

Knowledge Base

Information



Plug-in utility for extracting two-dimensional parts from meshes in Abaqus/CAE

Portfolio / Domain: SIMULIA Abaqus Unified FEA / SIMULIA Abaqus Unified FEA
Product: n/a

QA Article: QA00000008945e
Applicable Level: 6.6
Last Update Date: 31 Jul 2020
Rating: Not Rated
Views: 555

QUESTION

Can I create a two-dimensional (2D) geometric part from the perimeter of a 2D mesh in Abaqus/CAE?
Can I create a 2D orphan mesh part by passing a section through a 3D mesh in Abaqus/CAE?

ANSWER

(The following applies to Versions 6.6 and higher.)

An Abaqus/CAE plug-in utility that performs these operations is attached to this Answer. The plug-in provides a graphical interface for the following Python commands:

- Part2DGeomFrom2DMesh to extract the perimeter of a 2D mesh and automatically create a 2D geometric part.
- PartFromSection3DMeshByPlane creates a 2D orphan mesh part by cutting an existing 3D orphan mesh part with a plane.

To install the plug-in, save the attached files 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 next time Abaqus/CAE is started, a menu item named **Tools** will be available in the **Plug-ins** pull down menu. The **Tools** option has two suboptions, **Extract Perimeter** and **Cut Mesh**.

Extract Perimeter Tool

To use the **Extract Perimeter** tool select **Plug-ins** → **Tools** → **Extract Perimeter**. The utility may be invoked from any module except **Visualization**. The interface is shown in Figure 1:

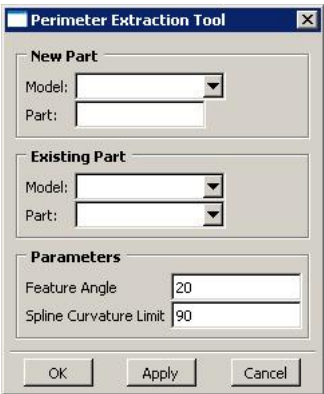


Figure 1: Perimeter Extraction Tool Interface

In the **New Part** panel, specify the model that will contain the geometric part to be created and the name of the new part. In the **Existing Part** panel, specify the model that contains the existing mesh part and the part name. The **Model** pull down lists in both panels will be populated with the names of all the models that are active in the current Abaqus/CAE session. This allows parts existing in one model to be used as the basis for parts in other models.

Two options are available in the **Parameters** panel. Specify the **Feature Angle** as the angle (in degrees) between line segments that triggers a break in the geometry. The default value is 20. Specify the **Spline Curvature Limit** as the traversal angle (in degrees) of the spline that triggers a break in the geometry. The default value is 90.

Cut Mesh Tool

To use the **Cut Mesh** tool select **Plug-ins** → **Tools** → **Cut Mesh**. The utility may be invoked from any module except **Visualization**. The interface is shown in Figure 2:

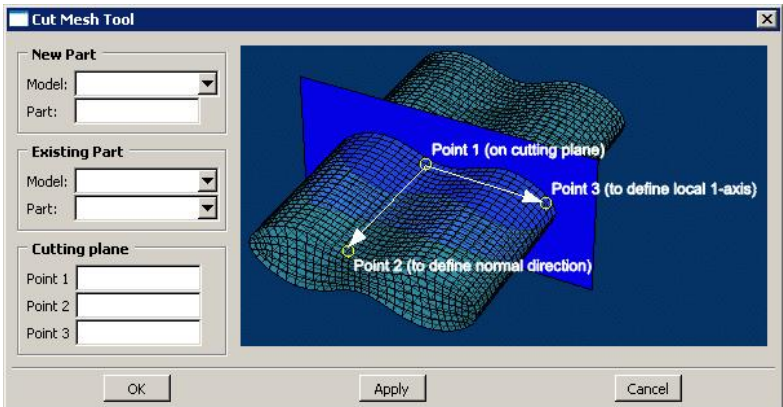


Figure 2: Cut Mesh Tool Interface

The **New Part** and **Existing Part** panels should be completed as described above. In the **Cutting plane** panel, specify the three points that define the cutting plane. The definitions of **Points 1, 2, and 3** are shown in Figure 2. Note that the resulting mesh may contain distorted elements depending on the angle of the cut relative to the underlying mesh. In this case the **Extract Perimeter** tool can be used to create a geometric part for subsequent remeshing.

Disclaimer

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

KEYWORDS plugin, plug-in, customization, gui, python, scripting, 2414

ATTACHMENT	meshTools_plugin.py	meshToolsIcons.py	Answer_2414_Figure1.jpg	Answer_2414_Figure2.jpg
------------	-------------------------------------	-----------------------------------	---	---

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