

Knowledge Base

Information



Abaqus/CAE plug-in for creating exterior node sets and surfaces

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

QA Article: QA00000023100e
Applicable Level: 6.1 (6.1)
Last Update Date: 07.10.2020
Rating: Not Rated
Views: 698

QUESTION

I would like to create two things: a node set that contains all the nodes on the exterior of a part, and a surface on the exterior of the part. Is there a quick way to do this?

ANSWER

(The following applies to Abaqus 6.11 and higher)

An Abaqus/CAE plug-in for this purpose is attached below. This utility allows you create a node set containing all nodes on the exterior of a part and a surface using the elements on the part's exterior.

Installation

To install the plug-in, save the attached archive file to one of the following directories:

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 exteriorMesh.zip** at the command prompt. Folders named *abq_ExteriorMesh* and a file named *exteriorMesh_plugin.py* will be extracted.

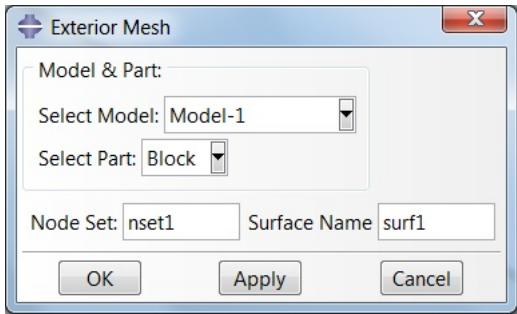
Note that the plug-in will not function properly if this procedure is not followed.

Usage

- 1. Load the desired model, either by opening an existing Abaqus/CAE model database or importing an input file.

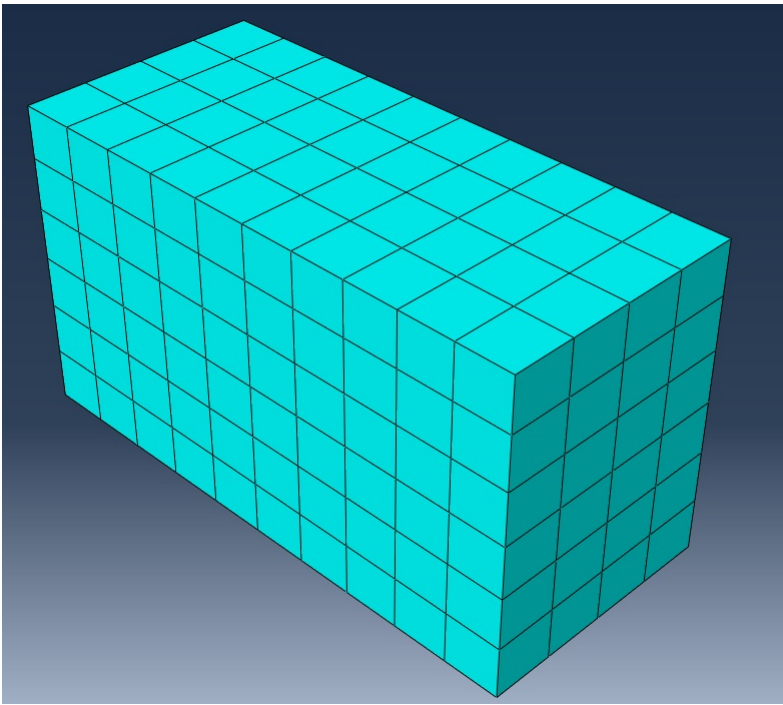


- 2. Click the **Exterior Mesh** icon to invoke the **Exterior Mesh** window:

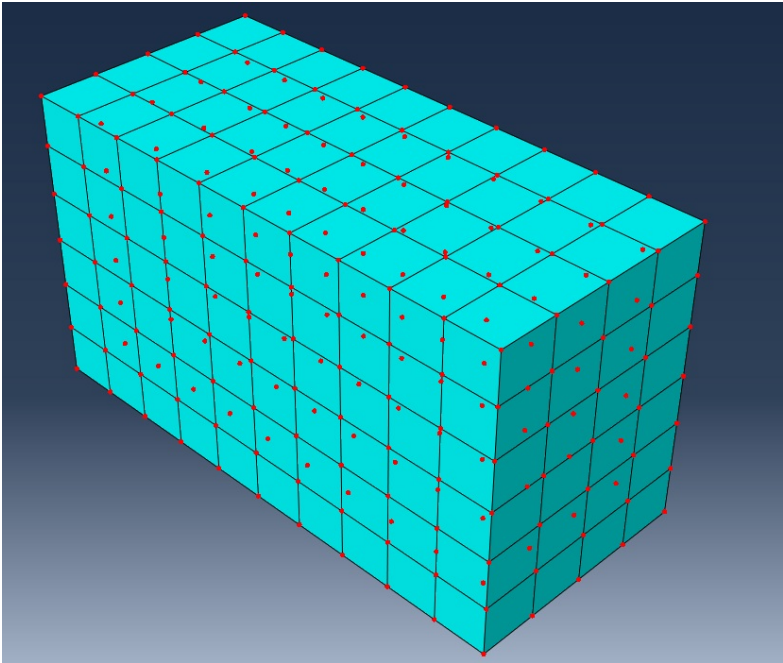


- 3. All the models in the model database will be listed in the **Select Model** drop-down menu and the corresponding parts will be listed in the **Select Part** drop-down menu.
 - 4. Enter a **Node Set** name and **Surface Name** in the respective text boxes.
 - 5. Select **Apply** or **OK** to create the node and surface.
- For example, a meshed part is shown below:

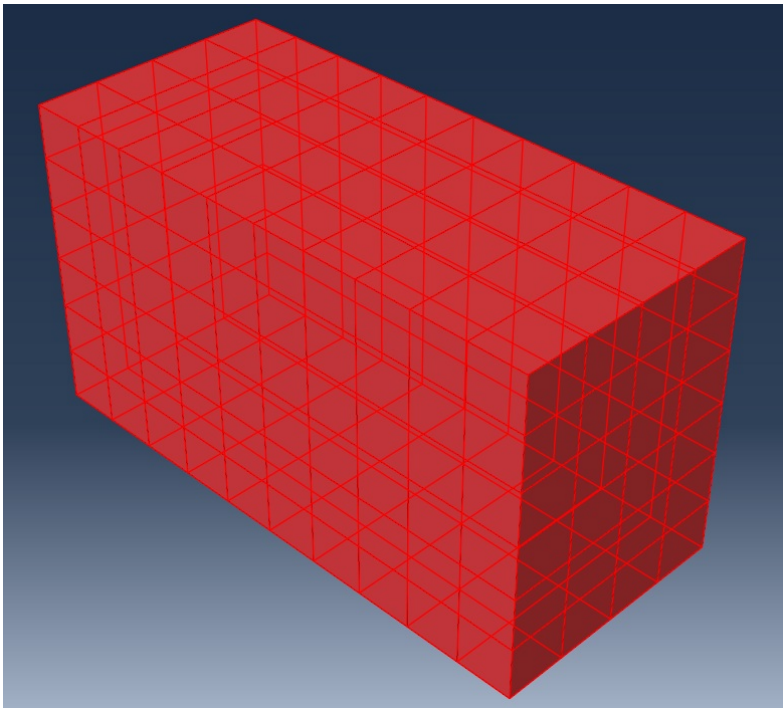
MY FAVORITE CONTENT



After the script is executed, the node set created will have the following nodes on the exterior of the body:



and the surface created will look like:



Notes

- Currently this utility only supports meshed 3D parts.

Disclaimer

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

KEYWORDS

free surface, freesurface, free, outside, outer, exterior mesh, node set, surface, script, plug, aut

ATTACHMENT

answer_5041_fig1.png	answer_5041_fig4.png	exteriorMesh.zip	answer_5041_fig5.png
answer_5041_fig3.png	answer_5041_fig2.png		

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