



ANSYS ACT 2019 R1 Known Issues and Limitations

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.

Contents

ANSYS ACT 2019 R1 Known Issues and Limitations	1
After installation, the ACT Start Page sometimes appears blank.....	1
VPN conflicts can arise if IP information is retained	1
ACT for Fluent is supported on Windows platform only	1
ACT Debugger is supported on Windows platform only	1
Limited ACT localization support.....	1
ACT is unable to create a chart from ANSYS Mechanical	2
Scripting for ANSYS Mechanical solution combinations.....	2
Limitations on ACT postprocessing of ANSYS Mechanical results.....	2
Scoping for custom results defined in ACT extensions not supported.....	2
Compressed result file not supported	2
A node merge action on the mesh is unsupported	2
Results availability for certain models	2
Coordinate system for custom results returned for beam element	3
Archived projects containing solved ACT custom results.....	3
Default location of ACT extensions for SpaceClaim	3
Limitations Unique to Linux.....	3
Some property controls are not supported in ANSYS Mechanical	3

Starting the ACT Start Page can cause an unexpected shutdown	4
Some Linux versions contain a JPEG library that conflicts with the ANSYS package.....	4

ANSYS ACT 2019 R1 Known Issues and Limitations

This document lists known issues and limitations in the ACT 2019 R1 release.

After installation, the ACT Start Page sometimes appears blank

If after installing ANSYS products, you see that the **ACT Start Page** is blank, right-click inside the blank window and select **Reload**.

VPN conflicts can arise if IP information is retained

When using a VPN, IP information can be retained, even after turning off the VPN and rebooting the computer. The workaround is to release the IP configuration using the following script:

```
echo Releasing IP
ipconfig /release > NUL
echo Acquiring new IP
ipconfig /renew > NUL
```

ACT for Fluent is supported on Windows platform only

When using ACT for Fluent customization, ACT is supported only when Fluent is running on the Windows platform.

ACT Debugger is supported on Windows platform only

The **ACT Debugger** is supported only on the Windows platform from the **Project** page in ANSYS Workbench and from DesignModeler and Mechanical.

Limited ACT localization support

Localization of ACT is limited to the languages currently supported in ANSYS Workbench. This limitation does not apply to the ability to manage various languages within the extension. For example, the property names created by an extension do not have to be in the same language as the current activated language in Workbench.

There is no mechanism to integrate localization for the property names defined by an extension. To manage different languages for your property names, you must develop localization yourself. Both

regional settings based on the "." or the "," decimal symbol are available. However, the implementation of the extension should use the "." symbol for any value defined at the XML or IronPython level.

ACT is unable to create a chart from ANSYS Mechanical

When using ACT to create a figure from the chart API, the following error prevents the graphics display in the Mechanical window:

```
Object reference not set to an instance of an object.
```

As a workaround, add the following code to your script to create an empty window in which the chart can display:

```
import clr
clr.AddReference("Ans.UI.Toolkit")
clr.AddReference("Ans.UI.Toolkit.Base")
import Ansys.UI.Toolkit
if Ansys.UI.Toolkit.Window.MainWindow == None:
Ansys.UI.Toolkit.Window.MainWindow = Ansys.UI.Toolkit.Window()
```

Scripting for ANSYS Mechanical solution combinations

To run an entire script involving solution combinations at once, you must add all of the desired base cases and load combinations upfront, before modifying their properties.

Limitations on ACT postprocessing of ANSYS Mechanical results

If you use ACT for postprocessing Mechanical results, you should be aware of the following limitations:

Scoping for custom results defined in ACT extensions not supported

Custom results do not support using a geometric path as scoping. You can only use a selection of nodes and elements as scoping.

Compressed result file not supported

The ACT postprocessing API does not support the compressed result file for Mechanical, which is created by the Mechanical APDL command `/FCOMP`.

A node merge action on the mesh is unsupported

The ACT postprocessing API does not support a node merge action on the mesh.

Results availability for certain models

Full results availability through the ACT postprocessing API is not guaranteed for Mechanical models containing:

- Bolt pretension loads
- Beam connections
- Link elements

- Pipe elements
- Elbow elements

Method `GetElementValues ([Ids],boolean)` does not support shell and layer elements

The method `GetElementValues ([Ids],boolean)`, which takes an array of element IDs as an input parameter, does not support shell and layer elements. For shell and layer elements, you must loop on the element IDs and use the method `GetElementValues (id)`, which takes only one element ID as in input. For example:

```
for id in collector.Ids:  
    values=rs.GetElementValues(id)  
    collector.SetValues(id,values)
```

Coordinate system for custom results returned for beam element

Custom results returned for the beam element are based on the global coordinate system and not the local coordinate system.

Archived projects containing solved ACT custom results

When you restore an archived project containing previously solved ACT custom results, you might encounter a solve failure after modifying the model data and updating the analysis in ANSYS Mechanical. As a workaround:

1. Right-click the **Solution** object in the Mechanical tree and select **Clear Generated Data** from the context menu.
2. From the **Solution** object, access the solver files and delete all files in the directory.
3. Re-solve the model.

Default location of ACT extensions for SpaceClaim

The default location in which SpaceClaim is to look for ACT extensions is `%ANSYSversion_DIR%\scdm\Addins`. However, SpaceClaim is not currently recognizing this location. Workarounds include either installing extensions in `%APPDATA%\Ansys\v192\ACT\extensions` or using the gear icon on the graphic-based **Extension Manager** accessed from the **ACT Start Page** to add the folder for the default location or the folder to which you installed the extension.

Limitations Unique to Linux

If you are running on Linux, you should be aware of the following limitations:

Some property controls are not supported in ANSYS Mechanical

The following property controls are not supported on Linux in Mechanical:

- `FileOpen`
- `FolderOpen`
- `PropertyTable`

These controls are implemented using the ANSYS UI Toolkit, which is currently not supported on Linux when executed within Mechanical.

Starting the ACT Start Page can cause an unexpected shutdown

When you start the **ACT Start Page** on Linux, an unexpected shutdown can occur. On some Linux operating system variants such as Red Hat, removing the package `totem-mozplugin` resolves the issue:

```
yum remove totem-mozplugin
```

Some Linux versions contain a JPEG library that conflicts with the ANSYS package

In some Linux versions, a JPEG library conflicts with the ANSYS package. When using ACT wizards with JPEG images, this conflict can produce a SIGSEV error and an application crash. If this occurs, you can convert your images to other file formats such as GIF or PNG.

Graphics API issues in ANSYS DesignModeler and Mechanical when no extensions are loaded

There are some limitations on the Graphics API from the **ACT Console** in ANSYS DesignModeler and Mechanical when no extensions are loaded. For instance, the Factory2D does not work. Therefore, you should load one or more extensions before using the Graphics API from the **ACT Console**.