ANSYS ACT 19.2 Migration Notes

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 Note*s 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.