12.4.7 Evaluating hyperelastic and viscoelastic material behavior

Abaqus/CAE provides a convenient **Evaluate** option that allows you to view the behavior predicted by a hyperelastic or viscoelastic material and that allows you to choose a suitable material formulation. You can evaluate any hyperelastic material, but a viscoelastic material can be evaluated and viewed only if it is defined in the time domain and includes hyperelastic and/or elastic material data. If your material definition includes viscoelastic data defined in the frequency domain, you cannot evaluate its viscoelastic material behavior in Abaqus/CAE, but its material evaluation data are written to the data (`.dat`) file. The **Evaluate** option prompts Abaqus/CAE to perform one or more standard tests using an existing material. (For information on standard tests for hyperelastic and viscoelastic materials, see "Hyperelasticity," Section 22.5 of the Abaqus Analysis User's Guide, and "Linear viscoelasticity," Section 22.7 of the Abaqus Analysis User's Guide, respectively.) Once the standard tests are completed, Abaqus/CAE enters the Visualization module and displays the test results in new viewports as *X*–*Y* plots. (For more information on *X*–*Y* plots, see Chapter 47, "X–Y plotting.") Abaqus/CAE also displays an informational dialog box containing the stability limits and coefficients for each hyperelastic strain energy potential and the viscoelastic material parameters for the viscoelastic response. The information from the evaluation is saved in the *`material_name_i`*`.dat` file, where *i* starts at `1` and is incremented for subsequent evaluations of the same material. You can review the evaluation results and adjust the material definition as necessary.

To initiate the evaluation procedure, select **Material**⟶**Evaluate**⟶***material name*** from the main menu bar. Alternatively, you can select the material of interest in the **Material Manager** and then click **Evaluate**. The **Evaluate Material** dialog box appears in which you can specify how you want Abaqus/CAE to perform the standard tests. For detailed instructions on evaluating hyperelastic material behavior, see "Displaying *X*–*Y* plots of hyperelastic material behavior," Section 12.7.7. For detailed instructions on evaluating viscoelastic material behavior, see "Displaying *X*–*Y* plots of viscoelastic material behavior," Section 12.7.8.

> **Note:** The material evaluation procedure generates jobs with the same names as the materials; therefore, these material names must adhere to the same rules as job names (see "Using basic dialog box components," Section 3.2.1, for more information on naming objects).

The **Evaluate** option is particularly useful in the following scenarios:

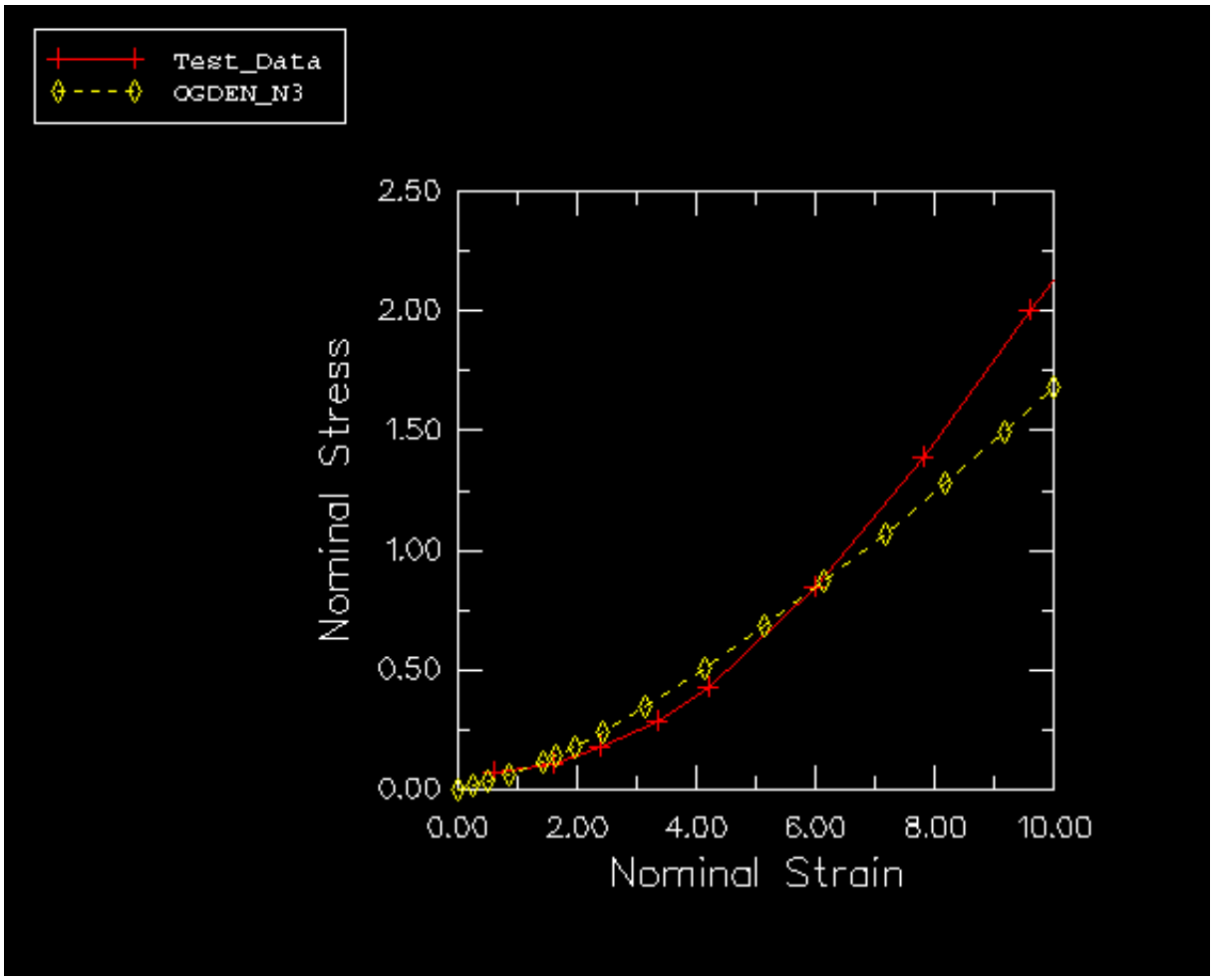
Comparing test data with the behavior predicted by a particular strain energy potential

When you define a hyperelastic material using experimental data, you also specify the strain energy potential that you want to apply to the data. Abaqus uses the experimental data to calculate the coefficients necessary for the specified strain energy potential. However, it is important to verify that an acceptable correlation exists between the behavior predicted by the material definition and the experimental data.

You can use the **Evaluate** option to calculate the material response based on the experimental data using the strain energy potential that you have specified in the material definition. When the tests are complete, Abaqus/CAE enters the Visualization module and displays *X*–*Y* plots of the test results. Each plot includes the experimental data and a curve for each evaluated strain energy potential. Abaqus/CAE also opens a dialog box containing the stability limits and coefficients for each strain energy potential.

For example, the *X*–*Y* plot in Figure 12–13 shows the results of a planar test using the Ogden N=3 strain energy potential.

Figure 12–13 Results of a planar test.



In addition, the following information is reported to the data (`.dat`) file:

- The coefficients calculated for the strain energy potential.
- Any material instabilities that were detected during the tests.

The path to the data (`.dat`) file appears in the message area of the Abaqus/CAE main window once the analysis has completed successfully.

Evaluating multiple strain energy potentials

If you are defining a hyperelastic material using experimental data and you are unsure which strain energy potential to specify, you can select **Unknown** from the **Strain energy potential** list in the material editor. You can then use the **Evaluate** option to perform standard tests with the experimental data using multiple strain energy potentials.

When the tests are complete, Abaqus/CAE enters the Visualization module and displays an *X*–*Y* plot for each test and a dialog box containing the stability limits and coefficients for each strain energy potential. Each plot includes the experimental data and a curve for each evaluated strain energy potential. You can visually compare the strain energy potential curves and the experimental data curve and select the strain energy potential that provides the best fit.

Once you have determined which strain energy potential provides the best fit with the experimental

data, you must return to the material editor in the Property module and change the **Strain energy potential** selection from **Unknown** to the strain energy potential that you have chosen.

Viewing behavior predicted by coefficients

If you have acquired coefficients for a particular strain energy potential (either by evaluating one or more hyperelastic strain energy potentials, as described above, or from another source), you may want to verify that the behavior predicted by the strain energy potential acceptably matches your experimental data or meets other criteria.

You can use the **Evaluate** option to plot a curve of the strain energy potential using the coefficients you provided in the material definition. If the material definition also includes experimental data, a curve for that data also appears in the plot.

Viewing response curves for viscoelastic materials

If you have shear or volumetric test results, you may want to verify that the creep and relaxation behavior predicted by Abaqus acceptably matches your experimental data or meets other criteria. Likewise, if you have frequency data, you may want to verify that the predicted storage and loss components of the shear and bulk moduli match your data.

You can use the **Evaluate** option to plot curves using the coefficients you provided in the material definition. If the material definition includes experimental data, curves for those data also appear in the plots. The types of curves produced depend on the material definition. For viscoelastic materials defined using a Prony series, creep test data, or relaxation test data for time, you can produce creep and relaxation plots versus time. For viscoelastic materials defined using frequency data for time, you can produce plots of the storage and loss components of the shear and bulk moduli versus a logarithmic scale of frequencies.

Adjusting material data

If you are unsatisfied with the fit between the test data and the behavior predicted by the material, you can return to the Property module and adjust the test data and then evaluate the material again. You can repeat this process until you are satisfied with the material behavior. In some cases it may be possible to use this approach to optimize the coefficient values included in a hyperelastic material definition. For more information, see “Improving the accuracy and stability of the test data fit,” in “Hyperelastic behavior of rubberlike materials,” Section 22.5.1 of the Abaqus Analysis User's Guide.

For detailed instructions on evaluating materials, see the following sections :

- “Displaying *X–Y* plots of hyperelastic material behavior,” Section 12.7.7
- “Displaying *X–Y* plots of viscoelastic material behavior,” Section 12.7.8

For more information on the strain energy potentials available in Abaqus, see “Strain energy potentials,” in “Hyperelastic behavior of rubberlike materials,” Section 22.5.1 of the Abaqus Analysis User's Guide.