Analyzing Viscoelastic Materials

Mechanical solutions from ANSYS have convenient tools for calculating deformation of materials in which stiffness changes as a function of loading, time and temperature.

By Sheldon Imaoka, Technical Support Engineer, ANSYS, Inc.

Viscoelastic materials have an interesting mix of material properties that exhibit viscous behavior (like the gradual deformation of molasses) as well as elasticity (like a rubber band that stretches instantaneously and quickly returns to its original state once a load is removed). The clearest way to visualize the behavior of a material containing both elastic and viscous components is to think of a spring (exerting forces to return to its unstressed state) in series with a dashpot (a damper that resists sudden motion, similar to the pneumatic cylinder that prevents a storm door from slamming shut). With these properties, the stresses of a viscoelastic material gradually relax over time when a constant displacement is applied. Conversely, under a constant applied force, elastic strains continue to accumulate the more it is deformed.

Various materials exhibit viscoelasticity, with deformation depending on load, time and temperature. That is, given enough loading over a period of time, many materials will gradually undergo some level of deformation — and the process may speed up as the material gets hotter. For example, an amorphous solid such as glass may act more like a liquid at elevated temperatures, at which its time-dependent response can be measured in seconds. On the other hand, at room temperature, its stiffness is much greater, so glass may still flow, but the time-dependent response is measured in years or decades.

Viscoelastic behavior is similarly found in other materials such as wood, polymers, human tissue and solid rocket propellants, to name a few. Because of this complex behavior, the use of linear material properties is generally inadequate in accurately determining the final shape of a viscoelastic material, the time taken to arrive at that geometry, and the stresses on the part. In these cases, the material's viscoelasticity must be taken into account in the simulation.

Viscoelastic Material Models

In mechanical solutions from ANSYS, viscoelasticity is implemented through the use of Prony series. The shear and volumetric responses are separated, and the well-known relationships between shear modulus G and bulk modulus K are shown below:

$$G = \frac{E}{2(1+\nu)}$$

$$K = \frac{E}{3(1-2\nu)}$$

Instead of having constant values for G and K (and by extension, elastic modulus E and Poisson's ratio ν), these are represented by Prony series in viscoelasticity:

$$G(t) = G_0\left[\alpha_\infty^G + \sum_{i=1}^{n_G} \alpha_i^G e^{-\frac{t}{\tau_i^G}}\right]$$

$$K(t) = K_0\left[\alpha_\infty^K + \sum_{i=1}^{n_K} \alpha_i^K e^{-\frac{t}{\tau_i^K}}\right]$$

These equations imply that the shear and bulk moduli are represented by a decaying function of time t. Simply stated, the user provides pairs of relative moduli α_i and relaxation time τ_i, which represent the amount of stiffness lost at a given rate.

For simplicity, only the shear term G will be considered for subsequent discussions. Start off at time t equal to 0 with the full stiffness (instantaneous shear moduli). Hence:

$$G(0) = G_0\left[\alpha_\infty^G + \sum_{i=1}^{n_G} \alpha_i^G\right]$$

This implies that the sum of the input relative moduli α_i must be less than or equal to 1.0. Consider the extreme case at infinite time, which gives:

$$G(\infty) = G_0\left[\alpha_\infty^G\right]$$

This means that the infinite modulus α_∞ represents the percentage of remaining stiffness. The user-input relative moduli α_i, on the other hand, is the percentage of stiffness that is lost, with τ_i representing the time constant.

Example Problem

Figure 1 shows a rubber bumper being compressed by two rigid bodies in which the right body is fixed and the left body is displaced to compress the rubber part in the middle. The rubber bumper was defined with a neo-Hookean hyperelastic material model in ANSYS Workbench *Simulation*.

Fluids Simulation for Viscoelastic Materials

Viscoelastic materials experience behavior that may be characterized as both viscous and solid. In addition to tools for addressing deformation (ANSYS Mechanical software), the ANSYS portfolio also contains technology in ANSYS POLYFLOW software that can be used to investigate deformation as well as applications that are more fluids related, such as rubber profile extrusion, blow molding or fiber spinning. Both modeling approaches reveal a great deal about the uncommon behavior of the wide range of viscoelastic materials.

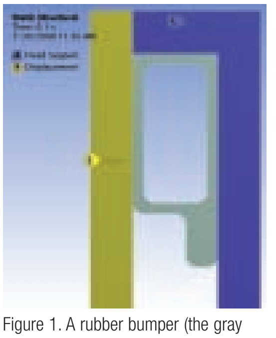

Figure 1. A rubber bumper (the gray part in the middle) is compressed by a fixed and moveable body on its right and left.

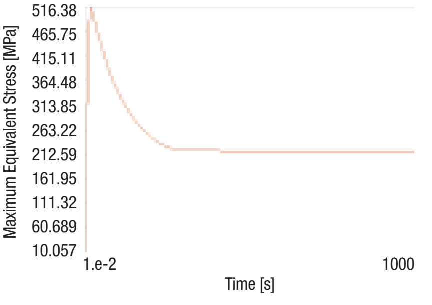

Figure 2. Maximum equivalent stress in the rubber part as a function of time

A `Commands` object was inserted under the rubber part with the following contents:

```
tb,prony,MATID,1,1,shear
tbdata,1,0.5,100
```

For this example, a single Prony pair was defined for the shear behavior. The TB command activates the definition of the Prony pair, and the `TBDATA` command defines the Prony pair values. In this case, a relative modulus of 0.5 was assumed to have a relaxation time of 100 seconds. This means that at infinite time, half of the shear stiffness will be lost at a decay rate, such that at 100 seconds 0.5(e^{-1}), or 18 percent, of the stiffness is relaxed.

Figure 2 shows the maximum equivalent stress in the rubber part as a function of time. Note that the decay is rapid in the beginning. This is due to the exponential function in the Prony series. The response becomes asymptotic, showing that the maximum stress at time equal to 1,000 seconds decreases to nearly half of the maximum stress at the beginning of the solution, as expected. Note that relaxation starts to occur at the beginning of the solution, and this model experiences a multiaxial state of stress, explaining why the long-term maximum stress value is not exactly half of the peak value.

Figure 3 displays the model at time equal to 10 seconds. Note the self-contact that occurs due to the large imposed displacement. A comparison with Figure 4 — displaying equivalent stress at time equal to 1,000 seconds — shows not only the reduction of stress but also some redistribution that occurs due to stress relaxation.

In this way, the Prony series provides an effective tool in mechanical solutions from ANSYS for calculating deformation of materials where stiffness changes as a function of loading, time and temperature. ■

Contact the author at sheldon.imaoka@ansys.com for the complete paper from which this column is excerpted.

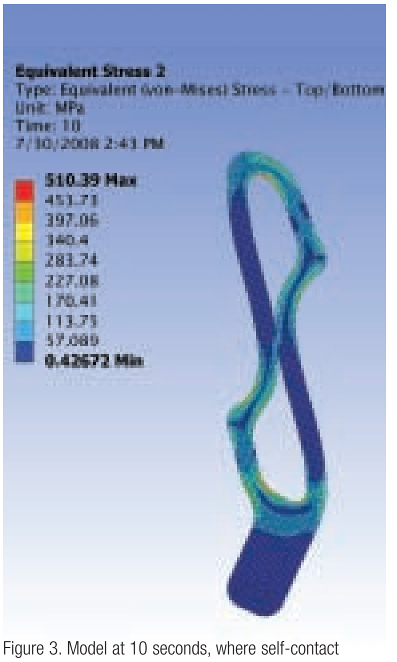

Figure 3. Model at 10 seconds, where self-contact occurs due to the large imposed displacement

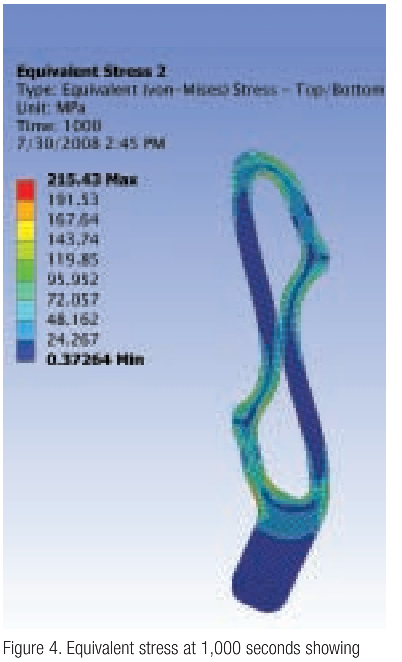

Figure 4. Equivalent stress at 1,000 seconds showing reduction in stress and some redistribution due to stress relaxation