JANCAE

Japan Association for Nonlinear CAE

UMMDp

Unified Material Model Driver for Plasticity

Users' Guide

180728

1. Preface

The Unified Material Model Driver for Plasticity (UMMDp) library is distributed as Fortran source codes. The Fortran compiler specified by the software vender in your analysis environment must be prepared. Please see the manual of your FE code for the details of the environment for compiling the user subroutines.

In this guide, the sections of each FE code related to the use of the UMMDp are described below. Please read these sections and confirm that the sample of the user subroutine provided by the developer of each FE code work normally before using the UMMDp.

In the next section, the details of how to use the UMMDp for each FE code are provided. The common notations are:

dir_ummdp: directory containing the UMMDp source code files
dir_run: directory where the program is executed
solvername: name of the FE code (e.g., abaqus and lsdyna)
jobname: job name (name of the input file).

In the following operation procedure, the prompts "%" and "/>" represent the examples in UNIX/Linux and Windows, respectively.

The common steps for all FE codes are as follows.

(1) Preparation of UMMDp source.
Merge UMMDp source files into one file.

(2) Writing procedure to call UMMDp in the input data.
To call the material user-subroutine, specific keywords are required to be written in the input data. The keywords include the material constants, such as the coefficients of the yield function.

(3) Execution of FE program with the UMMDp.
When a command is typed to execute the FE program, options are added to compile the UMMDp and to link to the FE program.

2. Abaqus

2.1 Related section of User's Manual

(1) Command to execute program with user-subroutine

Abaqus Analysis User's Guide

3.2.2 Abaqus/Standard, Abaqus/Explicit, and Abaqus/CFD execution.

(2) Option for execution without compile

Abaqus Analysis User's Guide

3.2.18 Making user-defined executables and subroutines.

(3) Keywords for set-up of the UMMDp

Abaqus Keywords Reference Guide

*DEPVAR: define the number of solution-dependent state variables

*ORIENTATION: define local material axis for anisotropy

*USER MATERIAL: define the material constants used in UMAT

*USER OUTPUT VARIABLE: define the number of user output variables.

(4) Specification of Abaqus user-subroutines used in the UMMDp

Abaqus User Subroutines Reference Guide

1.1.44 UMAT: user subroutine to define a material's mechanical behavior

1.1.58 UVARM: user subroutine to generate element output.

(5) User defined mechanical properties with UMAT

Abaqus Analysis User's Guide

26.7.1 User-defined mechanical material behavior.

2.2 Usage

(1) Preparation of program source file

Put the UMMDp source codes into one file with the "plug-in" first.

```
unix_linux% cd    dir_ummdp
unix_linux% cp    plug_ummdp_abaqus.f    jobname_ummdp.f
unix_linux% cat    ummdp*.f    >>    jobname_ummdp.f
```

```
windows> cd    dir_ummdp
windows> copy    plug_ummdp_abaqus.f    jobname_ummdp.for
windows> type    ummdp*.f    >>    jobname_ummdp.for
```

(2) Preparation of the input file

This section describes the keywords in the input data file for use in the UMMDp.

(a) Definition of the material model (the details will be provided later)

```
*Material, name=ummdp
*User Material, constants=26
0, 1000.0, 0.3,          2, -0.069888, 0.936408, 0.079143, 1.003060
0.524741, 1.363180, 0.954322, 1.023770, 1.069060, 0.981171, 0.476741, 0.575316
0.866827, 1.145010, -0.079294, 1.404620, 1.051660, 1.147100, 8.0,        0
1.0, 0
```

(b) Definition of the principal axis for the material anisotropy

Define the material axis for orthotropic anisotropy (e.g., rolling direction and transversal direction). For the details of the settings, please refer to the manual.

```
*Orientation, ....
```

(c) Define the number of internal state variables

Set the number of state variables to 1+NTENS, where NTENS is the number of components of the tensor variables. NTENS=3 for plane stress or a shell element, and NTENS=6 for a solid element. The 1st "1" is reserved for the equivalent plastic strain, and NTENS is reserved for the plastic strain components. The following example corresponds to a solid element:

```
*Depvar
7,
```

(d) Define the user output variables (optional)

UMMDp can output the following two user variables:

(d1) current equivalent stress (the value calculated by substituting the stress components for the yield function)

(d2) current yield stress (the value calculated by substituting the equivalent plastic strain for the function of the isotropic hardening curve).

To output these two variables, define two user variables:

```
*User Output Variables
2,
```

(e) Define output variables for post processing

This keyword controls the output variables (e.g., equivalent plastic strain and equivalent stress) for post processing.

```
*Output, Field
*Element Output
SDV, UVARM
```

(3) Execution of program

Type the following command. Option “user=” specifies the UMMDp source file.

```
%> abaqus    job=jobname    user=jobname_ummdp
```

3. ADINA

3.1 Related section of User's Manual

(1) Command to execute program with user-subroutine

%ADINAHOME%¥usrdll¥README.txt: Procedure for compiling and replacing dll file. (¥ is backslash)

(2) Keywords for set-up of the UMMDp

ADINA User Interface Command Reference Manual Volume I: ADINA Solids & Structures

MATERIAL USER-SUPPLIED: define the material constants used in user-subroutine

AXES*****: define local axis

SET-AXES-MAT *****: assign local axis to material.

(3) Specification of ADINA user-subroutines used in the UMMDp

ADINA Theory and Modeling Guide Volume I: ADINA Solids & Structures

3.18 User-coded material model

(4) Description on mechanical properties in ADINA

ADINA Theory and Modeling Guide Volume I: ADINA Solids & Structures

3.1 Stress and strain measures in ADINA

3.2 Usage

(1) Preparation of program source file

Put the UMMDp source codes into one file with the "plug-in" first. ADINA has two types of user material subroutines; UCMAT2 and UCMAT3 for the 2D and 3D solid elements, respectively. These subroutines are separated and one should be selected as the plug-in for the UMMDp, depending on the analysis. ADINA does not support a material user subroutine for the shell element.

```
unix_linux% cd    dir_ummdp
unix_linux% cp    plug_ummdp_adina_UCMAT3(or 2).f    jobname.f
unix_linux% cat    ummdp*.f    >>    jobname.f
```

```
windows> cd    dir_ummdp
```

```
windows> copy    plug_ummdp_adina_UCMAT3(or 2).f    jobname.f
windows> type    ummdp*.f    >>    jobname.f
```

Copy the combined file to the %ADINAHOME%usrdll directory, and edit "Makefile.adusr" in this directory as follows:

for a 2D solid element

```
MAT2D_OBJ = jobname.obj
```

for a 3D solid element

```
MAT3D_OBJ = jobname.obj
```

Type the following command to compile all files in the directory:

```
nmake /f Makefile.adusr
```

Copy the file "adusr.dll" created here to the %ADINAHOME%x64 directory (making a backup of the original "adusr.dll" is recommended).

For the details of this operation, refer to the README.txt in the same directory.

(2) Preparation of input file

This section describes the keywords in the input data file for use in the UMMDp.

(a) Definition of the material model (the details will be described later)

```
MATERIAL USER-SUPPLIED 1,
INTEG=FORWARD AUTOLEN=NO,
LENGTH1=13 LENGTH2=0 LENGTH3=13 LENGTH4=0,
NCTI=26 NSCP=0,
CTI1=0,
CTI2=1000.0 CTI3=0.3,
CTI4=2,
CTI5=-0.069888    CTI6=0.936408    CTI7=0.079143    CTI8=1.003060,
CTI9=0.524741    CTI10=1.363180    CTI11=0.954322    CTI12=1.023770,
CTI13=1.069060    CTI14=0.981171    CTI15=0.476741    CTI16=0.575316,
CTI17=0.866827    CTI18=1.145010    CTI19=-0.079294    CTI20=1.404620,
CTI21=1.051660    CTI22=1.147100    CTI23=8,
CTI24=0,
CTI25=1.0,
```

```
CT126=0
```

(b) Definition of the principal axis for material anisotropy

Define the material axis for orthotropic anisotropy (e.g., rolling and transversal directions). For the detail of the settings, please refer to the manual.

```
AXES CONSTANT NAME=1,
AX=1.0 AY=0.0 AZ=0.0,
BX=0.0 BY=1.0 BZ=0.0
*
SET-AXES-MAT VOLUMES
@CLEAR
1 1 1 2
@
```

(c) Definition of internal state variables

ARRAY(1) for equivalent plastic strain

ARRAY(2)–(7) for plastic strain components

ARRAY(8) –(13) for stress components.

The numbers of the internal state variables are defined in LENGTH1–4.

Set AUTOLEN “NO”

LENGTH1: number of real type state variables, including variables that are not saved to the output file

LENGTH2: number of integer type state variables including variables that are not saved to the output file

LENGTH3: number of real type state variables to be saved to the output file

LENGTH4: number of real type state variables to be saved to the output file

Be sure to set LENGTH 3 ≦ LENGTH 1, LENGTH 4 ≦ LENGTH 2.

To fix the state variables within the subroutine, set the AUTOLEN parameter in MATERIAL USER - SUPPLIED to YES.

(d) Define user output variables

In default, the output of the variables to the result file is restricted. To remove the limit, use the following keyword:

```
PORTHOLE RESULTS-=NO
```

In post-processing, “USER_VARIABLE” can be selected to view the internal state variables.

(3) Execution of program

There are two ways to execute the program with a user-subroutine.

(a) Execution by the user interface.

Load *.in file with the AUI.exe program

Execute analysis by “Solution>Data File/Run...”.

(b) Execution by batch file.

Assign dat output in *.in file.

```
adina file=''
```

Edit batch file as follows, and execute (jobname is the name of the *.in file).

```
%ADINAHOME%¥x64¥aui   -b jobname.in
%ADINAHOME%¥x64¥adina   -b -s jobname.dat
```

4. ANSYS

4.1 Related section of User's Manual

(1) Required environment to use ANSYS user subroutines.

Installation and Licensing

- Installation and Licensing Documentation
  - Linux Installation Guide
    - 2. Platform Details
    - 2.2. Compiler Requirements for Linux Systems: Linux
  - Windows Installation Guide
    - 2. Platform Details
    - 2.2. Compiler Requirements for Windows Systems: Windows

(2) How to compile and link a user-subroutine of ANSYS

Mechanical APDL

- Programmer's Reference
  - II. Guide to User-Programmable Features
    - Chapter 1: Understanding User Programmable Features (UPFs)
    - 1.9. Compiling and Linking UPFs on Linux Systems
    - 1.10. Compiling and Linking UPFs on Windows Systems

(3) Overview of ANSYS user-subroutine

Mechanical APDL

- Advanced Analysis Guide
  - Chapter 9: User-Programmable Features and Nonstandard Uses
  - 9.1. User-Programmable Features (UPFs)

(4) Specification of ANSYS user-subroutines used in the UMMDp

Mechanical APDL

- Programmer's Reference
  - II. Guide to User-Programmable Features
    - Chapter 2: UPF Subroutines and Functions
    - 2.4. Subroutines for Customizing Material Behavior
    - 2.4.1. Subroutine UserMat (Creating Your Own Material Model)

Mechanical APDL

- Material Reference
  - Chapter 4: Nonlinear Material Properties

4.19. Custom Material Models

4.19.1. User-Defined Material Model (UserMat)

4.19.4. Using State Variables with User-Defined Materials

(5) ANSYS commands related to the UMMDp set-up

Mechanical APDL

Command Reference

XXI. T Commands

TB - Activates a data table for material properties or special element input.

TBDATA - Defines data for the material data table.

(6) Definition of axes for anisotropy

Mechanical APDL

Structural Analysis Guide

Chapter 13: Shell Analysis and Cross Sections

13.2. How to Create Cross Sections

13.2.2. Defining Layer Data

Mechanical APDL

Command Reference

XX. S Commands

SECDATA - describes the geometry of a section

SECTYPE - associates section type information with a section ID number.

4.2 Usage

(1) Preparation of program source file

There are 3 ways to use user-subroutines in ANSYS as follows:

1. create a DLL file
2. use the /UPF command
3. create a customized ansys.exe.

Here, "1. create a DLL file" is introduced, because it is easy to operate in the Workbench environment.

Step 1

Copy all Fortran files (ummdp*.f files and plug_ummdp_ansys.f) to the following directory.

C:¥Program Files¥ANSYS Inc¥v180¥ansys¥custom¥user¥winx64 (¥ is backslash)

Note 1: If you cannot find the directory, "ANSYS Customization Files for User

Programmable Features" needs to be additionally installed with the ANSYS installer.

Node 2: The backing up of the directory before operation is recommended.

Step 2

Rename "plug_ummdp_ansys.f" to "usermat.f".

Step 3

Launch the VS2012 x64 Native Tools command prompt.

In windows, select [Start] > [All programs] > [Microsoft Visual Studio 2012] > [Visual Studio Tools] > [VS2012 x64 Native Tools Command prompt]

Step 4

Change current directory to

C:¥Program Files¥ANSYS Inc¥v180¥ansys¥custom¥user¥winx64.　(¥ is backslash)

Step 5

Execute "ANSUSERSHARED.bat".

Step 6

When "Enter a User Programmable Feature Source Filename:" is prompted, type “usermat” to compile the user-subroutines.

Step 7

The following message is displayed when compilation is finished and a dynamic link library "usermatLib.dll” is created.

```
***********************************************************************
usermatLib.dll HAS BEEN SUCCESSFULLY BUILT.
Set the environment variable ANS_USER_PATH to the directory where the
usermatLib.dll resides and run ansys180 to use your newly generated
user shared library.
***********************************************************************
```

Step 8

Copy "usermatLib.dll" to an arbitrary directory (such as "C:¥AnsCustom" in the example).

Step 9

Set the path environment variable on Windows as follows:

ANS_USER_PATH=C:¥AnsCustom

Step 10

After all of the steps, the user-subroutine can be used in ANSYS. When ANSYS is executed, the following two lines will appear if the set-up is complete:

```
User Link path (ANS_USER_PATH): C:¥AnsCustom
```

Note - This ANSYS version was linked by Licensee

(2) Preparation of the input file

This section describes the keywords in the input data file for use in the UMMDp.

(a) Definition of material model (the details will be described later)

```
! material property(ummdp)
tb,user,1,,26
tbdata, 1, 0                    ! nela
tbdata, 2, 1000.0               ! E
tbdata, 3, 0.3                  ! nu
tbdata, 4, 2                    ! nyld (Yld2004)
tbdata, 5, -0.069888            ! c'12
tbdata, 6, 0.936408             ! c'13
tbdata, 7, 0.079143             ! c'21
tbdata, 8, 1.003060             ! c'23
tbdata, 9, 0.524741             ! c'31
tbdata, 10, 1.363180            ! c'32
tbdata, 11, 0.954322            ! c'44
tbdata, 12, 1.023770            ! c'55
tbdata, 13, 1.069060            ! c"66
tbdata, 14, 0.981171            ! c"12
tbdata, 15, 0.476741            ! c"13
tbdata, 16, 0.575316            ! c"21
tbdata, 17, 0.866827            ! c"23
tbdata, 18, 1.145010            ! c"31
tbdata, 19, -0.079294           ! c"32
tbdata, 20, 1.404620            ! c"44
tbdata, 21, 1.051660            ! c"55
tbdata, 22, 1.147100            ! c"66
tbdata, 23, 8.0                 ! exponent a
tbdata, 24, 0                   ! nih constant yield stress
tbdata, 25, 1.0                 ! yield stress
tbdata, 26, 0                   ! nkin (must be 0)
```

(b) Definition of the principal axis for material anisotropy

Define the material axis for orthotropic anisotropy (e.g., rolling and transversal directions). For the details of the settings, please refer to the manual. An example is given below.

```
LOCAL, 11, ...              ! define local coordinate system
ESEL, S, ...                ! select elements
EMODIF, ALL, ESYS, 11       ! assign element coordinate system
ESEL, ALL                   ! release selected elements
```

The components of the stress and strain are defined in the result coordinate system. The result coordinate system in default is the global Cartesian coordinate system. If required, to set the result coordinate system to the local coordinate system defined in the element, the following command can be typed in post-processing.

```
RSYS, SOLU
```

(Please see the manual for details.)

(c) Define the number of internal state variables

Set the number of state variables to 2+NTENS, where NTENS is the number of components of the tensor variables. NTENS=3 for plane stress or a shell element and NTENS=6 for a solid element.

1st variable: equivalent plastic strain

2nd variable: thickness strain of a shell element

3rd to 2+NTENS variables: plastic strain components

The following example corresponds to a solid element.

```
tb,state,1,,8
```

(d) Define the user output variables (optional)

To view the results of the internal state variables in post-processing, type the following command before the analysis.

```
OUTRES, SVAR, ALL
```

In post-processing, you can output the state variables as SVAR (element solution).

```
PLESOL, SVAR, 1      ! Contour plot ustatev(1)
PRESOL, SVAR, 1      ! List ustatev(1)
```

(3) Execution of program

Edit the batch file as shown below and execute it.

```
"%ANSYS180_DIR%¥bin¥%ANSYS_SYSDIR%¥ansys180.exe" -b -i jobname.dat -o jobname.out
```

Note: To use the user-subroutine, the licenses of the ANSYS Mechanical Enterprise products are required.

5. LS-DYNA

5.1 Related section of User's Manual

LS-DYNA® KEYWORD USER'S MANUAL VOLUME I

APPENDIX A: User Defined Materials

LS-DYNA® KEYWORD USER'S MANUAL VOLUME II Material Models

AOPT in *MAT_002 *(MAT_ORTHTROPIC_ELASTIC)

(the definition method of the material axis in the LS-DYNA User Material Model is the same as that of MAT_002. Please refer to MAT_002 for the details of the anisotropic axis).

5.2 Usage

(1) Preparation of program source file with the UMMDp Here the method using a static link is explained. To build the LS-DYNA executable module, the subroutine development kit (e.g., ls-dyna_smp_d_r7_1_2_winx64_ifort131_lib.zip, usually called as "object version of LS-DYNA") and an appropriate version of Intel Fortran are required. Please contact your distributer of LS-DYNA to obtain the subroutine development kit.

Step 1

Obtain the subroutine development kit from your distributer and Unzip it.

Step 2

Rename the following two files.

Rename Makefile to Makefile_org

Rename dyn21.f to plug_ummdp_lsdyna_other.f

Step 3

Open plug_ummdp_lsdyna_other.f that is renamed above with text editor. And change name of the subroutines.

From subroutine umat41 to subroutine umat41_org

From subroutine utan41 to subroutine utan41_org

Step 4

Locate all UMMDp source files and Plug-in for LS-DYNA in above directory.

(CAUTION : The plug-in was developed for LS-DYNA R712. If you want to use another version of LS-DYNA, please check the arguments in subroutines.)

Step 5

For Windows

Execute "nmake.exe" in the directory. After compilation and link, you can see "ls-dyna_ummdp_win.exe" that is LS-DYNA executable module with UMMDp.

For Linux

Type "make" in the directory. After compilation and link, you can see "ls-dyna_ummdp_linux" that is LS-DYNA executable module with UMMDp.

(2) Preparation of input file

This section shows the keywords in input data file to use UMMDp.

(a) Definition of material model (Details will be described later)

CAUTION : Number of the material constants in LS-DYNA user defined material (LMC) is fixed as 40.

```
*MAT_USER_DEFINED_MATERIAL_MODELS
$        MID        RO        MT       LMC       NHV    IORTHO     IBULK        IG
       UMMDP    7.8E-9        41        40       100         1         2         3
$      IVECT     IFAIL    ITHERM    IHYPER      IEOS      LMCA
           0         0         0         0         0         0
$       AOPT      MACF        XP        YP        ZP        A1        A2        A3
         2.0         1       0.0       0.0       0.0       1.0       0.0       0.0
$         V1        V2        V3        D1        D2        D3      BETA
         0.0       0.0       0.0       0.0       1.0       0.0       0.0
$       nela         K         G       nyf    cm( 5)    cm( 6)    cm( 7)    cm( 8)
           1  166666.7  76923.08         2 -0.069888  0.936408  0.079143  1.003060
$     cm( 9)    cm(10)    cm(11)    cm(12)    cm(13)    cm(14)    cm(15)    cm(16)
    0.524741  1.363180  0.954322  1.023770  1.069060  0.981171  0.476741  0.575316
$     cm(17)    cm(18)    cm(19)    cm(20)    cm(21)    cm(22)    cm(23)    cm(24)
    0.866827  1.145010 -0.079294  1.404620  1.051660  1.147100       8.0         2
$     cm(25)    cm(26)    cm(27)    cm(28)    cm(29)    cm(30)    cm(31)    cm(32)
       541.0    0.0036     0.249         0
$     cm(33)    cm(34)    cm(35)    cm(36)    cm(37)    cm(38)    cm(39)    cm(40)
```

Please set IBULK=2, IG=3, and cm(1): nela=1. In LS-DYNA, the time increment in the dynamic explicit procedure is calculated from the bulk modulus and shear modulus (rigidity).

IBULK=2 means that the bulk modulus is located in cm(2).

IG=3 means that the shear modulus is located in cm(3).

cm(1)=1 means that the elastic properties are defined with the bulk and shear moduli in the UMMDp.

(b) Definition of the principal axis for material anisotropy

The axis of the anisotropy is defined in:

```
*MAT_USER_DEFINED_MATERIAL_MODELS
```

Set IORTHO=1 to define the anisotropy, and write the directions of the principal axes in Card3–4 in the keyword. For the details, please see the AOPT parameter in the *MAT_002 section of LS-DYNA® KEYWORD USER'S MANUAL VOLUME II Material Models.

(c) Define the number of internal state variables

The number of internal state variables is defined as “NHV” in:

```
*MAT_USER_DEFINED_MATERIAL_MODELS
```

You can use the “HSV” array in the LS-DYNA user subroutines.

(d) Define user output variables (optional)

If required, “HSV” can be output for post processing by setting “NEIPH” for the solid element and “NEIPS” for shell element in:

```
*DATABASE_EXTENT_BINARY
```

(3) Execution of program

For Windows

(1) Using the LS-DYNA program manager

Solver > Select LS-DYNA Solver

Choose the above-mentioned “ls-dyna_ummdp_win.exe”.

(2) Using the command prompt

Type “ls-dyna_ummdp_win.exe i=input” in the command prompt.

For Linux

Type command in the terminal:

%> ls-dyna_ummdp_linux i=input

Here, input “i=input” to assign the input data file of LS-DYNA.

If the memory size is insufficient for the execution, please use the “memory =” option.

6. Marc

6.1 Related section of User's Manual

(1) Execution of job with user-subroutine
Volume A: Theory and User Information
    Program Initiation: How to assign user subroutine in run_marc command
(2) Parameters list related to execute the UMMDp
Volume C: Program Input
    HYPOELASTIC: call user-subroutine hypela2
    MATUDS: define material parameters used in hypela2
    STATE VARS: define internal state variables
    ORIENTATION: define local axis of material anisotropy
    POST: assign post processing variables
(3) Specification of Marc user-subroutines used in the UMMDp
Volume D: User Subroutines and Special Routines
    MD_HYPELA2: user-subroutine for the hypoelastic material
    PLOTV: user-subroutine to output the user variables to the post (t16/t19) file.

6.2 Usage

(1) Preparation of the program source file
Put the UMMDp source codes into one file with the "plug-in" first.

```
unix_linux% cd  dir_ummdp
unix_linux% cp  plug_ummdp_marc.f  jobname_ummdp.f
unix_linux% cat  ummdp*.f  >>  jobname_ummdp.f
```

```
windows> cd  dir_ummdp
windows> copy  plug_ummdp_marc.f  jobname_ummdp.for
windows> type  ummdp*.f  >>  jobname_ummdp.for
```

(2) Preparation of input file

This section describes the keywords in the input data file for use in the UMMDp.

(a) Definition of the material model (the details will be described later)

```
hypoelastic


1,0,11,0,1,ummdp
1,0,0,0,0,0,0
0,0,0,0,0,0,0
         1
matuds
hypela2,1,0,0,26,0,1
0, 1000.0, 0.3,          2, -0.069888, 0.936408, 0.079143, 1.003060
0.524741, 1.363180, 0.954322, 1.023770, 1.069060, 0.981171, 0.476741, 0.575316
0.866827, 1.145010, -0.079294, 1.404620, 1.051660, 1.147100, 8.0,     0
1.0, 0
```

(b) Definition of the principal axis for the material anisotropy

Define the material axis for the orthotropic anisotropy (e.g., rolling and transversal directions). There are several ways to define the local axis. For the details of the settings, please refer to the manual.

```
orientation
.....
```

(c) Define the number of internal state variables.

Set the number of state variables to 2+NTENS, where NTENS is the number of components of the tensor variables. NTENS=3 for plane stress or a shell element and NTENS=6 for a solid element.

1st variable: is reserved for the temperature

2nd variable: equivalent plastic strain

3rd to 2+NTENS variables: plastic strain components

The example of a solid element is given below.

```
state vars       8
```

(d) Define user output variables (optional)

To view the results of the internal state variables, such as the equivalent stress and equivalent plastic strain in post-processing, the post codes must be specified. The post codes for the user variables have negative values.

```
post
15, 16, ......
......
    -1    0        equivalent plastic strain
    -2    0        equivalent stress
    -3    0        flow stress
```

(3) Execution of program

Type the command below. The "–user" option specifies the user-subroutine to be compiled.

```
%> run_marc   -jid   jobname   -user   jobname_ummdp
```

7. Set up material data for UMMDp

The material data in UMMDp as defined as follows.

(1) Parameters for elastic properties

(2) Parameters for yield function

(3) Parameters for isotropic hardening

(4) Parameters for kinematic hardening (2018/7 now on preparation)

The detail of data is shown as follows. In addition, the examples of input data for each FE code are described.

7.1 Parameters for elastic properties

prela(1) : ID for elastic properties
prela(2~) : Data depend on ID

Now, only isotropic Hooke elastic property can be defined. There are 2 ways to set them

(Example 1) ID=0 using Young's modulus and Poisson's ratio
prela(1)=0
prela(2)=200.0E+3 : Young's modulus E
prela(3)=0.3 : Poisson's ratio ν

(Example 2) ID=1 using bulk modulus (volumetric modulus) and modulus of rigidity
prela(1)=1
prela(2)=166666.7 : Bulk modulus $K=E/3/(1-2\nu)$
prela(3)= 76923.08 : Modulus of rigidity $G=E/2/(1+\nu)$

Note : ID=1 is useful for LS-DYNA. LS-DYNA defines the time increment from input data for dynamic explicit procedure. You can specify ibulk=2, ig=3 with ID=1 mode.

7.2 Parameters for yield function

pryld(1) : ID for yield function
(Negative value specifies plane stress yield function)
pryld(2~) : Data depend on ID

Here, ID for yield function and original paper are introduced. Please refer the original paper for the detail of the parameters.

ID=0 : von Mises isotropic (1913)[1]
pryld(1)=0 No subsequent data

ID=1 : Hill quadratic (1948)[2]
pryld(1) =1 # of subsequent data : 6
pryld(1+1) = F
pryld(1+2) = G
pryld(1+3) = H
pryld(1+4) = L
pryld(1+5) = M
pryld(1+6) = N
Note : The parameters (FGHLMN) are same as Hill's original paper. When F=G=H=1, L=M=N=3, Hill's function is identical to von Mises's one.

ID=2 : Barlat yld2004-18p (2005)[3] IJP v.21(2005) p1009-1039.
pryld(1)=2 # of subsequent data : 19
pryld(1+ 1) = c'12
pryld(1+ 2) = c'13
pryld(1+ 3) = c'21
pryld(1+ 4) = c'23
pryld(1+ 5) = c'31
pryld(1+ 6) = c'32

---

[1] von Mises, R. (1913). Mechanik der festen Körper im plastisch deformablen Zustand. Göttin. Nachr. Math. Phys., **1**: 582–592.
[2] R. Hill. (1948). A theory of the yielding and plastic flow of anisotropic metals. Proc. Roy. Soc. London, **193**:281–297. (http://rspa.royalsocietypublishing.org/content/royprsa/193/1033/281.full.pdf)
[3] Barlat F, Aretz H, Yoon JW, Karabin ME, Brem JC, Dick RE (2005). Linear transformation-based anisotropic yield functions., Int. J. Plast. **21**:1009–1039

pryld(1+ 7) = c'44　(c'66 in original paper)
pryld(1+ 8) = c'55　(c'55 in original paper)
pryld(1+ 9) = c'66　(c'44 in original paper)
pryld(1+10) = c"12
pryld(1+11) = c"13
pryld(1+12) = c"21
pryld(1+13) = c"23
pryld(1+14) = c"31
pryld(1+15) = c"32
pryld(1+16) = c"44　(c"66 in original paper)
pryld(1+17) = c"55　(c"44 in original paper)
pryld(1+18) = c"66　(c"55 in original paper)
pryld(1+19) = a (exponent)
Note：Stress components in Voigt notation is $\{\sigma_{xx}, \sigma_{yy}, \sigma_{zz}, \sigma_{yz}, \sigma_{zx}, \sigma_{xy}\}$ in the original paper, but $\{\sigma_{xx}, \sigma_{yy}, \sigma_{zz}, \sigma_{xy}, \sigma_{yz}, \sigma_{zx}\}$ in UMMDp.

ID=3：Cazacu (2006)[4]
pryld(1)=3　　　# of subsequent data：14
pryld(1+ 1) = C11
pryld(1+ 2) = C12
pryld(1+ 3) = C13
pryld(1+ 4) = C21
pryld(1+ 5) = C22
pryld(1+ 6) = C23
pryld(1+ 7) = C31
pryld(1+ 8) = C32
pryld(1+ 9) = C33
pryld(1+10) = C44
pryld(1+11) = C55
pryld(1+12) = C66
pryld(1+13) = a (exponent)
pryld(1+14) = k (tension compression ratio)

ID=4：Karafillis-Boyce (1993)[5]

[4] Cazacu, O., Plunkett, B., Barlat, F., (2006). Orthotropic yield criterion for hexagonal close packed metals., Int. J. Plasticity **22**: 1171–1194.

pryld(1)=4 # of subsequent data : 8

pryld(1+ 1) = C

pryld(1+ 2) = alpha1

pryld(1+ 3) = alpha2

pryld(1+ 4) = gamma1

pryld(1+ 5) = gamma2

pryld(1+ 6) = gamma3

pryld(1+ 7) = c

pryld(1+ 8) = k (k of exponent 2k)

ID=5 : Hu (2005)[6]

pryld(1)=5 # of subsequent data : 10

pryld(1+ 1) = X1

pryld(1+ 2) = X2

pryld(1+ 3) = X3

pryld(1+ 4) = X4

pryld(1+ 5) = X5

pryld(1+ 6) = X6

pryld(1+ 7) = X7

pryld(1+ 8) = C1

pryld(1+ 9) = C2

pryld(1+10) = C3

ID=6 : Yoshida 6th polynomial (2011)[7]

pryld(1)=6 # of subsequent data : 16

pryld(1+ 1) = C1

pryld(1+ 2) = C2

pryld(1+ 3) = C3

pryld(1+ 4) = C4

pryld(1+ 5) = C5

pryld(1+ 6) = C6

---

[5] Karafillis, A. P.; Boyce, M. C. (1993). A general anisotropic yield criterion using bounds and a transformation weighting tensor., J. Mech. Phys. Solids, **41**:1859-1886.

[6] Hu, W. (2005). An orthotropic yield criterion in a 3-D general stress state. Int. J. Plasticity **21**:1771–1796.

[7] Yoshida, F., Hamasaki, H., Uemori, T. (2013). A user-friendly 3D yield function to describe anisotropy of steel sheets. Int. J. Plasticity, **45**:119-139.

pryld(1+ 7) = C7

pryld(1+ 8) = C8

pryld(1+ 9) = C9

pryld(1+10) = C10

pryld(1+11) = C11

pryld(1+12) = C12

pryld(1+13) = C13

pryld(1+14) = C14

pryld(1+15) = C15

pryld(1+16) = C16

ID=-1 : Gotoh biquadratic (1978)[8]

pryld(1) =-1 # of subsequent data : 9

pryld(1+ 1) = A1

pryld(1+ 2) = A2

pryld(1+ 3) = A3

pryld(1+ 4) = A4

pryld(1+ 5) = A5

pryld(1+ 6) = A6

pryld(1+ 7) = A7

pryld(1+ 8) = A8

pryld(1+ 9) = A9

ID=-2 : Barlat YLD2000-2d (2003)[9]

pryld(1)=-2 # of subsequent data : 9

pryld(1+ 1) = alpha1

pryld(1+ 2) = alpha2

pryld(1+ 3) = alpha3

pryld(1+ 4) = alpha4

pryld(1+ 5) = alpha5

pryld(1+ 6) = alpha6

---

[8] Gotoh, M., (1977). A theory of plastic anisotropy based on a yield function of fourth order (plane stress state)—I, Int. J. Mech. Sci. , **19**-9 : 505-512.
(See also J.JSTP, **19**(1978) :337-385)

[9] Barlat, F., Brem, J.C., Yoon, J.W., Chung, K., Dick, R.E., Lege, D.J., Pourboghrat, F., Choi, S.H., Chu, E. (2003). Plane stress yield function for aluminium alloy sheets-part 1: theory, Int. J. Plasticity, **19**:1297-1319.

pryld(1+ 7) = alpha7

pryld(1+ 8) = alpha8

pryld(1+ 9) = a (exponent)

ID=-3 : Vegter (2006)[10]

pryld(1)=-3 # of subsequent data : 3+4*n

pryld(1+ 1) =n (max of i)

pryld(1+ 2) = f_bi0

pryld(1+ 3) = r_bi0

pryld(1+ 3+(i-1)*4+1) = phi_uniaxial(i)

pryld(1+ 3+(i-1)*4+2) = phi_shear(i)

pryld(1+ 3+(i-1)*4+3) = phi_planestrain(i)

pryld(1+ 3+(i-1)*4+4) = omega(i)

ID=-4 : Banabic BBC2005[11]

pryld(1)=-4 # of subsequent data : 9

pryld(1+ 1) = k (k of exponent 2k)

pryld(1+ 2) = a

pryld(1+ 3) = b

pryld(1+ 4) = L

pryld(1+ 5) = M

pryld(1+ 6) = N

pryld(1+ 7) = P

pryld(1+ 8) = Q

pryld(1+ 9) = R

ID=-5 : Barlat YLD89[12]

pryld(1)=-5 # of subsequent data : 4

pryld(1+ 1) = M (exponet)

pryld(1+ 2) = a

---

[10] Vegter, H., den Boogaard, A.H. van (2006). A plane stress yield function for anisotropic sheet material by interpolation of biaxial stress states, Int. J. Plasticity, **22**:557-580.

[11] Banabic, D., Aretz,, D.S. Comşa, H., Paraianu, L.(2005). An improved analytical description of orthotropy in metallic sheets, Int. J. Plasticity **21**:493–512.

[12] Barlat, F., Lian, J.(1989). Plastic behavior and stretchability of sheet metals. Part I : a yield function for orthotropic sheets under plane stress conditions. Int. J. Plasticity. **5**:51–66

pryld(1+ 3) = h
pryld(1+ 4) = p

ID=-6：Banabic BBC2008[13]
pryld(1)=-6 # of subsequent data：2+8*s
pryld(1+1) = s (max of i)
pryld(1+2) = k (k of exponent 2k)
pryld(1+2+(i-1)*8+1) = l1
pryld(1+2+(i-1)*8+2) = l2
pryld(1+2+(i-1)*8+3) = m1
pryld(1+2+(i-1)*8+4) = m2
pryld(1+2+(i-1)*8+5) = m3
pryld(1+2+(i-1)*8+6) = n1
pryld(1+2+(i-1)*8+7) = n2
pryld(1+2+(i-1)*8+8) = n3

ID=-7：Hill1990
Now on preparation

[13] Comsa,D.S., Banabic,D. (2008). Plane-Stress Yield Criterion For Highly-Anisotropic Sheet Metals, Proc. of NUMISHEET 2008.
https://certeta.utcluj.ro/downloads/researchfields/NUMISHEET_2008_2.pdf

7.3 Parameters for isotropic hardening

prihd(1) : ID for isotropic hardening curve.
prihd(2~) : Data depend on ID

The equation of flow curve is introduced as bellow. Here p is equivalent plastic strain.

ID=0 : Sy Perfectly plastic
prihd(1) =0 # of subsequent data : 1
prihd(1+ 1) = Sy (Constant flow stress, no hardening)

ID=1 : Sy0+h*p Linear hardening
prihd(1) =1 # of subsequent data : 2
prihd(1+ 1) = Sy0 Initial yield stress
prihd(1+ 2) = h Hardening slope or plastic coefficient

ID=2 : c*(e0+p)^n Swift type
prihd(1) =2 # of subsequent data : 3
prihd(1+ 1) = c
prihd(1+ 2) = e0
prihd(1+ 3) = n

ID=3 : sy0+c*p^n Ludwick type
prihd(1) =3 # of subsequent data : 3
prihd(1+ 1) = sy0 Initial yield stress
prihd(1+ 2) = c
prihd(1+ 3) = n

ID=4 : sy0+q*(1-exp(-b*p)) Voce type
prihd(1) =4 # of subsequent data : 3
prihd(1+ 1) = sy0 Initial yield stress
prihd(1+ 2) = q
prihd(1+ 3) = b

ID=5 : sy0+q*(1-exp(-b*p))+h*p Voce + Linear type prihd(1) =5 # of subsequent data : 4

prihd(1+ 1) = sy0 Initial yield stress

prihd(1+ 2) = q

prihd(1+ 3) = b

prihd(1+ 4) = h

ID=6 : a(sy0+q*(1-exp(-b*p)) + (1-a)(c*(e0+p)^n) Voce + Swift type

prihd(1) =6 # of subsequent data : 7

prihd(1+ 1) = a fraction

prihd(1+ 2) = sy0

prihd(1+ 3) = q

prihd(1+ 4) = b

prihd(1+ 5) = c

prihd(1+ 6) = e0

prihd(1+ 7) = n

7.4 Parameters for kinematic hardening

prkin(1) : ID for type of kinematic hardening

Now on preparation (2018/7)
Set as follow.

ID=0 : No kinematic hardening
prkin(1) = 0 No subsequent data

7.5 Example of material data input for each FE codes

Example data

Elastic properties	E=200GPa, ν=0.3
Yield function	Yld2004-18p (Coefficients of A2090-T3 in the original paper)
Isotropic hardening	Swift type $\sigma_Y=541.0\ (0.0036+\varepsilon_{eq}{}^{p})^{0.249}$ MPa
Kinematic hardening	N/A

The parameters are stored in one dimensional array prop(i) in program.

:

prop(1)=0	: Elastic property ID=0
prop(2)=200000	: Young's modulus E
prop(3)=0.3	: Poisson's ratio nu
prop(4)=2	: Yield function ID=2 (Yld2004)
prop(5)=-0.069888	: c'12
prop(6)=0.936408	: c'13
prop(7)=0.079143	: c'21
prop(8)=1.003060	: c'23
prop(9)=0.524741	: c'31
prop(10)=1.363180	: c'32
prop(11)=0.954322	: c'44
prop(12)=1.023770	: c'55
prop(13)=1.069060	: c'66
prop(14)=0.981171	: c"12
prop(15)=0.476741	: c"13
prop(16)=0.575316	: c"21
prop(17)=0.866827	: c"23
prop(18)=1.145010	: c"31
prop(19)=-0.079294	: c"32
prop(20)=1.404620	: c"44
prop(21)=1.051660	: c"55
prop(22)=1.147100	: c"66
prop(23)=8.0	: a (exponent)
prop(24)=2	: Isotropic hardening ID=2 Swift type $\sigma=541.0\ (0.0036+p)^{0.249}$
prop(25)=541.0	: c
prop(26)=0.0036	: e0

prop(27)=0.249 : n
prop(28)=0 : Kinematic hardening ID=0

This prop(i) array divided into each properties in UMMDp as follow.

prela(i) : Elastic property
pryld(i) : Yield function
prihd(i) : Isotropic hardening
prkin(i) : Kinematic hardening

The model ID is stored in the top of these arrays.

Here we show the input examples of the material data for each FE code.
The red letter indicates model ID of each properties.

(1) Abaqus

```
*Material, name=ummdp
*User Material, constants=28
0, 200000.0, 0.3,        2, -0.069888, 0.936408, 0.079143, 1.003060
0.524741, 1.363180, 0.954322, 1.023770, 1.069060, 0.981171, 0.476741, 0.575316
0.866827, 1.145010, -0.079294, 1.404620, 1.051660, 1.147100, 8.0,        2,
541.0, 0.0036, 0.249,        0
```

(2) ADINA

```
material user-supplied 1,
integ=forward, autolen=no,
length1=13 length 2=0 length 3=13 length4=0,
ncti=28 nscp=0
cti1=0,
cti2=200000.0, cti3=0.3,
cti4=2,
cti5=-0.069888, cti6=0.936408, cti7=0.079143, cti8=1.003060
cti9=0.524741, cti10=1.363180, cti11=0.954322, cti12=1.023770,
cti13=1.069060, cti14=0.981171, cti15=0.476741, cti16=0.575316,
cti17=0.866827, cti18=1.145010, cti19=-0.079294, cti20=1.404620,
cti21=1.051660, cti22=1.147100, cti23=8.0,
cti24=2,
cti25=541.0, cti26=0.0036, cti27=0.249,
```

```
cti28=0
```

(3) ANSYS

```
!material property(ummdp)
tb,user,1,,28
tbdata, 1, 0                ! nela
tbdata, 2, 200000           ! E
tbdata, 3, 0.3              ! nu
tbdata, 4, 2                ! nyld (Yld2004)
tbdata, 5, -0.069888        ! c'12
tbdata, 6, 0.936408         ! c'13
tbdata, 7, 0.079143         ! c'21
tbdata, 8, 1.003060         ! c'23
tbdata, 9, 0.524741         ! c'31
tbdata, 10, 1.363180        ! c'32
tbdata, 11, 0.954322        ! c'44
tbdata, 12, 1.023770        ! c'55
tbdata, 13, 1.069060        ! c"66
tbdata, 14, 0.981171        ! c"12
tbdata, 15, 0.476741        ! c"13
tbdata, 16, 0.575316        ! c"21
tbdata, 17, 0.866827        ! c"23
tbdata, 18, 1.145010        ! c"31
tbdata, 19, -0.079294       ! c"32
tbdata, 20, 1.404620        ! c"44
tbdata, 21, 1.051660        ! c"55
tbdata, 22, 1.147100        ! c"66
tbdata, 23, 8.0             ! a (exponent)
tbdata, 24, 2               ! nih (Swift curve)
tbdata, 25, 541.0           ! c of c(e0+p)^n
tbdata, 26, 0.0036          ! e0 of c(e0+p)^n
tbdata, 27, 0.249           ! n of c(e0+p)^n
tbdata, 28, 0               ! nkin (must be 0)
```

(4) LS-DYNA

```
*MAT_USER_DEFINED_MATERIAL_MODELS
$        MID         RO         MT        LMC        NHV     IORTHO      IBULK         IG
       UMMDP     7.8E-9         41         40        100          1          2          3
$      IVECT      IFAIL     ITHERM     IHYPER       IEOS       LMCA
           0          0          0          0          0          0
$       AOPT       MACF         XP         YP         ZP         A1         A2         A3
         0.0          1        0.0        0.0        0.0        1.0        0.0        0.0
$         V1         V2         V3         D1         D2         D3       BETA
         0.0        0.0        0.0        0.0        1.0        0.0        0.0
$...... CM(*) of UMAT41 / PROP(*) of UMMDp
$  Use elastic model 1 (bulk modulus K and shear modulus G) for LS-DYNA
$  K=E/3/(1-2nu), G=E/2/(1+nu)
$       nela          K          G        nyf     cm( 5)     cm( 6)     cm( 7)     cm( 8)
           1   166666.7   76923.08          2  -0.069888   0.936408   0.079143   1.003060
$     cm( 9)     cm(10)     cm(11)     cm(12)     cm(13)     cm(14)     cm(15)     cm(16)
    0.524741   1.363180   0.954322   1.023770   1.069060   0.981171   0.476741   0.575316
$     cm(17)     cm(18)     cm(19)     cm(20)     cm(21)     cm(22)     cm(23)     cm(24)
    0.866827   1.145010  -0.079294   1.404620   1.051660   1.147100        8.0          2
$     cm(25)     cm(26)     cm(27)     cm(28)     cm(29)     cm(30)     cm(31)     cm(32)
       541.0     0.0036      0.249          0
$     cm(33)     cm(34)     cm(35)     cm(36)     cm(37)     cm(38)     cm(39)     cm(40)
```

(5) Marc

```
hypoelastic

1,0,11,0,1,ummdp
1,0,0,0,0,0,0
0,0,0,0,0,0,0
         1
matuds
hypela2,1,0,0,28,0,1
0, 200000.0, 0.3,        2, -0.069888, 0.936408, 0.079143, 1.003060
0.524741, 1.363180, 0.954322, 1.023770, 1.069060, 0.981171, 0.476741, 0.575316
0.866827, 1.145010, -0.079294, 1.404620, 1.051660, 1.147100,    8.0,    2,
541.0, 0.0036, 0.249,          0
```