

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: QA0000023100e
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.

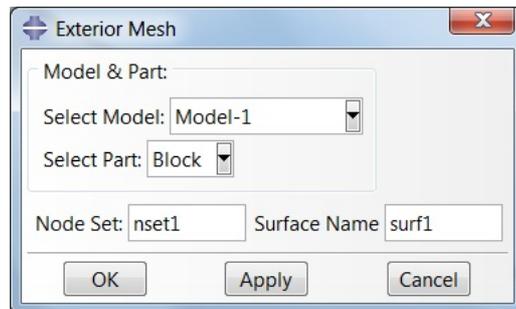
Note that the plug-in will not function properly if this procedure is not followed.

Usage

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

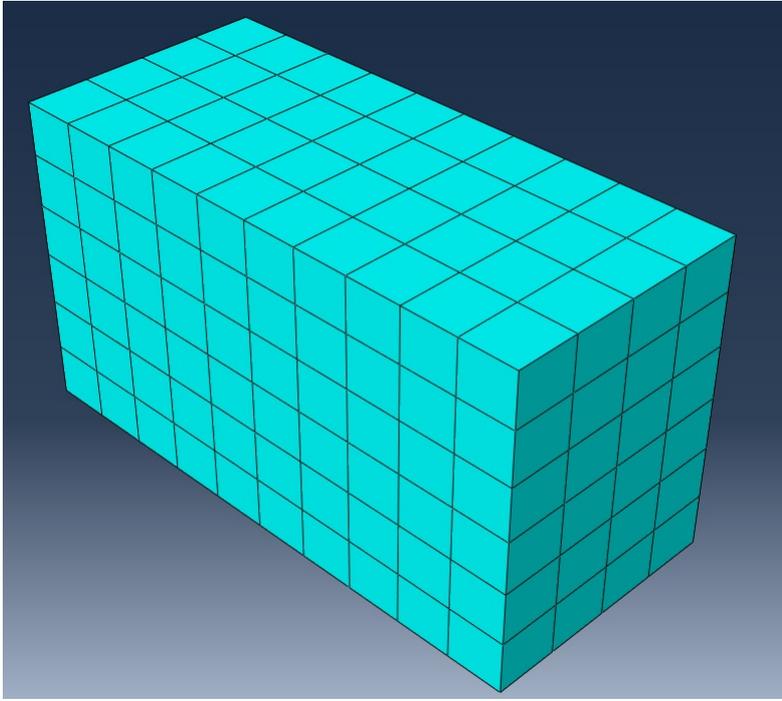


2. Click the **Exterior Mesh** icon to invoke the **Exterior Mesh** window:

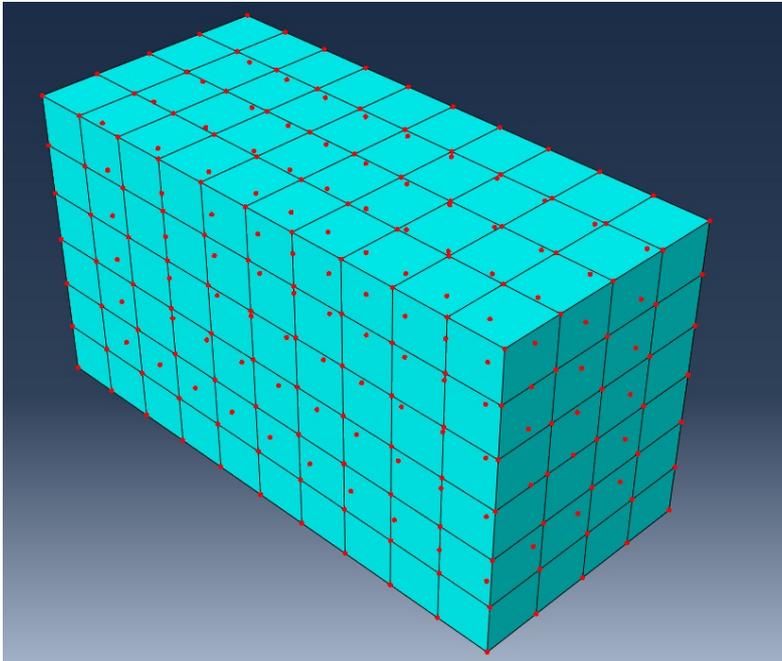


3. All the models in the model database will be listed in the **Select Model** drop-down menu and the corresponding parts will be listed in the **Select Part** drop-down menu.
4. Enter a **Node Set** name and **Surface Name** in the respective text boxes.
5. Select **Apply** or **OK** to create the node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

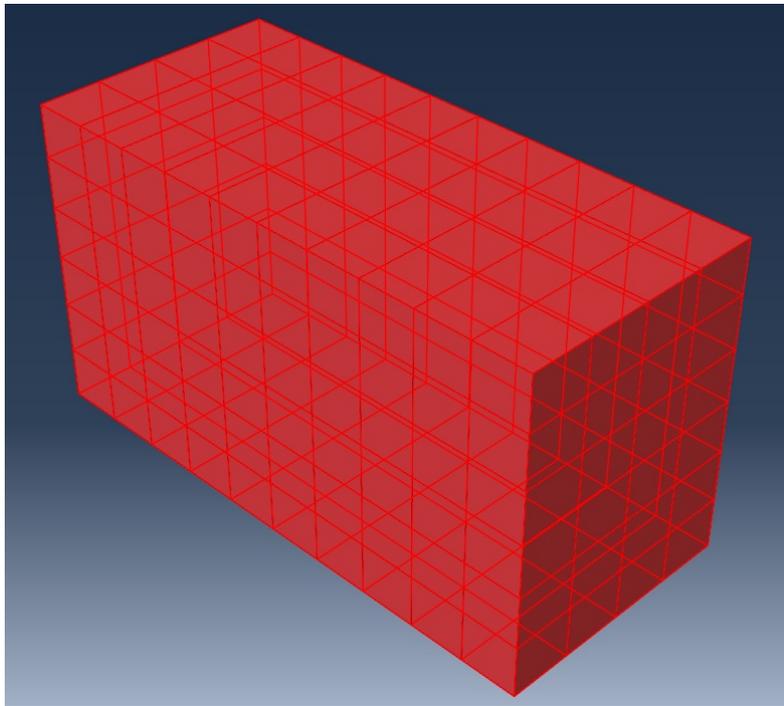
For example, a meshed part is shown below:



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