Simple guide on the use of UMAT-UMATHT subroutines for localizing gradient damage model

Leong Hien Poh (leonghien@nus.edu.sg)

The sample files are for selected examples in the article: "A simple implementation of localizing damage model in Abaqus" https://journals.sagepub.com/doi/10.1177/10567895221109622 Detailed discussions on the localizing gradient enhancement and its applications can be found in the papers below:

- Poh and Sun, 2017. Localizing gradient damage model with decreasing interactions. Int J Numer Meth Engng 110, 503.
- Xu and Poh, 2019. Localizing gradient Rousselier model for ductile fracture. Int J Numer Meth Engng 119, 826.
- Wang et al, 2019. Localizing gradient damage model with micro inertia for dynamic fracture. Comput Methods Appl Mech Eng 355, 492.
- Shedbale et al, 2021. A localizing gradient enhanced isotropic damage model with Ottosen equivalent strain for the mixed-mode fracture of concrete. Int J Mech Sci 199, 106410.
- Zhang et al, 2021. Size effect analysis of quasi-brittle fracture with localizing gradient damage model. Int J Damage Mech 30, 1012.

The sample files can serve as the basis for your incorporation of other constitutive models. If you find the sample files and subroutines / papers useful for your work, please cite the relevant papers. For academic use only. Thank you.

1. Files

The following files are listed in the folder:

- **MAIN.for**: Fortran subroutine of the localizing gradient damage model implemented using UMAT-UMATHT subroutines. The codes can be used for both 2D (plane stress/strain) and 3D analyses. The von Mises equivalent strain is considered;
- **CTS_2D.inp**: Abaqus inp file for 2D analysis of the CTS problem (including 2500 8-noded quadrilateral elements);
- **CTS_3D.inp**: Abaqus inp file for 3D analysis of the CTS problem (including 2500 20-noded brick elements);

2 Element types

The Abaqus in-built coupled temperature-displacement elements are utilized in the simplified implementation approach. The different element types and their definitions can be found in Abaqus Analysis User's Manual.

In the examples, the Fortran subroutine **MAIN.for** can run with the following input files for analyses in 2D and 3D:

- **CTS_2D.inp:** the element types CPE8T and CPS8T (Line 7711) can be selected for plane strain and plane stress analysis respectively;
- **CTS_3D.inp:** C3D20T (Line 18013) is adopted for 3D analysis.

It is not recommended to use modified (M) or reduced integration (R) elements in this implementation, since the thermal and mechanical fields are coupled through RPL.

3. Numerical parameters

The numerical parameters defined in UMAT-UMATHT should be adopted properly before performing a coupled thermo-mechanical analysis.

3.1 Material definition of UMAT

The *DEPVAR option is used to allocate the solution-dependent state variables (STATEV) at each Gauss point. Details of the STATEVs defined in UMAT are summarized as below:

STATEV No.	Notations	Comments
1	κ	A history variable to be passed into the new increment
2	ω	Be stored for post-processing
3	g	To be passed into UMATHT
4	$\frac{\partial g}{\partial \omega}$	To be passed into UMATHT
5	$\frac{\partial g}{\partial \kappa}$	To be passed into UMATHT
6*	1 / 0	A value controlling the element deletion flag

** Only in the subroutine with element deletion.*

For the constitutive model programmed in UMAT, the user-defined mechanical material behavior is given as:

```
*User Material, constants=4, UNSYMM, TYPE=MECHANICAL
        1e+9,    0.2,    10.0,    0.002
```

where the four material properties are corresponding to E (Young's modulus), υ (Poisson's ratio), k (the ratio between compressive and tensile strengths) and κ_0 (damage initiation threshold).

In addition, the material parameters α and β of the exponential damage evolution law (ω) are defined by "Alpha" and "Beta" respectively, the residual parameter R and the rate parameter η of the interaction function (g) are defined by "RR" and "Eta" respectively, i.e.,

```
          PARAMETER(Zero=0.0d0, One=1.0d0, Two=2.0d0, Three=3.0d0,
$ Six=6.0d0, TOL=1.0d-25, Alpha=1.0d0, Beta=9.0d0, RR=5.0d-3,
$ Eta=5.0d0)
```

3.2 Material definition of UMATHT

A transient heat analysis is considered in UMATHT. In the following, the thermal material behavior is defined:

```
*User Material, type=thermal, constants=2
        0.2,   2e-3
*Density
        1,
```

where the two constants are k (thermal conductivity) and c (specific heat capacity) respectively, the thermal conductivity corresponds to the length scale parameter:

$$k = l$$

Note that we assume density $\rho = 1$.

4. Starting a simulation

To start a simulation, the UMAT-UMATHT subroutine and the Abaqus inp file should be in the same folder. Then, submit the following command through the Abaqus Command window:

```
abaqus job = CTS_2D, user = MAIN
```

Note that a small time-step size is required to ensure the stability of transient heat analysis, e.g., a time-step size of 0.0005 is suggested when $c = 0.002$, i.e.,

```
*COUPLED TEMPERATURE-DISPLACEMENT,DELTMX=100.
5e-4, 1., 1e-5, 5e-4
```