UMAT_Driver - An Abaqus replacement for Materialpoint simulations

Nils Lange

January 3, 2023

1 Description

Often it is necessary to test material routines written for Abaqus independent of a macro simulation. Especially when an UMAT has to be tested very often for different Stress or Strain trajectories, using homogeneous Abaqus one Element simulations is unfavorable, since Abaqus takes a lot of time in booking out licenses etc. With UMAT_Driver material routines can be tested in parallel without changes in the UMAT code.

Features

- UMAT interface as defined by Abaqus (actual subroutine to be supplied by user)
- UEXTERNALDB interface as defined by Abaqus (actual subroutine to be supplied by user)
- ABQINTERFACE for explicit interfaces, has to be included by the users routines which use any Abaqus utility functions
- Abaqus utility functions: GETOUTDIR, GETJOBNAME, XIT, ROTSIG, SPRINC, SPRIND, SINV, STDB_ABQERR, SMAIntArrayCreate, SMAIntArrayAccess, SMAIntArrayDelete, SMAFloatArrayCreateSP, SMAFloatArrayCreateDP, SMAFloatArrayCreate, SMAFloatArrayAccess, SMAFloatArrayDelete
- unit=6 writes to a log.dat file, unit=7 writes to a log.msg file
- parallelization with openMP, each thread handles one step and mimics an Abaqus one Element, UEXTERNALDB is called in each thread at the in the Abaqus documentation mentioned occasions
- Inputfile similar to Abaqus
- Strain driven (DSTRAN or DFGRD1) or Stress driven simulations
- Trajectories as multi linear curves with linear interpolation
- Time step control

Requirements

- FORTRAN INTEL ifort with MKL (tested with intel one api 2022.0.1 under linux, and intel one api 2021.0.3 under windows)
- `ABQinterface` from uellib must be supplied together with the user routines

2 Compilation and Usage

Linux:

- Adapt the Makefile by setting `UMATSRC` to the correct source file (including all user routines, especially `UEXTERNALDB` and `ABQinterface` from uellib) and `additionalSRC` to any needed including directories, if not needed leave it blank
- Delete a old compilation binary by:

```
make clean
```

- Compile the program by: (`.o` will be saved to `bin` folder)

```
make
```

- Start the program with

```
./UMAT_Driver.o job=..jobname..  cpus=n_cpus
```

Windows:

- Adapt the Compile.bat batchscript by setting `UMATSRC` and `additionalSRC` correctly
- Compile the program by: (`.exe` will be saved to `bin` folder)

```
compile
```

- Start the program with

```
UMAT_Driver.exe job=..jobname..  cpus=n_cpus
```

3 Inputfile

The inputfile is similar to the Abaqus inputfile, but tailored to the needs of pure material point simulations. None of the following keywords are optional, but may be defined in arbitrary order before the Step definition.
Material definition

```
*Material
..Materialname..
n_constants
C_1, C_2, ... , C_n
*Depvar
n_state variables
```

Number direct stress components

```
*NDI
n
```

Number shear stress components

```
*NSHR
n
```

Now the step definition will be shown. At least one Step should be defined.
Stepname

```
*Step
*name
..Stepname..
```

Large deformations

```
*Nlgeom
YES .or.  NO
```

Time stepping

```
*Static
```

$\Delta t_{\texttt{begin}},\ t_{\texttt{end}},\ \Delta t_{\texttt{min}},\ \Delta t_{\texttt{max}}$

The stress or deformation measure has to be input as a multi linear curve with its respective values at certain time steps. Within the time steps the values will be interpolated linearly.
Cauchy stress definition (NDI+NSHR)

```
*STRESS
```

t^0
$\sigma_1^0,\ \sigma_2^0,\ \sigma_3^0,\ \sigma_4^0,\ \sigma_5^0,\ \sigma_6^0$
t^1
$\sigma_1^1,\ \sigma_2^1,\ \sigma_3^1,\ \sigma_4^1,\ \sigma_5^1,\ \sigma_6^1$
$\dots$
t^n
$\sigma_1^n,\ \sigma_2^n,\ \sigma_3^n,\ \sigma_4^n,\ \sigma_5^n,\ \sigma_6^n$

.or. True strain definition (NDI+NSHR)

```
*STRAN
```

t^0
$\varepsilon_1^0,\ \varepsilon_2^0,\ \varepsilon_3^0,\ \gamma_4^0,\ \gamma_5^0,\ \gamma_6^0$
t^1
$\varepsilon_1^1,\ \varepsilon_2^1,\ \varepsilon_3^1,\ \gamma_4^1,\ \gamma_5^1,\ \gamma_6^1$
$\dots$
t^n
$\varepsilon_1^n,\ \varepsilon_2^n,\ \varepsilon_3^n,\ \gamma_4^n,\ \gamma_5^n,\ \gamma_6^n$

.or. deformation gradient definition (9)

```
*DFGRD
```

t^0
$H_{11}^0,\ H_{21}^0,\ H_{31}^0,\ H_{12}^0,\ H_{22}^0,\ H_{32}^0,\ H_{13}^0,\ H_{23}^0,\ H_{33}^0$
t^1
$H_{11}^1,\ H_{21}^1,\ H_{31}^1,\ H_{12}^1,\ H_{22}^1,\ H_{32}^1,\ H_{13}^1,\ H_{23}^1,\ H_{33}^1$
$\dots$
t^n
$H_{11}^n,\ H_{21}^n,\ H_{31}^n,\ H_{12}^n,\ H_{22}^n,\ H_{32}^n,\ H_{13}^n,\ H_{23}^n,\ H_{33}^n$

Note : H_{ij} is the displacement gradient with $F_{ij} = H_{ij} + \delta_{ij}$

4 Example

A simple example is given, where a elastic-plastic UMAT outputs the stress through UEXTERNALDB at the end of each step to a file. Abaqus and UMAT_Driver yield the same result.

5 Version history

date	description
2022-11-22	original version
2022-12-19	Tested under Windows + compile batch script
2022-12-21	Timestepping now written to Message .msg file