

Knowledge Base

Information



Abaqus/Viewer plugin to write nodes and elements displayed in the viewport to an input file

Portfolio / Domain: SIMULIA Abaqus Unified FEA / SIMULIA Abaqus Unified FEA
Product: SIMULIA Abaqus/CAE

QA Article: QA00000023036e
Applicable Level: 6.1 (6.1)
Last Update Date: 04.11.2020
Rating: 4.0
Views: 735

QUESTION I would like to write an input file for just the part of my model displayed in the Visualization module viewport. Is it possible to do this operation?

ANSWER (The following applies to Abaqus 6.11 and higher)
An Abaqus/Viewer plug-in application for this purpose is attached below.
This utility will create an input file from the nodes and elements currently shown in the viewport of the Visualization module of Abaqus/CAE. This can be useful for creating an arbitrary subset model after the entire geometry has been meshed and stored in an ODB file. Note that only the definitions of the nodes and elements will be written - the file will not include any step definitions or other data necessary to execute the model.

There are two versions of the plugin available: one that writes the coordinates of nodes of the initial configuration (file: inputFile_initial_config_plugin.py and folder: abq_InputFile_initial_config) and one that writes the coordinates of the nodes in the final deformed configuration - i.e., the last frame of the ODB file (file: inputFile_deformed_config_plugin.py and folder: abq_InputFile_deformed_config).

Installation

To install the plug-in, save the attached archive file to one of the following directories:
abaqus_dir\code\python\lib\abaqus_plugins where *abaqus_dir* is the Abaqus parent directory
abaqus_dir\abaqus_plugins where *abaqus_dir* is the Abaqus parent directory
home_dir\abaqus_plugins where *home_dir* is your home directory
current_dir\abaqus_plugins where *current_dir* is the current directory

Note that if the abaqus_plugins directory does not exist in the desired path, it must be created. The *plugin_dir* directory can also be used, where *plugin_dir* is a directory specified in the abaqus_v6.env file by the environment variable **plugin_central_dir**. You can store plug-ins in a central location that can be accessed by all users at your site if the directory to which **plugin_central_dir** refers is mounted on a file system that all users can access. For example, `plugin_central_dir = r'\\fileServer\sharedDirectory'`

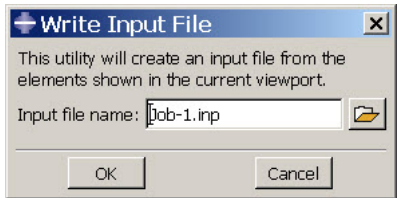
On Windows platforms, right click on the archive file and select **WinZip** → **Extract to here**. On Linux platforms, type **unzip writeInputFile.zip** at the command prompt. Folders named abq_InputFile_initial_config and abq_InputFile_deformed_config along with files named inputFile_initial_config_plugin.py and inputFile_deformed_config_plugin.py will be extracted.

Note that the plug-in will not function properly if this procedure is not followed.

Usage

Plug-in instructions:

1. Start Abaqus/Viewer or Abaqus/CAE Visualization module and open an ODB.
2. Use the **Display Group** tools to display just the elements of interest.
3. Invoke the plug-in by selecting **Plug-ins** → **Write Input File - Initial...** or **Plug-ins** → **Write Input File - Deformed...**to receive the following dialog:



4. Enter an input file name and click **OK**.

Disclaimer

The attachments to this article are subject to certain usage conditions. Please [click here](#) for details

KEYWORDS 5010

ATTACHMENT [answer_5010_figure1.jpg](#) [write_InputFile.zip](#)

MY FAVORITE CONTENT

SUBSCRIBE TO
CHANGES

RATING On a scale of 1-5, how would you rate the technical content of the article?

Please rate this article...

LET US KNOW
IF THIS ARTICLE
NEEDS TO BE
ENHANCED

UNCLEAR MISSING INFO DUPLICATE OUT OF DATE ERROR DETECTED

[See Comments \(1\)](#)

MY FAVORITE CONTENT