



## Using Abaqus/CAE to compute shape variations

A transitional capability for computing shape variations is provided by the internal Abaqus Scripting Interface command `_computeShapeVariations()`.

- [Defining a shape variation](#)

Using the command requires some familiarity with the Abaqus Scripting Interface; specifically, the user must understand the Abaqus object model (see [The Abaqus object model](#)) and know how to access the `rootAssembly` and `partInstance` objects.

### Defining a shape variation

1. Create and mesh the model in Abaqus/CAE. Output the corresponding input file by selecting **Job > Create** and **Job > Write Input** from the main menu bar in the Job module. (In the discussion that follows it is assumed that the model is named `Model-1`, the part is named `Part-1`, and the part instance is named `Part-1-1`).
2. Select **Model > Copy Model** from the main menu bar to copy `Model-1` to `Model-2`, for example. Select `Model-2` from the Model list located in the context bar. This model will be used in subsequent steps to compute the shape variation.
3. In the Part module select the part for which a shape variation must be calculated from the Part list located under the context bar. Select **Feature > Edit** from the main menu bar to edit the associated sketch. Select **Add > Dimension** and **Edit > Dimension** to change the design parameter. End the edit of the sketch, and indicate that the geometry should be regenerated automatically.

Editing the sketch will cause the mesh of `Model-2` to be deleted.

4. Use the Abaqus/CAE command line or select **File > Run Script** to execute the commands listed below. The `_computeShapeVariations()` command is accessed from the `rootAssembly` of either model and requires as input the original part instance, the modified part instance, and the name of the file to which the data lines of the shape variation option will be written. The `.inp` extension will be appended automatically to the specified file name.

The following sequence of commands is appropriate for the creation of a shape variable associated with a parameter  $h$ :

```
ra1 = mdb.models['Model-1'].rootAssembly
ra2 = mdb.models['Model-2'].rootAssembly
i1 = ra1.instances['Part-1-1']
i2 = ra2.instances['Part-1-1']
ra1._computeShapeVariations(originalInstance=i1,
                             modifiedInstance=i2,
                             fileName='shape_h')
```

5. To compute the shape variation, the mesh of `Model-1` is mapped to the changed geometry of `Model-2` and then smoothed. Examine the mapped mesh of `Model-2` to verify that the mesh was mapped as expected. (In general, any changes to the geometry should be small—around 1%—so as to avoid difficulties with the mapping of the mesh).

The shape variation is calculated simply by subtracting the initial node positions from the node positions calculated after mapping and smoothing the mesh to the changed geometry.

6. To use the shape variation data, copy to the input file for your analysis the data written by the `_computeShapeVariations()` command to the `shape_h.inp` file.

7. Verify the correctness of the data by running your analysis and using the Visualization module in Abaqus/CAE to view the shape variation.