







## **UVARM**

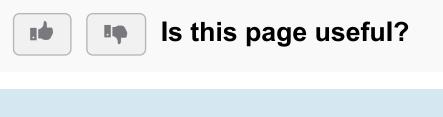




- will be called at all material calculation points of elements for which the material definition includes the specification of user-defined output variables;
- might be called multiple times for each material point in an increment, as Abaqus/Standard iterates to a converged solution;
- will be called for each increment in a step;
- allows you to define output quantities that are functions of any of the available integration point quantities listed in the Output Variable Identifiers table (Using Abaqus/Standard Output Variable Identifiers);
- allows you to define the material directions as output variables;
- can be used for gasket elements;
- can call utility routine GETVRM to access material point data;
- cannot be used with linear perturbation procedures, except for the static perturbation procedure;
- will be called once per load case in a static perturbation procedure with multiple load cases; and
- cannot be updated in the zero increment.

## This page discusses:

- Accessing Material Point Data
- <u>Using User-Defined Output Variables</u>
- <u>User Subroutine Interface</u>
- Variables to Be Defined
- Variables Passed in for Information
- Example: Calculation of Stress Relative to Shift Tensor



## See Also

Obtaining Material Point Information in an Abaqus/Standard Analysis

In Other Guides

\*USER OUTPUT VARIABLES

**UVARM** 

<u>Using Abaqus/Standard Output Variable</u> <u>Identifiers</u>