Knowledge Base

Abaqus/CAE plug-in to generate a spur or helical gear

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

Information

QA Article: QA(
Applicable Level: Aba Gol
Last Update Date: 07.
Rating:
Views: 373

QUESTION

I would like to generate a gear flexible body model for Simpack using Abaqus. Is there an Abaqus/CAE plug-in to generate a spur or a helical gear?

ANSWER

Simpack can perform rigid or flexible gear analysis. In order to perform flexible gear analysis, the Abaqus finite element model is required to generate a flexible body for the gear. An Abaqus/CAE plug-in for this purpose is attached below. The plug-in requires gear parameters similar to those required for the Simpack gear wheel primitive. Currently, micro-geometry, bevel angle, and backlash are not supported in this plug-in.

Installation

Native CAE plug-ins are included with the Abaqus/CAE installation. External plugins (those installed after installation of Abaqus/CAE) should not be put inside the Abaqus/CAE installation. To install the plug-in, save the attached archive file to one of the following directories:

```
home_dir\abaqus_plugins
current_dir\abaqus_plugins
```

where *home_dir* is your home directory and where *current_dir* is the current directory.

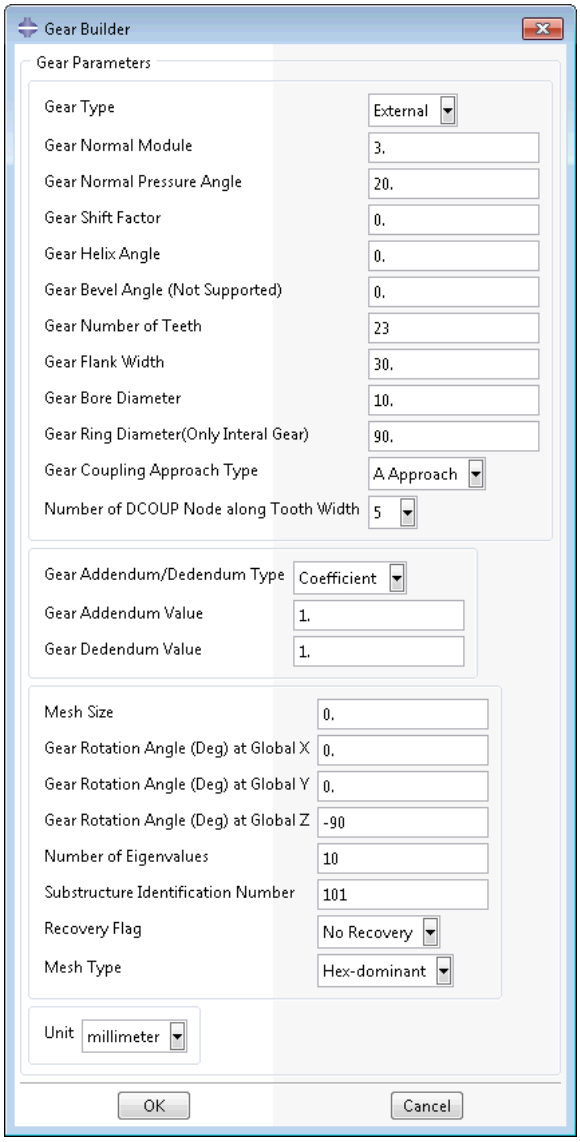
The *plugin_dir* directory can also be used, where *plugin_dir* is a directory specified by the environment variable **plugin_central_dir**. This parameter is used to define a specific directory where plugins are stored. This is typically a central location accessed by all users at your site if the directory is mounted on a file system that all users can access. **plugin_central_dir** can be defined in the abaqus_v6.env file or the Abaqus solver custom_v6.env file. For example,

```
plugin_central_dir = '\\\\fileServer\\share\\AbaqusPlugins'
```

On Windows platforms, right click on the archive file and select **WinZip** → **Extract to here**. On Linux platforms, type **unzip GEAR.zip** at the command prompt. Folders named abq_gear and a file named gear_plugin.py will be extracted. Note that the plug-in will not function properly if this procedure is not followed.

Usage

The plug-in can be accessed from the Part, Mesh, Assembly, or Interaction module of Abaqus/CAE. Select **Plug-ins** → **Tools** → **Gear ...** to invoke the following dialog:



The current plug-in supports external and internal gears for spur and helical types. The bevel gear type is not supported. Since this Abaqus/CAE plug-in is generating Abaqus finite element model for Simpack flexible gear, there are two ways to create coupling for each of the gear teeth. For detailed information please refer to the Simpack documentation (D.20.5.25 Gear Wheel in Simpack Assistant 2017). The two ways to create coupling can be found in the following images:

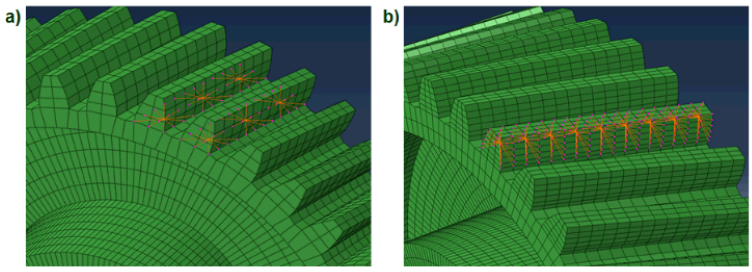


Figure D.20.28: Tooth node representation of a) modeling method 'A' and b) modeling method 'B' using Abaqus

The number of distributing coupling reference nodes along a gear width can be specified. By default, 5 nodes are defined.

If the meshsize is specified as 0.0, then the plug-in creates the mesh automatically. Otherwise, the meshsize should be specified as floating point number.

The current plug-in generates two procedures by default: frequency extraction and substructure generation. The necessary values for the number of eigenvalues for the frequency extraction step and substructure identification number for substructure generation step should be specified. In substructure generation, all modes extracted in the frequency extraction are selected. If the recovery flag is assigned as "Recovery", the keyword "*ELEMENT RECOVERY MATRIX, POSITION=AVERAGED AT NODES" is generated for stress and strain output variables. This allows you to visualize the stress and strain contour in Simpack Post.

Currently, only millimeter length units are supported.

The plug-in generates two Abaqus input files by default: GEAR01SUBSTR.inp and GEAR01SUBSTR-SimpackFlex.inp. The first input file is the substructure generation file and the second input file is for the substructure generation file to create the Simpack Flex Body (*.fbi). The result files from the second input file can be used with the Simpack FBI file generator script.

KEYWORDS Abaqus, Gear, Spur, Helical, Simpack, Flexible Body

ATTACHMENT 48128_Fig2.png 48128_Fig1.png GEAR.zip

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