6.2.5 Quasi-static analysis

Products: Abaqus/Standard Abaqus/CAE

Overview

A quasi-static stress analysis in Abaqus/Standard:

- is used to analyze problems with time-dependent material response (creep, swelling, viscoelasticity, and two-layer viscoplasticity);

- is used when inertia effects can be neglected; and

- can be linear or nonlinear.

See "Mass scaling," Section 11.6.1, and "Explicit dynamic analysis," Section 6.3.3, for information on conducting quasi-static analysis in Abaqus/Explicit. See "Implicit dynamic analysis using direct integration," Section 6.3.2, for information on conducting quasi-static analysis using a dynamic procedure in Abaqus/Standard.

Incrementation

You can control the time incrementation in a quasi-static analysis directly, or it can be controlled automatically by Abaqus/Standard. Automatic incrementation is preferred in almost all cases.

Fixed incrementation

If you specify the time increments in a quasi-static analysis directly, fixed time increments equal to the specified initial time increment will be used throughout the analysis.

Input File Usage: *VISCO

Abaqus/CAE Usage: Step module: **Create Step**: **General**: **Visco**

Automatic incrementation

If you select automatic incrementation, the size of the time increment is limited by the accuracy of the integration. The user-specified accuracy tolerance parameter limits the maximum inelastic strain rate change allowed over an increment:

$$tolerance \geq (\dot{\bar{\varepsilon}}^{cr}|_{t+\Delta t} - \dot{\bar{\varepsilon}}^{cr}|_{t})\Delta t,$$

where t is the time at the beginning of the increment, Δt is the time increment (so that $t + \Delta t$ is the time at the end of the increment), and $\dot{\bar{\varepsilon}}^{cr}$ is the equivalent creep strain rate. To achieve accuracy, the value chosen for the accuracy tolerance parameter should be on the order of σ_{err}/E for creep problems, where σ_{err} is an acceptable level of error in the stress and E is a typical elastic modulus, or on the order of the elastic strains for viscoelasticity problems.

Input File Usage: *VISCO, CETOL=*tolerance*

Abaqus/CAE Usage: Step module: **Create Step**: **General**: **Visco**: **Incrementation**: **Creep/swelling/viscoelastic strain error tolerance**: *tolerance*

Selecting explicit creep integration

Nonlinear creep problems ("Rate-dependent plasticity: creep and swelling," Section 23.2.4) that exhibit no other nonlinearities can be solved efficiently by forward-difference integration of the inelastic strains if the inelastic strain increments are smaller than the elastic strains. This explicit method is efficient computationally because, unlike implicit methods, iteration is not required. Although this method is only conditionally stable, the numerical stability limit of the explicit operator is in many cases sufficiently large to allow the solution to be developed in a reasonable number of time increments.

For creep at very low stress levels, however, the unconditional stability of the backward difference operator (implicit method) is desirable. In such cases Abaqus/Standard will invoke the implicit integration scheme automatically.

Explicit integration can be less expensive computationally and simplifies implementation of user-defined creep laws in user subroutine CREEP; you can restrict Abaqus/Standard to using this method for creep problems (with or without geometric nonlinearity included). See "Rate-dependent plasticity: creep and swelling," Section 23.2.4, for further details.

Input File Usage: *VISCO, CETOL=*tolerance*, CREEP=EXPLICIT

Abaqus/CAE Usage: Step module: **Create Step**: **General**: **Visco**: **Incrementation**: **Creep/swelling/viscoelastic strain error tolerance**: *tolerance* and **Creep/swelling/viscoelastic integration: Explicit**

Integration scheme for viscoelasticity and rate-dependent yield

Problems including "Time domain viscoelasticity," Section 22.7.1, are always integrated with an unconditionally stable operator. The time step in these problems is limited only by the accuracy tolerance parameter defined above.

Problems including "Rate-dependent yield," Section 23.2.3, and "Parallel rheological framework," Section 22.8.2, are always integrated using an implicit, unconditionally stable method. The accuracy tolerance parameter does not limit the inelastic strain rate change and can be set equal to any nonzero value to activate automatic time incrementation.

Unstable problems

Some types of analyses may develop local instabilities, such as surface wrinkling, material instability, or local buckling. In such cases it may not be possible to obtain a quasi-static solution, even with the aid of automatic incrementation. Abaqus/Standard offers the ability to stabilize this class of problems by applying damping throughout the model in such a way that the viscous forces introduced are sufficiently large to prevent

instantaneous buckling or collapse but small enough not to affect the behavior significantly while the problem is stable. The available automatic stabilization schemes are described in detail in "Automatic stabilization of unstable problems" in "Solving nonlinear problems," Section 7.1.1.

Initial conditions

Initial values of stresses, temperatures, field variables, solution-dependent state variables, etc. can be specified, as described in "Initial conditions in Abaqus/Standard and Abaqus/Explicit," Section 34.2.1.

Boundary conditions

Boundary conditions can be applied to any of the displacement or rotation degrees of freedom (1–6); to warping degree of freedom 7 in open-section beam elements; or, if hydrostatic fluid elements are included in the model, to fluid pressure degree of freedom 8. If boundary conditions are applied to rotation degrees of freedom, you must understand how Abaqus handles finite rotations. See "Boundary conditions in Abaqus/Standard and Abaqus/Explicit," Section 34.3.1.

Loads

The following types of loading can be prescribed in a quasi-static analysis:

- Concentrated nodal forces can be applied to the displacement degrees of freedom (1–6); see "Concentrated loads," Section 34.4.2.

- Distributed pressure forces or body forces can be applied; see "Distributed loads," Section 34.4.3. The distributed load types available with particular elements are described in Part VI, "Elements."

Predefined fields

The following predefined fields can be specified in a quasi-static analysis, as described in "Predefined fields," Section 34.6.1:

- Although temperature is not a degree of freedom in quasi-static analysis, nodal temperatures can be specified. Any difference between the applied and initial temperatures will cause thermal strain if a thermal expansion coefficient is given for the material ("Thermal expansion," Section 26.1.2). The specified temperature also affects temperature-dependent material properties, if any.

- The values of user-defined field variables can be specified. These values affect only field-variable-dependent material properties, if any.

Material options

The quasi-static procedure in Abaqus/Standard is generally used to analyze quasi-static creep and swelling problems, which occur over fairly long time periods ("Rate-dependent plasticity: creep and swelling," Section 23.2.4). This procedure can also be used to analyze viscoelastic materials ("Time domain viscoelasticity," Section 22.7.1, and "Parallel rheological framework," Section 22.8.2) and two-layer viscoplastic materials ("Two-layer viscoplasticity," Section 23.2.11). In addition, all material models that are valid in a static analysis procedure can be used.

Elements

Any of the stress/displacement elements in Abaqus/Standard (including those with temperature or pressure degrees of freedom) can be used in a quasi-static stress analysis—see "Choosing the appropriate element for an

Output

In addition to the usual output variables available in Abaqus/Standard (see "Abaqus/Standard output variable identifiers," Section 4.2.1), the following variables are provided specifically for creep problems:

Element integration point variables:

CEEQ   Equivalent creep strain, $\int_0^t \dot{\bar{\varepsilon}}^{cr}\, dt$.

CESW   Magnitude of the swelling strain.

CEMAG   Magnitude of the creep strain, $\sqrt{\frac{2}{3}\boldsymbol{\varepsilon}^{cr} : \boldsymbol{\varepsilon}^{cr}}$.

CEP   Principal creep strains.

CE   Output of all of the creep strain components and CEEQ, CESW, and CEMAG.

Input file template

```
*HEADING
...
*BOUNDARY
```
Data lines to specify zero-valued boundary conditions
```
*INITIAL CONDITIONS
```
Data lines to specify initial conditions
```
*AMPLITUDE
```
Data lines to define amplitude variations
```
**
*STEP (,NLGEOM)
*VISCO, CETOL=tolerance
```
Data line to define time incrementation and a "real" time scale
```
*BOUNDARY
```
Data lines to describe nonzero boundary conditions
```
*CLOAD and/or *DLOAD and/or *TEMPERATURE and/or *FIELD
```
Data lines to specify loading
```
*END STEP
```