RUN**
- **TEE**
- **VALVE**

3.10. Elements

This section describes changes to elements at Release 13.0.

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

[3.10.1. New Elements](#)

[3.10.2. Modified Elements](#)

[3.10.3. Undocumented Elements](#)

[3.10.4. Archived Elements](#)

3.10.1. New Elements

The following new elements are available in this release:

- **SURF159** -- This element models axisymmetric solid surface loads acting on general axisymmetric solid (**SOLID272** or **SOLID273**) elements. The element has quadratic displacement behavior on the master plane and is well suited to modeling irregular meshes on the master plane. It is defined by two or three nodes on the master plane, and nodes created automatically in the circumferential direction (based on the master plane nodes).
- **FLUID220** -- This 3-D 20-node acoustic fluid element models the fluid medium and the interface in fluid/structure interaction problems. This element is well suited for modeling sound wave propagation and submerged structure dynamics.
- **FLUID221** -- This 3-D 10-node acoustic fluid element models the fluid medium and the interface in fluid/structure interaction problems. This element is well suited for modeling sound wave propagation and submerged structure dynamics.
- **HSFLD241** -- This 2-D hydrostatic fluid element models fluids that are fully enclosed by 2-D planar and axisymmetric solids. This element is well suited for modeling fluid-solid interaction with incompressible or compressible fluids under uniform pressure. It can be used in geometrically linear as well as nonlinear static and transient dynamic analyses.
- **HSFLD242** -- This 3-D hydrostatic fluid element models fluids that are fully enclosed by 3-D solids or shells. This element is well suited for modeling fluid-solid interaction with incompressible or compressible fluids under uniform pressure. It can be used in geometrically linear as well as nonlinear static and transient dynamic analyses.
- **REINF263** -- This 2-D reinforcing element is used with a standard 2-D solid or shell element (referred to as the *base element*) to provide extra reinforcing to that element. It uses a smeared approach and is suitable for modeling evenly spaced reinforcing fibers that appear in layered form.
- **SOLID278** -- This 3-D 8-node thermal solid element is applicable to steady state and transient analyses. The element has two forms: homogeneous thermal solid and layered thermal solid. It is designed to be a companion element for structural solid element **SOLID185**.
- **SOLID279** -- This 3-D 20-node thermal solid element is applicable to steady state and transient analyses. The element has two forms: homogeneous thermal solid and layered thermal solid. It is designed to be a companion element for structural solid element **SOLID186**.

3.10.2. Modified Elements

The following elements have been enhanced in this release:

- **PLANE77** -- This element now has a plane thickness option (KEYOPT(3)).
- **FLUID116** -- This coupled thermal-fluid pipe element has a new KEYOPT(1) = 3 option to specify the PRES degree of freedom when it is connected to a hydrostatic fluid element (**HSFLD241** or **HSFLD242**). This option converts the fluid element mass flow rate to volume change for compatibility with the new hydrostatic fluid elements.
- **SURF151** -- For improved accuracy in convection analyses, this 2-D surface effect element has a new option for adding *two* extra nodes from **FLUID116** elements. For the one-extra-node option (KEYOPT(5) = 1), film effectiveness and free stream temperatures may now be input for convection surface loads.
- **SURF152** -- For the one-extra-node option (KEYOPT(5) = 1), film effectiveness and free stream temperatures may now be input for convection surface loads.

- [TARGE170](#), [CONTA173](#), and [CONTA174](#) -- These 3-D surface-to-surface contact and target elements now support a geometry correction feature that can be applied to spherical and revolute contact and target surfaces to reduce discretization errors associated with faceted surfaces.
- [CONTA173](#) and [CONTA174](#) -- These 3-D surface-to-surface contact elements support the new projection-based method specified by setting KEYOPT(4) = 3 for the contact detection option.
- [CONTA171](#) through [CONTA177](#) -- The following improvements are available for these contact elements:
 - KEYOPT(10), which controls the contact stiffness update method, has been simplified in these elements. Several of the options have been removed from this KEYOPT; KEYOPT(10) = 0 and 2 are still available.
 - A new real constant, STRM, allows you to specify the load step number in which the ramping option for initial contact penetration will take place. Used in conjunction with KEYOPT(9) = 2 or 4, this feature is useful for modeling multiple interference fits that take place sequentially over several load steps.
 - The following new contact output quantities are available: VREL -- slip rate; GGAP -- true geometric gap/penetration at current converged substep; FSTART -- fluid penetration starting time. (FSTART is available only for surface-to-surface contact elements.)
- [CONTA171](#) through [CONTA178](#) -- You can now input a coefficient of restitution via the new contact element real constant COR. When using impact constraints to model impact between rigid bodies, the coefficient of restitution can be used to model loss of energy during impact.
- [SHELL181](#) -- KEYOPT(4) has been removed from this four-node structural shell element. The element now uses the constitutive algorithm exclusively for nonlinear shell-thickness updates. Real constant support has been undocumented.
- [SHELL208](#), [SHELL209](#) -- KEYOPT(4) has been removed from these shell elements. The elements now use the constitutive algorithm exclusively for nonlinear shell-thickness updates.
- [PLANE223](#), [SOLID226](#), and [SOLID227](#) -- These coupled field elements have been enhanced with new nonlinear material capabilities. Plasticity, viscoelasticity, and viscoplasticity/creep material properties can now be specified via the **TB** command.
- [PLANE233](#), [SOLID236](#), [SOLID237](#) -- These current-technology elements now support stranded coil analyses via the new KEYOPT(1) = 2 option. The stranded coil analysis option is suitable for modeling a stranded winding with a prescribed current flow direction vector. The stranded coil may be voltage- or current-driven, as well as circuit-fed.
- [SHELL281](#) -- KEYOPT(2) and KEYOPT(4) have been removed from this eight-node structural shell element. The element now uses an advanced shell formulation that accurately incorporates initial curvature effects. The new formulation generally offers improved accuracy in curved shell structure simulations, especially when thickness strain is significant or the material anisotropy in the thickness direction cannot be ignored. Real constant support has been undocumented.

3.10.3. Undocumented Elements

The following legacy elements have been undocumented at this release, as follows:

Undocumented Legacy Element	Suggested Current-Technology Element	Recommendations
BEAM3	BEAM188 or BEAM189	Set KEYOPT(3) = 3. Constrain UZ, ROTX, and ROTY to simulate 2-D behavior. Issue a SECTYPE,,BEAM command.

BEAM23		Set KEYOPT(3) = 3. Constrain UZ, ROTX, and ROTY to simulate 2-D behavior. Issue a SECTYPE „BEAM command.
BEAM24		Set KEYOPT(3) = 3. Issue a SECTYPE „BEAM command.
BEAM44		Set KEYOPT(3) = 3. Issue a SECTYPE „BEAM or possibly a SECTYPE „TAPER command. A SECOFFSET command may be necessary.
BEAM54		Set KEYOPT(3) = 3. Constrain UZ, ROTX, and ROTY to simulate 2-D behavior. Issue a SECTYPE „TAPER command. A SECOFFSET command may be necessary.
COMBIN7	MPC184	Set KEYOPT(1) = 6.
LINK1		--
LINK8	LINK180	--
LINK10		To simulate LINK10 functionality, set the LINK180 tension/compression option (real constant <i>TENSKEY</i>).
LINK32	LINK33	--
PIPE17		--
PIPE20	PIPE288	--
PIPE60	ELBOW290	--
PLANE67	PLANE223	Set KEYOPT(1) = 110.
SHELL57	SHELL131	Set KEYOPT(3) = 2. Issue SECTYPE „SHELL.
SOLID69	SOLID226	Set KEYOPT(1) = 110.
p-elements: SOLID127 SOLID128 PLANE145 PLANE146 SOLID147 SOLID148 SHELL150	NA	The p-method has been undocumented.

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

3.10.4. Archived Elements

The following legacy elements have been moved to the *Feature Archive*:

Archived Legacy Element	Suggested Current-Technology Element	Recommendations
BEAM4	BEAM188 or BEAM189	Set KEYOPT(3) = 3. Issue a SECTYPE „BEAM command.

CONTAC12	CONTA178	Constrain the UZ degree of freedom to simulate 2-D behavior. CONTA178 does not support the circular gap option of CONTAC12.
PIPE16	PIPE288	--
PIPE18	ELBOW290	--
PLANE42	PLANE182	Set KEYOPT(1) = 3.
SOLID45	SOLID185	Set KEYOPT(2) = 3.
CONTAC52	CONTA178	--
PIPE59	PIPE288	Issue SOCEAN and ocean (OCxxxxxx) commands to apply ocean loading.
SHELL63	SHELL181	Set KEYOPT(3) = 2. May require a finer mesh.
PLANE82	PLANE183	--
SOLID92	SOLID187	--
SOLID95	SOLID186 (Homogenous Structural Solid)	Set KEYOPT(2) = 1. For nonlinear analysis, set KEYOPT(2) = 0 (default).

3.11. Other Enhancements

This section contains information about enhancements not listed elsewhere in this document.

3.11.1. Postprocessing

The following enhancements have been made to the POST1 general database results postprocessor.

- **Energy** -- The energy values can now be printed out for each of the components in a model via the **PREENERGY** command.
- **Maximum of derived stresses for complex results** -- When postprocessing complex results, the maximum of the derived stresses can be obtained via the **LCOPER** load-case operations command.
- **Modal Assurance Criterion (MAC)** -- The **RSTMAC** command now supports dissimilar meshes using mapping and interpolation of the solutions.

3.11.2. Documentation

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

3.11.2.1. Technology Demonstration Guide

The *Technology Demonstration Guide* includes a new example problem entitled [Modal and Harmonic Frequency Analyses of an Automotive Suspension Assembly Using CMS](#). The new example shows how to generate dynamic superelements for use in downstream linear dynamics analyses using [component mode synthesis \(CMS\)](#).

The example [cyclic symmetry centrifugal impeller analysis](#) has been enhanced to include [linear-perturbation solution approaches](#). For more information, see [Centrifugal Impeller Analysis Using Cyclic Symmetry and Linear Perturbation](#).

3.11.2.2. Feature Archive

This release includes the new *Feature Archive*, a repository for legacy element, command, theory and feature documentation. While ANSYS, Inc. continues to support these legacy capabilities for the immediate future, some may be undocumented in future releases. You are urged to consider moving to their recommended replacements.

3.11.2.3. Documentation Updates for Programmers

The following documentation updates are available for programmers:

3.11.2.3.1. Routines and Functions Updated

Routines and functions documented in the *Programmer's Manual* 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.

3.11.2.3.2. /UPF Command for Linking UPFs

The new **/UPF** command offers the simplest method for linking user-programmable features into Mechanical APDL. The format of the command is:

```
/UPF,RoutineName
```

where *RoutineName* is the name of a user routine that you want to link. The specified routine must reside in the current working directory.

When the Mechanical APDL program reads the input and detects this command, the program is relinked automatically. You can reissue the **/UPF** command as often as needed to include multiple user routines. For more information, see [Compiling and Linking UPFs on UNIX/Linux Systems](#) and [Compiling and Linking UPFs on Windows Systems](#) in the *Programmer's Manual*.

3.11.2.3.3. New Routines for Ocean Loading

The *Programmer's Manual* features two new user routines to support analyses involving ocean loading. The `userPanelHydFor` routine computes panel loads caused by ocean loading. This capability is accessed via the SURF154 element's KEYOPT(8), together with data read in via the `userOceanRead` subroutine.

3.12. Known Incompatibilities

The following incompatibilities with prior releases of are known to exist at Release 13.0.

- 3.12.1. Surface Elements
- 3.12.2. Change in Default Byte-Swapping Behavior for Binary Files
- 3.12.3. Results File Format Change
- 3.12.4. Spin Softening Default
- 3.12.5. Ocean Environment Definition
- 3.12.6. Rate-Dependent Plastic (Viscoplastic) Material Model Option
- 3.12.7. Lumped Matrix Formulation with Beam, Pipe, or Shell Elements
- 3.12.8. Contacting Area for Contact Elements

3.12.1. Surface Elements

For [SURF151](#), [SURF152](#), [SURF153](#), and [SURF154](#), the seventh real constant (TKI, the thickness at node I) takes a value of 0.0 rather than the former value of 1.0 if that constant is input as zero or blank. The new default value may affect the volume and mass of the indicated surface elements.

For [SURF151](#) and [SURF152](#), the change may also affect the element heat generation and specific heat logic. For [SURF153](#) and [SURF154](#), the change may also affect the surface-tension logic.

3.12.2. Change in Default Byte-Swapping Behavior for Binary Files

In previous releases, Mechanical APDL performed byte-swapping on most Windows and Linux platforms before writing any file data or after reading any file data; byte-swapping was not performed on most UNIX platforms.

To improve performance, byte-swapping *is no longer performed* on Windows and Linux platforms at this release, but *is performed* on most UNIX platforms.

This incompatibility affects only third parties who interface with ANSYS binary files (that is, read/write ANSYS binary files in their own programs). Affected third parties must modify their code to accommodate this change.

BINLIB and the demonstration routines provided with the release (documented in the [Programmer's Manual](#)) have been upgraded to reflect this change, and can read either format.

3.12.3. Results File Format Change

The results file has the following new or changed records (as documented in the [Programmer's Manual](#)):

- Records for material property data have been added.
- Records for section data have been added.
- The degree-of-freedom records for reaction forces and master degrees-of-freedoms are LONGINT (64-bit) instead of integers (32-bit).

The results file access routines provided with the release (also documented in the [Programmer's Manual](#)) have been upgraded to reflect this change, and can read current results files as well as files from prior releases.

3.12.4. Spin Softening Default

Spin softening is now **activated by default** in prestressed modal analyses when any rotational velocities are specified. If you did not explicitly request it in prior releases, you may notice different frequencies.

3.12.5. Ocean Environment Definition

Defining an ocean environment using non-table data has changed slightly when the specified ocean data type is "wave" (**OCTYPE**,WAVE). Valid wave-theory values for specifying the API (for computing particle velocities and accelerations due to waves and current) have been changed from 10+ to 101+, as follows:

- 101 through 200 -- Data preprocessed (via the default Small Amplitude Airy Wave Theory logic [*KWAVE* = 0]).
- 201+ -- Data not preprocessed.

3.12.6. Rate-Dependent Plastic (Viscoplastic) Material Model Option

An option (*TBOPT*) on the **TB,RATE** command has changed. The former *TBOPT* = CHABOCHE option is now *TBOPT* = EVH. The option offers exponential visco-hardening with nonlinear kinematic hardening using von Mises or Hill plasticity. Specify this option as follows: **TB,RATE,,,6,EVH**

3.12.7. Lumped Matrix Formulation with Beam, Pipe, or Shell Elements

The mass inertia contributions to rotational degrees of freedom are now considered when using the lumped matrix formulation (**LUMPM,ON**) with element types **BEAM188**, **BEAM189**, **PIPE288**, **PIPE289**, **SHELL181**, and **SHELL281**. This change means that you can now account for dynamic torsional effects for beams or pipes in your analysis when using the **LUMPM,ON** command. Accuracy may be affected by a small amount.

3.12.8. Contacting Area for Contact Elements

The contacting area output quantity CAREA is now reported as a single-valued element quantity for contact elements **CONTA171** through **CONTA177**. Individual nodal quantities for CAREA are no longer reported.

3.13. The ANSYS Customer Portal

If you have a password to the **ANSYS Customer Portal** (<https://www1.ansys.com/customer/>), you can view additional documentation information and late changes. Navigate to **Product Information > Product Documentation > Readme files and late document 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.

Chapter 4: AUTODYN

The following new features are exposed in ANSYS AUTODYN for Release 13.0:

4.1. Euler Solver Enhancements

The efficiency of the 3D multi-material Euler solver has been significantly increased. Speed-ups of between 1.5 and 1.75 have been observed over previous versions of the software.

4.2. Interaction Enhancements

4.2.1. Automatic Coupling Set-Up

A Fully Auto option has been added when using the Fully Coupled Euler-Lagrange coupling method. This option will fully automate the set up of the Euler-Lagrange/Shell coupling. All structural Parts will be coupled to the Eulerian Parts in the model, and all shell Parts in the model will be thickened in such a way that they will fit the region of the Euler mesh they initially reside in.

There is also a Semi-Auto option where only the set-up of the shell coupling is automatic. The coupling properties of any other structural parts remain unchanged.

Significant improvements to the robustness of shell coupling at T-sections (or locations where more than 2 shell faces are connected along a common edge) have been made. Only the normal of relevant shell elements is used to obtain a smooth coupling surface at those locations, which is a similar procedure as is available in the existing Joins options for the Manual coupling setup of structured parts.

4.2.2. Efficient Treatment of Fully-Constrained Rigid Parts with Full Coupling

Improvements have been implemented for the treatment of cover fractions for fully-constrained rigid parts. These cover fractions are now only calculated at the start of the analysis. Furthermore, such parts can be included in models also containing other moving rigid and/or flexible parts. Therefore, when using Full Coupling, fully-constrained rigid parts (and structured Fill parts) are more efficiently calculated in combination with other moving rigid and/or flexible parts.

4.3. Analytical Blast Boundary

A new **Analytical Blast** boundary condition based on the United States Army Technical Manual, TM 5-855-1 is available to allow efficient simulation of blast loading from air or surface explosions. The boundary condition is available for solid and shell elements in the AUTODYN component system.

4.4. Remote Points and Displacements

Remote points and displacements can now be defined and used within the **Explicit Dynamics system**. Remote points are transferred to AUTODYN as a rigid body connection. Remote displacements are applied to a single node of the remote point rigid body. Remote points and displacement objects cannot be modified within the AUTODYN component system.

4.5. Parallel Processing

4.5.1. HP-MPI Message Passing Protocol

The use of WMPI as the parallel message passing protocol has been discontinued and AUTODYN is now only made available using HP-MPI as the parallel message passing protocol. Please note that dynamic spawning of slave processes from the AUTODYN component system is no longer available and that the number of slave tasks is specified before starting the AUTODYN application.

4.5.2. Automatic Decomposition of Euler parts

The automatic parallel decomposition algorithm has been extended to include Multi-material Euler parts. Previously it only handled Ideal Gas Euler and unstructured grids. This facilitates the decomposing of complicated Euler/Lagrange coupled models—you need only stipulate the number of tasks over which the model should be assigned and AUTODYN will automatically produce a decomposition configuration with good load balancing qualities and minimal inter-processor communication.

4.6. Shells with Variable Thickness

AUTODYN supports the analysis of models generated in the Explicit Dynamics system that contain shell parts with variable thickness. Shells with a constant, tabular, and functional thickness definition are all supported. The generation of shell parts with a variable thickness in AUTODYN itself is not supported.

Chapter 5: ICEM CFD

5.1. Highlights of ANSYS ICEM CFD 13.0

Release 13.0 comprises improved implementation of ANSYS ICEM CFD meshing technology as a standalone application and within the ANSYS Workbench-based Meshing application.

5.2. Key New Features/Improvements

ANSYS ICEM CFD 13.0 includes the following new features and improvements:

- 5.2.1. Workbench Integration
- 5.2.2. Geometry
- 5.2.3. Hexa
- 5.2.4. Mesh Editing
- 5.2.5. Tetra/Prism
- 5.2.6. General

5.2.1. Workbench Integration

- Workbench is now included with the install of ANSYS ICEM CFD. This makes it easier for users to access the **Workbench Readers** and other Workbench-related functionality.
- The connection to the Workbench CAD readers has been enhanced and updated, including better support for mixed dimension (shells and solids) geometries.
- ANSYS Workbench Meshing now supports “ANSYS ICEM CFD Interactive”. This allows users to export their geometry and mesh into ANSYS ICEM CFD for meshing, blocking or mesh editing. The final ANSYS ICEM CFD mesh can be saved and brought back into ANSYS Workbench Meshing. Scripts and batch mode can also be used.

5.2.2. Geometry

- STL files containing multiple parts are now imported with multiple part names.
- The licensing check for STL export has been removed.

5.2.3. Hexa

- Error messages have been improved and minor error messages are now hidden by a **Verbose mode** setting under **Hexa/Mixed Meshing Options**.
- A Hexa selection option to select by vertex has been added. This is most useful in conjunction with error messages that reference vertex numbers.
- Handling of settings and edge parameters during the **Load Blocking**, **Create Block**, and **Update Sizes** operations has been improved.
- Hexa smoothing has been significantly enhanced, particularly for unstructured Hexa.
- Multi-zone quality, robustness and speed have been improved across the board.

- The reliability of imprinting and sweeping has been improved.

5.2.4. Mesh Editing

- The new ANSYS CFD “**Orthogonality Quality**” mesh metric has been added to ANSYS ICEM CFD.
- Other quality metrics have been organized and several quality criterion and display defects have been resolved.
- Unstructured smoothing has been enhanced in several ways, including enhanced min. edge smoothing for PI Tetra and gradient smoothing for unstructured nodes.
- The Hexahedral mesh smoothing has been significantly enhanced.
- Orthogonality based smoothing now handles curve projected nodes well and internal interpolation has been improved.

Structured smoothing methods such as Sorenson/Thomas & Middlecoff have also been added (beta options).

- The capability to merge tetra and hybrid mesh has been added. Previously you could only merge tetra with pure tetra or pure hexa mesh, but now hybrid mesh (combination of quad and tri faces) is supported.

5.2.5. Tetra/Prism

- Multi-threaded (SMP) mesh generation and smoothing has been added.
- .Octree Tetra can combine the **Visible geometry** option with the **Use existing mesh parts** option.
- Delaunay, robustness, flood fill and fill from quad mesh have been improved.
- Prism smoothing settings have been improved.

5.2.6. General

- A number of other defects and minor feature requests have been taken care of.
- A number of User Interface improvements have been made for easier accessibility of options.
- Overall graphics speedup makes it significantly faster to enable/disable elements. Histogram display is also faster.

5.3. Documentation

All documentation for **ANSYS ICEM CFD 13.0** is accessible using the Help menu. Please contact us if you would like to attend training. Please visit the [ANSYS ICEM CFD website](#) for more information.

5.3.1. Tutorials

5.3.1. Tutorials

Some tutorial examples are available within the Help. Additional tutorials, input files, as well as the solved tutorials are available at <http://www.ansys.com/tutorials>.

Chapter 6: TurboGrid

This section summarizes the new features in ANSYS TurboGrid Release 13.0.

New Features and Enhancements

The following is a list of new features and enhancements in ANSYS TurboGrid:

- A significant new feature is the ATM Optimized topology definition. This feature enables you to create high-quality meshes with minimal effort; there is no need for control point adjustment. It is an alternative to the standard topologies. To use this feature, set **Topology Definition** > **Placement** to *ATM Optimized*, in the **Topology Set** object. For more information about this feature, see [ATM Optimized Topology in the TurboGrid User's Guide](#). (Note that when this feature was a Beta feature, you could only use it for blades without 'cut-off or square' leading or trailing edges. This limitation has since been removed.)
- The expression editor was enhanced in the following ways:
 - There is now syntax highlighting for components of expressions (variables, locators, functions, and expressions).
 - There is a shortcut menu for selecting variables, locators, expressions, and functions while entering an expression.
 - Expressions are organized in a tree view that comes equipped with a shortcut menu for managing expressions.
 - ANSYS TurboGrid has support for using ANSYS Workbench input parameters and ANSYS Workbench output parameters.
- You can use expressions to set the values of many of the settings found in a CCL object, directly from the associated object editor (that is, without having to use the **Command Editor** dialog box or other means to change the CCL directly).

Chapter 7: FLUENT

7.1. Introduction

ANSYS FLUENT 13.0 contains new features and defect fixes. The sections that follow provide information on *New Features in ANSYS FLUENT 13.0* (p. 55), *Supported Platforms for ANSYS FLUENT 13.0* (p. 58), *Known Limitations in ANSYS FLUENT 13.0* (p. 59), *Limitations That No Longer Apply in ANSYS FLUENT 13.0* (p. 61), and *Updates Affecting Code Behavior* (p. 61).

7.2. New Features in ANSYS FLUENT 13.0

New features available in ANSYS FLUENT 13.0 are listed below.

- Solver
 - Pseudo-transient relaxation method
 - Conservation of rothalpy transport equation
- Models
 - Turbulence
 - SAS turbulence model
 - Embedded/zonal LES (E-LES)
 - Enhanced wall treatment for the omega transport equation
 - Compatibility of the k-omega turbulence model with multiphase models
 - Turbulence transition model for rough walls
 - Heat transfer
 - Shell conduction zone manager
 - Improved shell conduction model performance
 - Cluster-to-cluster viewfactor calculations
 - Ability to compute solar loads in the parallel solver
 - Discrete Ordinates and P-1 radiation models compatible with Eulerian-Eulerian multiphase (non-granular)
 - Unlimited number of gray bands with the Discrete Ordinates model
 - Multi-band modeling now available with the P-1 radiation model
 - Improved parallel performance of the ray-tracing module
 - Species transport, reactions and combustion
 - Multiple spark model
 - Veynante extended coherent flame model (ECFM)

- Option to model Arrhenius inter-phase reactions
- Improved surface chemistry solver robustness
- Improved multi-component solidification model
- Faster unsteady NOx pollutant modeling
- Characteristic time model
- G-equation model
- Chemistry agglomeration for faster detailed chemistry
- Chemical mechanism dimension reduction
- Compatibility of non-premixed model with the real gas equation of state
- Thickened flame model
- Unburnt partially-premixed properties extended to include a second mixture fraction
- Ability to solve for detailed chemistry as a postprocessing step on a frozen flow field solving for selected pollutant species using constrained chemical equilibrium
- Ability to report element flux balances as an additional check of convergence
- Ability to input a single value of URF and Spatial Discretization to be used for all species
- Discrete phase model
 - KHRT breakup model
 - Transient mass flow rate and velocity for injections
 - Dense DPM extended to include the packing limit
 - Improved DPM parallel performance
 - Ability to export particle data to CFD-Post for postprocessing
- VOF
 - Compressive scheme
 - Zone specific VOF discretization
 - BGM scheme for steady-state VOF
 - Turbulence damping sources near free surfaces
 - Ability to specify higher-order waves
 - Numerical beach option for open channel flows
 - Coupled level-set method
- Eulerian multiphase model
 - Multi-velocity sectional population balance module
 - Ability to model sub-cooled boiling, including non-equilibrium sub-cooled boiling
 - Improved treatment of volume fraction gradients for gas-liquid flows (improved robustness)
 - Ability to include real gas properties in Eulerian multiphase simulations
- Population balance

- Laakkonen kernel
- Inhomogeneous discrete population balance model
- Multi-velocity sectional population balance module
- Boundary conditions
 - Subcritical flow enhancement for open channel boundary condition (VOF)
 - Bounded second order discretization in time
 - Ability to assign a velocity inlet boundary condition with compressible flows
 - Hybrid initialization
 - Average pressure specification boundary condition method compatible with the PBNS solver
- Material properties
 - Droplet material properties expanded to include DPM vapor pressure data up to the critical point
 - Cryogenic droplet materials added
 - Peng-Robinson, Redlich-Kwong and Soave-Redlich-Kwong real-gas equations of state
 - Real gas models extended to sub-critical regime
 - Multiple mixture materials for species transport
 - Ability to use other materials with user-defined real gas
- Data import and export
 - Ability to append case and data files in the parallel solver
 - Ability to export custom field function data in parallel to Fieldview
 - Ability to import TecPlot 360 meshes (including polyhedral cells)
 - Ability to export data in ASCII format in the parallel solver
- Mesh
 - Transport equation-based (diffusion-based) mesh smoothing
 - Key frame mesh swapping
 - Ability to include adjacent boundaries during zone remeshing
 - Ability to replace zones in parallel
 - Steady non-conformal interfaces are now preserved in transient simulations for improved performance
 - Mesh morpher and optimizer
 - Ability to identify and improve poor quality cells
 - Cartesian remeshing (without boundary layer remeshing)
 - Mesh check progress indicator
- Moving meshes
 - Ability to define an MRF zone within an MRF zone
 - Ability to specify a moving reference frame independent of the movement of the mesh for the same zone
- Porous media
 - Ability to define porous jump as a non-transparent surface when using the solar load model

- Ability to specify contact angle on porous jump boundaries (VOF)
- Parallel processing
 - No longer necessary to encapsulate hexcore meshes
 - Improved performance for case file I/O
 - Optimized .pdat files for cases with large numbers of zones
 - Optimization for multicore architectures
 - Improved scalability for cases with sliding interfaces
 - Parallel I/O support for Lustre on Linux
 - Speedup of parallel checkpointing
 - Extended parallel file system support
 - Ability to couple FLUENT to the Remote Solver Manager (RSM). (Serial or local (shared memory) parallel jobs only)
- Memory management
 - User-defined memory for nodes
 - Optimized memory usage for Tmerge utility
- Graphics, postprocessing, and reporting
 - Ability to report fluxes for dense DPM phases
 - Ability to enable/disable in-cylinder specific output
 - Optimized monitor data writing
 - Ability to view mesh interfaces from the mesh interface dialog box
 - Selected boundary surfaces are now highlighted in the graphics window when selected
 - Ability to export DPM particle data to CFD-Post
 - Ability to plot radiative heat flux at flow boundaries
 - Ability to display and highlight selected surfaces/boundaries in the graphics window
 - Wild-card support for selecting surfaces with post-processing TUI commands
 - FLUENT window title text prepended by case name
- User-defined functions (UDFs) and user-defined scalars (UDSs)
 - UDF access to MRF specifications for non-constant motion
 - Spark model user hooks
 - UDF access to model non-constant frames of motion in moving reference frames
 - UDF access to the absorption coefficient computed by the WSGGM
- User interface
 - Toolbar buttons now available for standard views
- Multi-physics
 - FLUENT coupling with HFSS/Maxwell/Q3D Extractor

7.3. Supported Platforms for ANSYS FLUENT 13.0

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

7.4. Known Limitations in ANSYS FLUENT 13.0

The following is a list of known limitations in ANSYS FLUENT 13.0.

- File import/export
 - Data export to Mechanical APDL result file is not available on the linux64 and linia64 platforms. (Mechanical APDL data export to .cdb file is available on all platforms)
 - When exporting EnSight Case Gold files for transient simulations, the solver cannot be switched between serial and parallel, and the number of compute nodes cannot be changed for a given parallel run. Otherwise, the exported EnSight Case Gold files for each time step will not be compatible
 - EnSight export with topology changes is not supported
 - To properly view Fieldview Unstructured (.fvuns) results from a parallel ANSYS FLUENT simulation
 - Mesh files must be exported from the parallel solver via the TUI command fieldview-unstruct-grid
 - Mesh and data files should all be exported from parallel ANSYS FLUENT sessions with the same number of nodes
 - Import of 3D ABAQUS files in .odb format is not available on the IBM platform
 - Tecplot file import does not support the Tecplot360 file format
 - Import of ANSYS CFX definitions or results files is no longer supported on the hpia64 platform
- Mesh
 - Boundary zone extrusion is not possible from faces that have hanging nodes
 - The following features are incompatible with polyhedral cell types:
 - Moving/deforming mesh
- Models
 - ANSYS FLUENT supports the Chemkin II format for Oppdif flamelet import only
 - The surface-to-surface (S2S) radiation model does not work with sliding and moving/deforming meshes
 - The work pile algorithm is not compatible with the wall film boundary condition
 - The shell conduction model is not applicable on moving walls
 - The heat exchanger model is not compatible with mesh adaption
 - The FLUENT/REACTION DESIGN KINetics coupling is not available on the win64 platform
 - DO-Energy coupling is recommended for large optical thickness cases (> 10) only
 - FMG initialization is not available with the shell conduction model
 - FMG initialization is not compatible with the unsteady solver
 - The MHD module is not compatible with Eulerian multiphase models
 - Bounded 2nd order discretization in time is not compatible with moving and deforming mesh
- Parallel processing
 - These features are currently unavailable in the parallel solver:
 - Discrete transfer radiation model (DTRM)
 - Continuous Fiber Model (CFM) add-on module

→ Data export to non-native formats other than EnSight, FIELDVIEW, Tecplot, and the generic heat flux data file

- Platform support and drivers

- Windows machines using HPMPPI are required to be logged onto a network domain. For shared memory parallel runs on a machine disconnected from the network, MPICH2 is recommended
- ANSYS FLUENT 13 is not compatible with the job scheduler on HPC Server 2008 with the packaged version of HPMPPI. The default MPI (MSMPI) should be used
- The minimum OS requirements for Linux are SLES 10 or Red Hat Enterprise 5.0
- The path name length to the cpropep.so library (including the lib name) is limited to 80 characters. (Linux Opteron cluster using Infiniband interconnect only)
- OS level 5300-07 (maintenance level 7) or higher is required on the IBM AIX platform
- Visit the User Services Center for the latest Windows graphics FAQ. Version 2.0 or higher of .NET Framework must be installed in order to run ANSYS FLUENT on the winx64 platform

Please view the FLUENT Product Page on the [User Services Center](#) for more information.

- Solver

- The non-iterative time advancement (NITA) solver is applicable with only a limited set of models. See the ANSYS FLUENT User's Guide for more details.
- NITA (using fractional time step method) is not compatible with porous media
- The following models are not available for the density-based solvers:
 - Volume-of-fluid (VOF) model
 - Multiphase mixture model
 - Eulerian multiphase model
 - Non-premixed combustion model
 - Premixed combustion model
 - Partially premixed combustion model
 - Composition PDF transport model
 - Soot model
 - Rosseland radiation model
 - Melting/solidification model
 - Enhanced Coherent Flamelet model
 - Inert model: transport of inert species (EGR in IC engines)
 - Dense discrete phase model
 - Shell conduction model
 - Floating operating pressure
 - Spark ignition and auto-ignition models
 - Physical velocity formulation for porous media
 - Selective multigrid (SAMG)

- The following models are not available for the pressure-based solver:
 - Non-reflecting boundary conditions
- The pressure-based coupled solver is not available with the following features:
 - Porous jump boundary condition
 - Fixed velocity
- User-defined functions (UDFs)
 - The DEFINE_RW_FILE macro is not supported on the Windows platform
 - Interpreted UDFs cannot be used while running in parallel with an Infiniband interconnect. The compiled UDF approach should be used in this case
- Third-party software
 - FLUENT-Platform LSF integration is not supported on the MS Windows platform
 - FLUENT-SGE integration is supported only on the AIX 5.3 and Linux platforms
 - Wave and GT-Power coupling are available only with stand-alone ANSYS FLUENT and not in the Workbench environment
 - Wave 8.0 does not support IBM platforms
 - ANSYS FLUENT 13 uses the CHEMKIN-CFD KINetics library 2.4. This version no longer supports the hpia64 and linia64 platforms
 - GT-Power is supported on the 32- and 64-bit Linux and Windows platforms, as well as the IBM platform
- Other
 - The IRIS Image and HPGL hardcopy formats are no longer supported in ANSYS FLUENT

7.5. Limitations That No Longer Apply in ANSYS FLUENT 13.0

- It is no longer necessary to encapsulate hexcore meshes
- It is now possible to replace zones in the parallel solver
- It is now possible to append case and data files in the parallel solver
- Solar loads can now be computed in the parallel solver
- The discrete ordinates and P-1 radiation models are now compatible with Eulerian-Eulerian multiphase
- Solid convection can now be modeled using higher order discretization schemes

7.6. Updates Affecting Code Behavior

- The wall treatment for omega in the k-omega turbulence model has been improved to be less sensitive to the near-wall mesh resolution than in previous FLUENT versions. The previous treatment method can be recovered with the rpvar setting "(rpsetvar 'kw-wall-omega-treatment-r13? #f)"
- More realistic (non-constant) epsilon and omega fields are now used for initialization when the turbulence model is changed between k-omega and k-epsilon to improve convergence at restart
- The use of the PDF and Turbulent Schmidt numbers (Sc) in FLUENT's partially premixed combustion model has changed from previous FLUENT releases. The table below describes the assignment of these variables to the model's mixture fraction and reaction progress equations for the current and previous releases. (CL71141)

Partially Premixed Combustion Model		
FLUENT Version	Mixture Fraction Equations	Reaction Progress Equation
6.3 and Earlier Releases	PDF Sc	PDF Sc
12 and 12.1	Turbulent Sc	Turbulent Sc
13	PDF Sc	Turbulent Sc

The PDF Schmidt number is set in the **Viscous Model** dialog box (default = 0.85) and the Turbulent Schmidt number is set in the **Species Model** dialog box (default = 0.70)

- The Surface-to-surface (S2S) radiation model GUI panel is improved for usability through more clearly labeled settings and more intuitive organization. Additionally, defaults have been updated to correspond to current best-practices, and options that are not generally recommended are no longer present in the GUI.
- The default view factor computation method for the surface-to-surface radiation model has been changed from the hemicube method to the ray tracing method
- The default number of sub-divisions for the hemicube view factor computation method has been changed from 10 to 5
- The Adaptive view factor computation method has been removed from the GUI as an option for the Surface-to-surface radiation model
- Least squares smoothing has been removed from the GUI as a view factor calculation option for the Surface-to-surface radiation model
- The **smooth** option has been removed from the Surface-to-surface radiation model
- The default value for the maximum number of radiation iterations for the DTRM and Surface-to-surface radiation models has been changed from 10 to 5
- The determination of the collision volume for discrete phase particles has been modified for 2d axisymmetric cases when the spray collision/coalescence model is used. (CL71429)
- The calculation of mean NO_x production rate has been modified in FLUENT 12 and later versions. (CL68308)
- The two-competing rates devolatilization model has been modified to correct a problem with devolatilization at partition boundaries (parallel solver only) (CL71435)
- Solid convection can now be modeled using higher order discretization schemes (selected in Solution Methods panel). In previous FLUENT versions solid convection used first order upwind spatial discretization regardless of the discretization scheme selected. (CL70951)
- The default droplet material profile for vapor pressure has been changed from constant to piecewise linear. (CL71648)
- The improved wall treatment rpvvar default for the VOF model has been changed from "true" to "false" following the discovery of several cases showing unphysical results. (CL62137)
- Before defining a monitor on an iso-surface that is dependent on solver data (e.g. velocity, pressure, custom field function etc.) the solution must now be initialized first or an error will result. (CL72914)
- The cell-based Weighted Sum of Gray Gases model (WSGGM) has been removed from the GUI. This option is still available in Fluent 13 as a TUI-only feature.
- The boiling temperature and latent heat property inputs were unnecessary when defining a real gas equation of state with the DPM model and have been removed.
- The method 1 TUI option for target mass flow rate has been removed since the method 2 option covers all cases. Method 1 is still available as an rpvvar controlled option.

- The Zimont model has been renamed the C-equation model in FLUENT 13.0.
- Improvements have been made to the vortex method for applying turbulence boundary conditions for Large Eddy Simulations. Results may vary from previous releases.
- The TUI command `Use max cell edge for LES length scale?` has been removed and is now only available via the `rpvar des-maxedge?`

Chapter 8: CFX

This section summarizes the new features in ANSYS CFX and CFD-Post Release 13.0.

[8.1. New Features and Enhancements](#)

[8.2. Incompatibilities](#)

8.1. New Features and Enhancements

New features and enhancements to ANSYS CFX and CFD-Post introduced in Release 13.0 are highlighted in this section.

8.1.1. ANSYS CFX in ANSYS Workbench

The operation of ANSYS CFX in ANSYS Workbench is described in *ANSYS CFX in ANSYS Workbench* (p. 3).

8.1.2. ANSYS CFX in General

Solids can now be included in porous domains to model conjugate heat transfer and/or the transfer of Additional Variables.

There is a new six-degrees-of-freedom (6DOF) Rigid Body feature that enables you to model a rigid body as a collection of 2D wall boundaries. You can also make an immersed solid act like a rigid body. In both cases, fluid forces and specified external forces/torques act on the rigid body. Each approach has its advantages and disadvantages:

- When modeling a rigid body as a collection of wall boundaries, wall boundaries are sharply resolved, but the mesh must be distorted to follow the boundary motion.
- When modeling an immersed solid as a rigid body, the mesh undergoes no distortion but the wall boundaries are not sharply resolved.

ANSYS CFX now works with Remote Solve Manager (RSM).

For this release, ANSYS CFX also introduces the following features:

- Particle Tracking: Particle Source Bounding
- Turbulence Bounded CDS
- Boundary Condition Radial Equilibrium
- Particle Tracking Sommerfeld Virtual Wall
- Soave Redlich Kwong Equation of State
- RIF Extensions in CFX-Pre

8.1.3. ANSYS CFX Documentation

ANSYS CFX documentation now appears in the ANSYS Help Viewer, which makes it easier for you to find information about other ANSYS products that you can use with ANSYS CFX.

There have been numerous incremental improvements to the organization and content of the documentation to improve clarity and usability. In addition there is a new tutorial: [Modeling a Buoy using the CFX Rigid Body Solver in the CFX Tutorials](#).

8.1.4. ANSYS CFX-Pre

This section highlights the new features supported in this release of CFX-Pre.

8.1.4.1. Efficient Handling of Large Numbers of Renderable Objects

Algorithmic changes have been made to the way render information for objects, such as color and shading, is stored and processed by CFX-Pre. These changes have been shown to give significant speed improvements for model manipulation where large numbers of objects are visible in the viewer window.

8.1.4.2. Stereo Viewer Capabilities

Stereo Viewing capabilities, previously available only in CFD-Post, are now also available in CFX-Pre. This allows full stereo-viewing capability for appropriate hardware. This feature can be enabled from the Viewer branch of the **Edit > Options** menu.

8.1.4.3. Automatic Domain Interfaces

Several improvements have been made to the way Automatic Domain Interfaces are created and checked inside CFX-Pre. This includes more robust checking of multiply-connected domains and problem regions relating to many-to-one connections.

8.1.4.4. Additional ANSYS Element Type Support

The following element types can now also be imported from ANSYS cdb files into CFX-Pre:

- Element Type 28 (Shear/Twist Panel), KEYOPT (0)
- Element Type 212 (2D 4-Node Coupled Pore-Pressure Mechanical Solid), KEYOPT (0)
- Element Type 213 (2D 8-Node Coupled Pore-Pressure Mechanical Solid), KEYOPT (0)
- Element Type 215 (3D 8-Node Coupled Pore-Pressure Mechanical Solid), KEYOPT (0)
- Element Type 216 (2D 20-Node Coupled Pore-Pressure Mechanical Solid), KEYOPT (0)
- Element Type 217 (2D 10-Node Coupled Pore-Pressure Mechanical Solid), KEYOPT (0)
- Element Type 233 (2D 8-Node Electromagnetic Solid), KEYOPT (0)
- Element Type 236 (3D 20-Node Electromagnetic Solid), KEYOPT (0)
- Element Type 237 (3D 10-Node Electromagnetic Solid), KEYOPT (0)
- Element Type 241 (2D Hydrostatic Fluid), KEYOPT (0)
- Element Type 242 (3D Hydrostatic Fluid), KEYOPT (0)
- Element Type 285 (3D 4-Node Tetrahedral Structural Solid with Nodal Properties), KEYOPT (0)

To obtain a full list of supported ANSYS element types, type the following at a command prompt

```
install_dir/v130/CFX/bin/ImportANSYS.exe -S
```

8.1.4.5. License Server Checking Improvements

License server checking has been improved, leading to shorter waiting times for diagnostic messages, particularly when license servers are unavailable.

8.1.4.6. Full User Interface Support for New Solver Models

All new CFX-Solver models in Release 13.0 can be accessed directly from the CFX-Pre graphical user interface.

8.1.5. ANSYS CFX-Solver Manager

New features and enhancements to the CFX-Solver Manager introduced in Release 13.0 are highlighted in this section.

8.1.5.1. Automatic Display of Electro-Magnetism Plots

Plot lines of related residual quantities for solution of EMAG/MHD calculations (such as Electric Potential) now appear automatically in the CFX-Solver Manager under the **Electromagnetism** tab.

8.1.5.2. License Server Checking Improvements

License server checking has been improved, leading to shorter waiting times for diagnostic messages, particularly when license servers are unavailable.

8.1.6. ANSYS CFX-Solver

New features and enhancements to the CFX-Solver introduced in Release 13.0 are highlighted in this section.

8.1.6.1. CFX-Solver

This section highlights the new features supported in this release of the CFX-Solver.

8.1.6.1.1. Turbulence

Support for the individual specification of Turbulent Prandtl and Turbulent Schmidt numbers has been added. Support has also been added for specifying these as CEL expressions.

8.1.6.1.2. Particle Tracking

The liquid evaporation model has been extended so that modeling of evaporation of more than one component of a multi-component particle is supported.

8.1.7. ANSYS CFD-Post

Volume Rendering

Volume Rendering enables you to visualize field variables throughout the entire domain by varying the transparency and color of the plot as a function of the variable value. For example, you can make realistic images of smoke and analyze how it spreads and how it affects visibility.

Chart

You can now position the chart legend on the chart area, in addition to the surrounding chart area where the axes are found.

Chart axis numbers can now be formatted in either standard or scientific notation.

Comparisons Mode

The automatic detection of the same mesh is much faster.

You can now tell CFD-Post that meshes are either same or different in the Case Comparison panel, which can make the comparison mode much faster.

Turbo Postprocessing

Hub-to-Shroud turbo line points can now be distributed based on mesh density.

Table

Table updates are much faster.

Stereo Viewer

Support is added for graphics display on typical stereo hardware.

ANSYS FLUENT Files

Particle tracks can now be exported from ANSYS FLUENT and post-processed in CFD-Post.

CFD-Post can now read interior face zones from ANSYS FLUENT files. Set this ability from the **Edit > Options > CFD-Post > Files** panel.

CFD-Post now performs more accurate interpolation inside concave polyhedra in ANSYS FLUENT files. This resolves issues with streamlines stopping in the middle of the domain when hitting a concave polyhedron.

CFD-Post can now perform more accurate (cell based) evaluation of user-defined variables.

8.2. Incompatibilities

This sections highlights differences in the behavior between Release 12.1 and Release 13.0 of ANSYS CFX and CFD-Post.

8.2.1. CFX-Pre

The following change has been made to CFX-Pre:

- The abbreviation for dram (dm) has been deprecated and now stands for decimeter.

8.2.2. CFX-Solver

Below is a list of numerics improvements and other changes made for the CFX-Solver in Release 13.0. The changes are believed to be generally helpful and should be reverted only in the event of a problem.

Convergence behavior changes (that do not affect the converged solution):

- Multiphase flow:

- The density-pressure linearization for compressible multiphase flows has been modified to improve robustness, but still recovers the same linearization as single phase flows as the volume fraction approaches 1. Revert by setting the expert parameter `compressible linearisation option = 1`.
- A problem with the homogeneous nucleation model of the small droplet condensation model is now fixed. As well, non-clipped area densities are used for the condensation rates, which improves conservation in regions of significant re-evaporation. These changes can be reverted by setting the expert parameters `nes nucleation fix = f` and `ipmt area density clip nes = t`.
- Miscellaneous:

A minor change has been made to the transient term on rothalpy for compressible rotating systems. Revert by setting the expert parameter `transient compressible rotation option = 2`.

Discretization changes (that affect the converged solution):

- Boundary conditions/GGI interfaces:
 - Flux boundary conditions for energy and scalars now evaluate profiles at integration points rather than face centers. Revert by setting the expert parameter `use bip flux = f`.
 - The treatment of gradient extrapolation at boundaries has changed so that it is now consistent in serial and parallel. Revert by setting the expert parameter `boundary vertex extrapolation option = 1`.
 - For moving mesh cases, the numerical details of moving gaps/overlaps at GGI interfaces have been modified for better conservation properties. Revert by setting the expert parameter `ggi moving mesh option = 2`.
 - The continuous phase volume fractions used in the Gidaspow and Wen Yu drag correlation are now clipped to 0.001. Revert by setting the CCL parameter `Minimum Volume Fraction for Correction = 0.0`.
 - For moving mesh cases with total energy, a problem has been fixed that caused temperature oscillations at the interface. The moving mesh contribution to the pressure work term in the total energy equation was accidentally accounted for twice.
- Multiphase:
 - For cavitation with thermal effects modeled, the interfacial temperature is now assumed to be the liquid temperature rather than the bulk temperature. Revert by setting the expert parameter `cavitation tint liquid = f`.
 - The discretization of some non-drag forces (the Favre averaged drag force and the solids pressure force) has been made more robust. The previous implementation can be recovered by setting the expert parameter `vfr gradient force option = 0`.

Reverting the calculation of non-drag forces to the original source term implementation by using the expert parameter settings of `vfr gradient force option = 0` and `virtual mass force option = 0` can improve the accuracy at GGI interfaces, at the expense of overall robustness and overall accuracy.
- Particle Tracking:
 - The discretization of the particle turbulent dispersion has been corrected. This improves convergence and robustness for particle cases using this option. It is not possible to revert this change.
 - A problem has been fixed in the Sommerfeld collision model which can lead to slightly different answers.

- In the Elsaesser wall interaction related routines, clipping of model correlations and weighting factors were introduced to help improve convergence for cases with evaporating wall particles.
- Miscellaneous:
 - A problem has been fixed for FSI cases with non-overlapping faces at the coupling interface. It is not possible to revert this change.
 - Boundary advection on GGI interfaces between solids with solid motion is now switched off if the materials in the solids are not identical. This can be reverted by setting the CCL parameter `Boundary Advection = On` in the “BOUNDARY CONDITION | SOLID MOTION ADVECTION” sections of the interface boundaries.
 - Updates between coupling iterations and time steps of quantities derived from solution variables have changed for all transient two-way couplings that utilize multiple coupling iterations per step. The changes, introduced due to a defect correction, are most evident in the convergence history rather than in the actual results. It is not possible to revert this change.
- Parallel
 - A bug has been fixed for the calculation of the RMS Courant number used in the adaptive time step control.

Other changes:

- Turbulence: SST SAS turbulence model

Since Release 12.0 the two model versions, 2005 and 2007, have existed. When creating a new SST SAS setup in CFX-Pre, the correct default version, 2007, was used. However, when the parameter `Model version` was missing in a setup (for example, a CFX-11 SST-SAS setup or a setup of another turbulence model switched to SST-SAS by editing the CCL), the previous model version, 2005, was used in Release 12.0. This has been corrected in Release 13.0 so that the new model version, 2007, is used.

8.2.3. CFX-Solver Manager

The following changes have been made to CFX-Solver Manager:

- The monitor data generated by the solver is now written to the results file using 8 significant figures by default (previously, the default was 5). This will increase the size of the monitor data significantly, although for most runs the increase in size of the results file will be small. To revert the behavior, set the expert parameter `monitor digits = 5`.

In addition, the precision of the monitor data exported from CFX-Solver Manager (exported by, for example, using the right-mouse button on a specific plot window, and then selecting **Export Plot Data**), and the precision of the data extracted and used by the utility `cfx5mondata`, has been changed to be consistent with the above change for the solver (that is, 8 significant figures). Previously the data was exported using a free format with undefined precision.

8.2.4. CFD-Post

This section describe procedural changes (actions that have to be done differently in this release to get an outcome available in previous releases) as well as support changes (functionality that is no longer supported) in 13.0 of CFD-Post.

Procedural Changes

When CFX-Solver Manager is opened from ANSYS Workbench, the **Custom Executable** and **Arguments** fields are no longer present on the Solver Tab of the CFX-Solver Manager Define Run dialog (although these fields remain available in the Standalone version of CFX-Solver Manager). You can set those properties via the Solution cell Properties view (this capability is a Beta feature).

Listing Files from a Transient Simulation

In Release 12.0, the files present in the working directory were checked first and all the files with same base name were listed in CFD-Post. If the base name of none of the files matched, CFD-Post listed any appropriate files found in 'autosave/solution-points'.

In Release 13.0, CFD-Post lists any appropriate files found in 'autosave/solution-points'. If no appropriate file is found, the working directory is checked and files with same base name are listed. The new sequence gives higher priority to the files from the chosen DAT/CDAT file.

Chart Legends

A Release 12.0 chart that has its legend on the inside of the chart area may display with the legend in a slightly different position in Release in 13.0.

Display of Mean Molecular Weight Values from FLUENT Files

In Release 12.0, CFD-Post showed incorrect units for Mean Molecular Weight in results files exported from FLUENT; for example, a value that should have appeared as 23.1577 kg/kmol would display as 23157.7 kg/kmol. In Release 13.0, the same case will display a Mean Molecular Weight of 23.1577 kg/mol.

Maximum Temperature Limit of Zero Pressure Polynomials

In Release 12.0, the maximum temperature limit of zero pressure polynomials was 1000 [K] for the CFX-Solver, 3000 [K] in the RULES file, and 5000 [K] for CFX-Pre.

In Release 13.0, maximum temperature limits for zero pressure polynomials are now consistent in RULES, CFX-Pre, and the CFX-Solver (= 1000 K). Temperature limits for table generation are set by CFX-Pre: 5000 K for ideal gases, 1000 K for real gases.

Support Changes

There are no support changes in this release.

Chapter 9: POLYFLOW

9.1. Introduction

ANSYS POLYFLOW 13.0 is the second version of **ANSYS POLYFLOW** to be integrated into **ANSYS Workbench**. In version 12.1, **ANSYS POLYFLOW** users were able to create interlinked systems with geometry, meshing, solution setup, solver and postprocessing inside **ANSYS Workbench**, using shared licensing and HPC. Blow molding and extrusion application-specific versions of **ANSYS POLYFLOW** were introduced to allow specific industrial processes to be simulated. With regard to modeling, two new models were introduced: the volume of fluid (VOF) model for free surface modeling in a fixed domain; and the discrete ordinates (DO) model for radiation.

In **ANSYS POLYFLOW** 13.0, the **ANSYS Workbench** integration, licensing, and modeling capabilities have been further enhanced to meet the needs of **ANSYS POLYFLOW** users.

Note

ANSYS POLYFLOW 13.0 is installed under `ANSYS Inc\v130\polyflow` on Windows and `ansys_inc/v130/polyflow` on UNIX/Linux platforms.

ANSYS POLYFLOW 13.0 is available within **ANSYS Workbench** for Windows and UNIX/Linux platforms.

9.2. New Features

The new features in **ANSYS POLYFLOW** 13.0 are as follows:

- The message handling between the **ANSYS POLYFLOW** application and the **ANSYS Workbench** environment is enhanced:
 - Error messages and status updates during setup and solution is displayed in the **ANSYS Workbench Messages** window.
 - For transient and evolution problems, **ANSYS POLYFLOW** displays status messages in the **ANSYS Workbench Messages** window during the solver run.
 - The **Pause** button in **ANSYS POLYDATA** has been removed, so you can seamlessly switch between **ANSYS POLYDATA** and other **ANSYS Workbench** applications with shared licensing.
- The licensing capabilities for **ANSYS POLYFLOW** are enhanced with additional capabilities:
 - **ANSYS POLYDATA** can use the **ANSYS CFD PrepPost** license, which allows you to perform meshing, setup and postprocessing.
 - A new license named **ANSYS POLYFLOW Solver** is available, which can be used for **ANSYS POLYDATA** and **ANSYS POLYFLOW** solver runs only.
 - You no longer need to set the order of licensing preferences when using the blow molding and extrusion application-specific versions of **ANSYS POLYFLOW**.

- The volume of fluid (VOF) model is enhanced to allow the robust computation of complex 3D flows with viscoelasticity, contact, and non-isothermal effects.
- The internal optimization capabilities are extended such that you can specify all scalar variables as design variables.
- Rectilinear mold motion with specified forces can be performed for blow molding and thermoforming simulations.
- A sliding mesh feature is available, which makes it possible to simulate transient flows with internal moving parts (e.g., single screw extruders, stirring tanks, non-intermeshing batch mixers).
- Boundary conditions are enhanced for ease of use:
 - Wall boundaries with a zero wall velocity can be specified directly in the boundary conditions menu.
 - It is possible to specify an inlet mass flow rate for extrusion problems.
- Mesh and results export and import are enhanced to allow the seamless transfer of data into and out of **ANSYS POLYFLOW**:
 - **HyperMesh** meshes can be imported.
 - The thickness data field can be exported to **ANSYS Mechanical Classic (Mechanical APDL)** as `.cdb` files using the Mechanical APDL format.
 - You can convert shell and 3-D meshes and results to **LS-DYNA**.
- You can use the Remote Solve Manager (RSM) service in **ANSYS Workbench** to perform **ANSYS POLYFLOW** solver computation on other CPUs or a cluster on your network.
- A graphics toolbar has been added to the **Graphics Display** window in **ANSYS POLYDATA**, in order to allow you to manipulate the view. The buttons in this toolbar are consistent with other **ANSYS** components, such as **ANSYS DesignModeler**, **ANSYS Mesher**, and **ANSYS CFD-Post**.

9.3. Defect Fixes

The defect fixes in **ANSYS POLYFLOW** 13.0 are as follows:

- The licensing layer has been corrected for the blow molding and extrusion application-specific versions of **ANSYS POLYFLOW**.
- The integration rule for 2x2 on tetrahedral meshes has been improved.
- **ANSYS POLYDATA** has been improved to allow for transient flow rates in a VOF simulation.
- The `vof_sample` example has been corrected for Windows.
- The warning for linear velocity has been removed (it is valid with stabilization).
- Force and torque along a moving part can be exported as probe files.
- Various corrections have been applied to the VOF model, including viscoelasticity.
- Pressure stabilization has been corrected for axisymmetric problems.
- Fixed probes can be created for a given geometric location.
- Errors in the temperature field have been corrected for axisymmetric problems with non-conformal meshes.
- The minimum and maximum values computed for scalar fields have been corrected for **ANSYS CFD-Post**.
- Corrections have been made to ensure that all fields defined along interfaces between sub-domains are exported to **ANSYS CFD-Post**.

- The conversion of evolution tasks has been improved to avoid the failure of VOF tasks.
- Problems in **ANSYS POLYDATA** with the interruption criterion on the thickness field have been corrected.
- The AMF direct + secant solver has been corrected to address the wrong dimension of an internal table that stops the solver.
- The loading time of large meshes in the Windows version of **ANSYS POLYDATA** has been improved.
- Corrections have been made to ensure the proper importation of **GAMBIT** neutral files that contain external PMeshes.
- Improvements have been made so that PMeshes that contain duplicates are automatically fixed.
- A variety of corrections have been initiated in **ANSYS POLYMAN**, **ANSYS POLYFUSE**, **ANSYS POLYSTAT**, and **ANSYS POLYCURVE**, including relative pathnames, slicing in reverse order, and saving of session files.
- The tracking of integral viscoelastic calculations has been corrected with regard to revised geometrical tolerance.

9.4. Known Limitations

The known limitations for **ANSYS POLYFLOW** 13.0 are as follows:

- The **Interrupt** action in **ANSYS Workbench** has no effect on an **ANSYS POLYFLOW** solver run.
- You cannot perform any actions that modify an **ANSYS POLYFLOW** system (e.g., saving or closing a project, duplicating an **ANSYS POLYFLOW** system) while an **ANSYS POLYFLOW** tool is open. In some cases, **ANSYS Workbench** will allow such an action, but an error is generated.

Chapter 10: Icepak

10.1. Introduction

ANSYS Icepak 13 is a release of ANSYS Icepak that has new features and defect fixes. New features are listed in the following section of this document. A list of defects fixed in this release is accessible from our online FLUENT User Services Center (www.fluentusers.com/icepak/index.htm).

10.2. New and Modified Features in ANSYS Icepak 13

- Graphical User Interface
 - The ANSYS viewer will display ANSYS Icepak documentation. See [Accessing Online Help](#) of the User's Guide.
 - Material property browsing with bubble help for quick verification of material properties. See [Material Properties](#) of the User's Guide.
 - Block object face highlighting to indicate sides with specified properties. See [Display Options](#) of the User's Guide.
 - Automatic update orthotropic material properties for move/rotate operations. See [Editing an Existing Material](#) of the User's Guide.
- ECAD Import/Export
 - ANF files can now be imported for detailed printed circuit board and package modeling. See [Importing Trace Files](#) of the User's Guide.
 - Automatic creation of meshing plates including modifications to the plates using the **Model layers separately** option. See [Importing Trace Files](#) of the User's Guide.
 - Purge Inactive Objects option in the **IDF Import** dialog box. See [Reading an IDF File Into ANSYS Icepak](#) of the User's Guide.
- Model Import/Export
 - Transient setup of models is now possible using CSV files. This capability enables import/export of all transient parameters including piecewise linear property curves using CSV files. See [Displaying the Variation of Transient Parameters with Time](#) of the User's Guide.
 - Network objects can now be imported and exported in CSV format. This aids in the quick editing of complicated networks. See [Networks](#) and [Networks](#) of the User's Guide.
 - CSV import/export capability has been enhanced to include import/export of geometric and non-geometric parameters.
- Modeling and meshing
 - Zero slack is allowed for non-conformal assemblies. See [Non-Conformal Meshing Procedures for Assemblies](#) of the User's Guide.
 - Solar load modeling is available. This capability allows users to automatically account for heat transfer at all surfaces due to solar loading. See [Modeling Solar Radiation Effects](#) of the User's guide.

- A cluster-based ray tracing radiation model is now available. See [Ray Tracing Radiation Modeling of the User's Guide](#).
- CAD shapes are now available for grilles, openings, fans and walls. See [CAD Objects](#) of the User's Guide.
- Delphi package characterization is available. This procedure allows users to automatically extract compact thermal models of packages. See [Delphi Package Characterization](#) of the User's Guide.
- Enhanced heat transfer coefficient boundary conditions are available for block sides. See [User Inputs for the Block Thermal Specification](#) of the User's Guide.
- Heat pipe macro is available. See [Heat Pipes](#) of the User's Guide.
- Optimization for package and PCB geometries is available. See [Meshing Options](#) of the User's Guide.
- Anisotropic or isotropic refinement is available for 2D multi-level meshing. See [Meshing Options](#) of the User's Guide.
- Postprocessing and reporting
 - **Summary report** now includes mesh option. This option allows users to report results on meshed areas of objects. See [Summary Reports](#) of the User's Guide.
 - **Heat Flux Vectors** are exported by ANSYS Icepak and can be read by CFD-Post. See [Advanced Solution Control Options](#) of the User's Guide.
- Miscellaneous
 - Enhanced material libraries including new heat spreader materials.
 - New libraries including ALPHA heatsinks.
 - Option to include temperature secondary gradients for skewed meshes is available. This option improves accuracy of the solution to energy equation for skewed meshes. See [Advanced Solution Control Options](#) of the User's Guide.
 - ANSYS Icepak 13 supports both the Fluent FLEXlm and ANSYS FLEXlm license managers.
 - ANSYS Icepak workflow in ANSYS Workbench 13 allows the automatic export of ANSYS Icepak geometry from Design Modeler to ANSYS Icepak.
 - ANSYS Icepak data transfer occurs in ANSYS Workbench 13.0 from ANSYS Icepak to Mechanical.

Chapter 11: CFD-Post

This chapter summarizes the new features and incompatibilities in CFD-Post Release 13.0.

11.1. New Features and Enhancements

Volume Rendering

Volume Rendering enables you to visualize field variables throughout the entire domain by varying the transparency and color of the plot as a function of the variable value. For example, you can make realistic images of smoke and analyze how it spreads and how it affects visibility.

Chart

You can now position the chart legend on the chart area, in addition to the surrounding chart area where the axes are found.

Chart axis numbers can now be formatted in either standard or scientific notation.

Comparisons Mode

The automatic detection of the same mesh is much faster.

You can now tell CFD-Post that meshes are either same or different in the Case Comparison panel, which can make the comparison mode much faster.

Turbo Postprocessing

Hub-to-Shroud turbo line points can now be distributed based on mesh density.

Table

Table updates are much faster.

Stereo Viewer

Support is added for graphics display on typical stereo hardware.

ANSYS FLUENT Files

Particle tracks can now be exported from ANSYS FLUENT and post-processed in CFD-Post.

CFD-Post can now read interior face zones from ANSYS FLUENT files. Set this ability from the **Edit > Options > CFD-Post > Files** panel.

CFD-Post now performs more accurate interpolation inside concave polyhedra in ANSYS FLUENT files. This resolves issues with streamlines stopping in the middle of the domain when hitting a concave polyhedron.

CFD-Post can now perform more accurate (cell based) evaluation of user-defined variables.

11.2. Incompatibilities

This section describe procedural changes (actions that have to be done differently in this release to get an outcome available in previous releases) as well as support changes (functionality that is no longer supported) in 13.0 of CFD-Post.

Procedural Changes

When CFX-Solver Manager is opened from ANSYS Workbench, the **Custom Executable** and **Arguments** fields are no longer present on the Solver Tab of the CFX-Solver Manager Define Run dialog (although these fields remain available in the Standalone version of CFX-Solver Manager). You can set those properties via the Solution cell Properties view (this capability is a Beta feature).

Listing Files from a Transient Simulation

In Release 12.0, the files present in the working directory were checked first and all the files with same base name were listed in CFD-Post. If the base name of none of the files matched, CFD-Post listed any appropriate files found in 'autosave/solution-points'.

In Release 13.0, CFD-Post lists any appropriate files found in 'autosave/solution-points'. If no appropriate file is found, the working directory is checked and files with same base name are listed. The new sequence gives higher priority to the files from the chosen DAT/CDAT file.

Chart Legends

A Release 12.0 chart that has its legend on the inside of the chart area may display with the legend in a slightly different position in Release in 13.0.

Display of Mean Molecular Weight Values from FLUENT Files

In Release 12.0, CFD-Post showed incorrect units for Mean Molecular Weight in results files exported from FLUENT; for example, a value that should have appeared as 23.1577 kg/kmol would display as 23157.7 kg/kmol. In Release 13.0, the same case will display a Mean Molecular Weight of 23.1577 kg/mol.

Maximum Temperature Limit of Zero Pressure Polynomials

In Release 12.0, the maximum temperature limit of zero pressure polynomials was 1000 [K] for the CFX-Solver, 3000 [K] in the RULES file, and 5000 [K] for CFX-Pre.

In Release 13.0, maximum temperature limits for zero pressure polynomials are now consistent in RULES, CFX-Pre, and the CFX-Solver (= 1000 K). Temperature limits for table generation are set by CFX-Pre: 5000 K for ideal gases, 1000 K for real gases.

Support Changes

There are no support changes in this release.

Chapter 12: AQWA

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 13.0. Please refer to the product specific documentation for full details of the new features

12.1. ANSYS AQWA

The Following New Features Provide Extended Capabilities in ANSYS AQWA:

- Bending stiffness may now be defined for segments of a composite catenary line when using cable dynamics.
- Non-linear stiffness characteristics may now be defined for segments of a composite catenary line when using cable dynamics.
- Mooring line significant tensions and spectral information at intermediate points (including the anchor) can now be reported when using cable dynamics.
- A new additional stiffness matrix may now be defined to model external stiffness effects (such as mooring line) other than hydrostatic. The stiffness matrices may be associated with connections between a vessel and the ground, or between multiple vessels.
- Spectral results from dynamic frequency based analyses have been extended to include:
 - Zero crossing period for center of gravity motions response spectrum
 - Velocity and acceleration response spectra and zero crossing period for center of gravity
 - Motions, velocity and acceleration response spectra and zero crossing period for specified nodes
- ISO wind spectrum may now be selected.

Hydrodynamic Analysis System Enhancements

New Analysis Settings. New Analysis Settings are available to control your Hydrodynamic analysis.

New Model Components have been added to the Hydrodynamic Analysis system:

- **Connection Points** (under Geometry)
- **Wind and Current Force Coefficients** (under Parts)
- **Cables** and **Connection Stiffness** (under Connections)
- **Catenary Section, Catenary Buoy and Catenary Clump Weight** (Under Catenary Data)

Result Graphs have been enhanced.

- **Hydrodynamic Diffraction Results Graphs.** Diffraction, Froude-Krylov, Diffraction+Froude-Krylov, RAO, Radiation Damping and Added Mass, and Steady Drift diagrams are easier to insert into the analysis. Newly available are 3-D plots for QTF graphs, Splitting Forces, and Bending Moment/Shear Force diagrams.
- New **Hydrodynamic Time Response Results Graphs** (Structure Position, Structure Velocity, Structure Acceleration, Structure Forces, Cable Forces, and Time Step Error) are available.

Parameterization

Many of the quantities shown in the Details panel can be parameterized by selecting the checkbox next to their name. These parameters will then be available to the Workbench project through the Parameter Set bar for additional post processing.

Reporting

You can click on the Report Preview tab in the main window pane to generate a summary of all of the objects in your Outline. The Details information for each object appears as tables in the report. Figures and images appear as specified in the Outline. Charts that appear in the outline are also included. You have a number of options available for saving the report.

Hydrodynamic Time Response Analysis System

The new Hydrodynamic Time Response analysis system allows you to apply ocean environment loading (wind, wave, current) and external boundary conditions (such as moorings) to a structure, and undertake dynamic response analysis in the time domain.

Ocean Environment and Forces Objects

The following new tree objects are available:

- Structure Force
- Regular Wave
- Irregular Wave
- Current
- Wind
- Cable Winch
- Cable Failure

Chapter 13: ASAS

This release of the ASAS products contains all capabilities from previous releases plus many new features and enhancements. The following enhancements are available in release 13.0. Please refer to the product specific documentation for full details of the new features.

13.1. ANSYS ASAS

The following new features are available in Release 13.0 of ANSYS ASAS:

Splinter can be run connected to the Mechanical application.

13.2. ANSYS BEAMCHECK

The following new features are available in Release 13.0 of ANSYS BEAMCHECK:

BEAMCHECK can be used in the Design Assessment system when Mechanical is used for the analysis.

13.3. ANSYS FATJACK

FATJACK provides the ability to assess the fatigue life of welded tubular joints. Joints can be defined with multiple nodes in the typical Y, K, T, X and KT forms, with chord and brace members automatically determined by default. Stress concentration factors can be automatically calculated using a number of built in formulas or manually entered.

When used in conjunction with Mechanical it can be used for deterministic or time history analysis; when used in conjunction with ASAS it can also be used for spectral analysis.

The following new features are available in Release 13.0 of ANSYS FATJACK:

- The [documentation](#) for the code checking module FATJACK is now available from the ANSYS on-line help system.
- FATJACK can be used in the [Design Assessment](#) system when Mechanical is used for the analysis.

13.4. FEMGV

No new features for this release.

