

**ANSYS**

**19.2**

RELEASE

©1971-2018 ANSYS, Inc.  
All Rights Reserved.  
Unauthorized use, distribution or duplication is prohibited.

## **ANSYS ACT 19.2 Migration Notes**

---

## **Copyright and Trademark Information**

© 2018 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

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

## **Disclaimer Notice**

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

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

## **U.S. Government Rights**

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

## **Third-Party Software**

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

Published in the U.S.A.

## Table of Contents

ANSYS ACT 19.2 Migration Notes .....	1
Change to Namespace for IMechanicalApplication .....	1
Change to units returned for ACT field output objects .....	1
Change to index for specifying a constant load value .....	1
Type changes for Mechanical Properties in ACT .....	2
Misspelling corrected for the property JointType in Mechanical .....	3
ACT extensions for AIM support custom boundary conditions for Fluent solvers only.....	3

## ANSYS ACT 19.2 Migration Notes

As improvements are made to ACT APIs and the way that they display and transmit data, great efforts are taken to ensure that changes are backwards-compatible. For your convenience, this section lists 19.2 API changes that might impact your existing extensions so that you can determine if any action is necessary before migrating them.

**Note:** The [ACT Resources page](#) of the ANSYS customer site provides links for stand-alone *ACT Migration Notes* for all 19.x releases under **Help and Support**.

### Change to Namespace for IMechanicalApplication

The interface `IMechanicalApplication` was moved from the namespace `Ansys.ACT.Interfaces.Common` to the namespace `Ansys.ACT.Interfaces.Mechanical`.

### Change to units returned for ACT field output objects

ACT field output objects now return the corresponding units in the active unit system instead of defaulting to Standard MKS units.

### Change to index for specifying a constant load value

As described in the Mechanical release note [Writing Constant Loading Values to the Input File](#), how Mechanical writes constant loading condition values and load magnitudes to the input file has changed. Mechanical now writes load data directly as a constant. As a result, for Static Structural, Steady-State Thermal, Electric, and Thermal-Electric analyses, the index for specifying a constant load value via the `SetDiscreteValue` method is changed from **1** to **0**.

For instance, assume in a Static Structural analysis, you want to set the magnitude of a pressure load to 1000 Pa. Before release 19.2, you wrote the Python command as:

```
pressure.Magnitude.Output.SetDiscreteValue(1, Quantity("1000 [Pa]"))
```

Now you write the Python command as:

```
pressure.Magnitude.Output.SetDiscreteValue(0, Quantity("1000 [Pa]"))
```

## Type changes for Mechanical Properties in ACT

The following Mechanical properties in ACT had their types changed from Double to the new type indicated:

Object	Property	New Type
BodyInteractions	LimitingTimestepVelocity	Velocity Quantity
ContactRegion	ElectricConductanceValue	Electric Conductance Per Unit Area Quantity
	MeanPitchDiameter	Length Quantity
	PitchDistance	Length Quantity
	ThermalConductanceValue	Heat Transfer Coefficient Quantity
Joint	RXMinimum	Angle Quantity
	RXMaximum	Angle Quantity
	RYMinimum	Angle Quantity
	RYMaximum	Angle Quantity
	RZMinimum	Angle Quantity
	RZMaximum	Angle Quantity
	XMinimum	Length Quantity
	XMaximum	Length Quantity
	YMinimum	Length Quantity
	YMaximum	Length Quantity
	ZMinimum	Length Quantity
	ZMaximum	Length Quantity

## Misspelling corrected for the property JointType in Mechanical

For the property **JointType**, **Gneral** rather than **General** was in use. This typo has now been corrected:

```
Joint1.Type =JointType.General
```

## ACT extensions for AIM support custom boundary conditions for Fluent solvers only

While AIM extensions support custom boundary conditions for Fluent solvers, they do not support them for non-Fluent solvers. If you need to use custom boundary conditions alongside the MAPDL solver, you must use ANSYS Mechanical instead of ANSYS AIM.

**Note:** ACT has superseded the ANSYS Workbench Software Development Kit (SDK) and External Connection Add-In as the best-in-class tool set for customizing ANSYS products. Support for the SDK and External Connection Add-in has ended as of 19.0. If you have used these deprecated tools for Workbench customizations, see the [ANSYS SDK and External Connection Add-in Migration Guide](#) on the [ACT Resources page](#) of the ANSYS customer site for migration information.