6.11 Example K: User Defined Inelastic Material Model

Sample Input File For A User Defined Material Model

Problem Summary:

Load Type:	Thermomechanical
Load History:	Cyclic
Load Control:	Strain
Load History Data:	Cool-down from 400 ^oC to 23 ^oC , hold temperature during mechanical loading $\dot{T}$ =0.419 ^oC/sec $\dot{\varepsilon}$ =1.667x10^{-4} /sec, ε_{max}=0.015, ε_{min}=0. $\Delta t_{thermal}$ = 0.5 sec.; Δt_{mech} =0.05 sec.
Micromechanics model:	Double Periodicity
Fiber Packing Arrangement:	User Input 4x2 RVE with 2 fibers
Integration Algorithm:	Forward Euler
Constituent Material Model:	Fiber 1: User Model (elastic) Fiber 2: User Model (elastic) Matrix: User Model (Bodner-Partom via USRMAT)
Constituents:	Fiber 1: Imaginary material, user function material properties. Elastic modulus is a function of longitudinal strain: E_{new} = 700- 80.(ε_{11})($E_{previous}$) with units of MPa Fiber 2: Imaginary material, temperature dependent properties input manually. Matrix: Imaginary material, user function temperature dependent material properties. (see USRFUN subroutine)

☞ **Note:** The purpose here is to demonstrate how to use the various user definable subroutines, i.e., **USRMAT, USRFUN**, **USRFORMDE, USRCPEVAL** see section **4.2.12** for more information.

```
test of user subroutines/input
*PRINT
 NPL=10 %
*LOAD
 LCON=3 LOP=1 LSS=1 %
*MECH
 NPTW=3 TI=0.,900.,990. LO=0.,0.,0.015 %
*THERM
 NPTT=3 TI=0.,900.,990. TE=400.,23.,23. %
*MODEL
 MOD=1 %
*SOLVER
 NTF=1 NPTS=3 TIM=0.,900.,990 STP=0.5,0.05 %
*FIBER
 NFIBS=2
 NF=1 MF=99 NDPT=2 MAT=U IFM=2
 NF=2 MF=99 NDPT=2 MAT=U IFM=1 NPE=2 NPV=0
 NTP=3
 TEM=21.,200.,400.
 E1=314.1E9,293.2E9,253.0E9
 E2=0.33,0.33,0.33
 ALPA=4.5E-6,5.3E-6,6.1E-6
 ALPT=4.5E-6,5.3E-6,6.1E-6
*MATRIX
 NMATX=1
 NM=1 MM=99 NDPT=2 MAT=U IFM=2 %
*MRVE
 IDP=99
 NB=4 NG=2
 H=1.,1.,1.,1.
 L=1.,1.
 CM=M1,M1
 CM=F2,M1
 CM=M1,M1
 CM=F1,M1
*CURVE
 NP=2 %
*MACRO
 NT=1
 NC=1 X=1 Y=7 NAM=apdx-k
*END
```

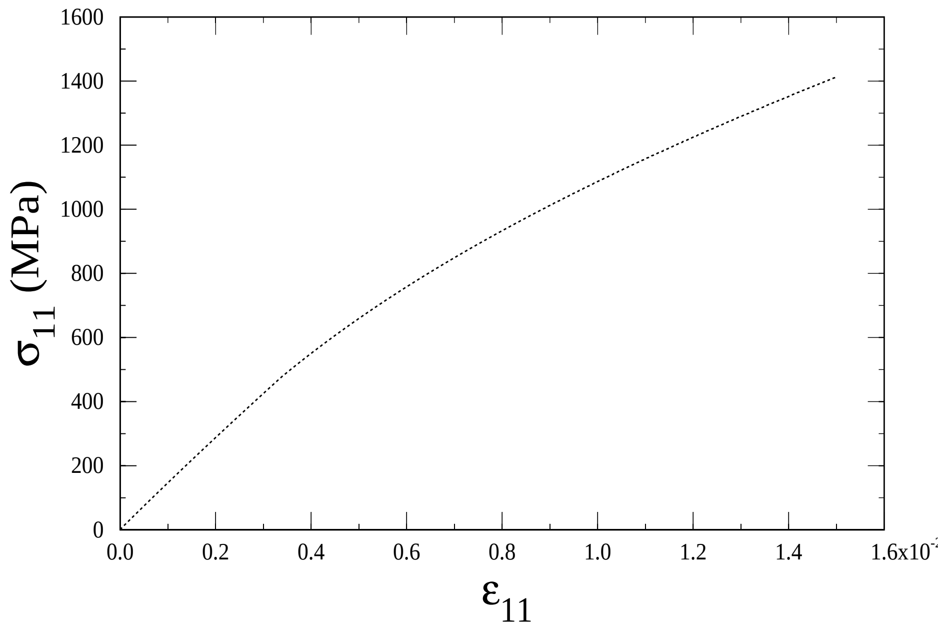

Figure was obtained from the x-y plot data file produced by the present example.

The USRMAT subroutine

The USRMAT subroutine is used here to implement the Bodner-Partom Visco-plastic model currently available in **MAC/GMC**

```
c########################################################################
      SUBROUTINE USRMAT(DSA, SA, PE, PV, D, LOCTISO, TIME, TSTEP,
     &     CTEMP, DTEMPR, NIO, NE, NV, NS, MN, CDUM, DMGF,
     &     NEP, NVP, NSASIZE)

c     purpose:  user material constitutive model for determination of
c               the inelastic strain and state variable rates
c               (used when ncmd = 99)

      IMPLICIT DOUBLE PRECISION (A - H, O - Z)


      CHARACTER*2 CDUM
      DIMENSION SS(6), S(6), R(6)
      DIMENSION DSA(NSASIZE), SA(NSASIZE)
      DIMENSION PV(NVP), PE(NEP), D(3)

************************************************************************
c  note: 1) in this subroutine, [SA] and [DSA] contain the
c           micro (subcell) quantities for aboudi's micromechanics model
c
c        2) arrangement of [dsa] & [sa]  arrays:
c           variable             location
c         +-------------------------------------------------------------
c         | strain rate          (1-6)  (contains ENGINEERING shears)
c         |-------------------------------------------------------------
c         | stress rate          (7-12)
c         |-------------------------------------------------------------
c         | inelastic
c         | strain rate          (13-18)  (contains ENGINEERING shears)
c         |-------------------------------------------------------------
c         | 12 "slots"           (19-30)
c         | for state variables
c         |-------------------------------------------------------------
c         | thermal strain rate  (31-36)
c         +-------------------------------------------------------------
************************************************************************
c   on entry:
c        SA         - vector of total (integrated) quantities (see above)
c        PE(NE)     - vector of elastic constants for material MN
c                     (where NE = # of elastic constants --> 9 MAX)
c        PV(NV)     - vector of viscoplastic constants for material MN
c                    (where NV = # of viscoplastic constants --> 19 MAX)
c        D(3)       - vector of direction cosines (for models 3, 7, & 9)
c        LOCTISO    - flag indicating if ANY material exhibits local
C                     transverse isotropy (and global anisotropy)
c                   = 0 - all materials are at most globally transversely
c                         isotropic (D not used)
```

```
c                  = 1 - at least one material is locally transversely
c                        isotropic (D used)
c         TIME      - current time
c         TSTEP     - current time step
c         CTEMP     - current temperature
c         DTEMPR    - time rate of change of temperature
c         NIO       - unit number of .out file
c         NE        - # of elastic constants --> 9 MAX
c         NV        - # of viscoplastic constants --> 19 MAX
c         NS        - subcell number
c         MN        - material number
c         CDUM      - material character/number designation
c                     (i.e. F1 = fiber #1)
c         DMGF      - damage factor - if damage is included the user
c                     should multiply material stiffness terms by DMGF
c                     when using such terms in his inelastic model.
c
c    expected on exit:
c         DSA       - vector of rate quantities (see above)
************************************************************************


      IF( NE .GT. NEP ) THEN
         CALL FATALERROR(NIO)
         WRITE(NIO, *) ‘TO MANY ELASTIC PROPERTIES FOR MATERIAL # ‘, MN
         WRITE(NIO, *) ‘   # USED      = ‘, NE
         WRITE(NIO, *) ‘   # ALLOCATED = ‘, NEP
         STOP
      ENDIF

      IF( NV .GT. NVP ) THEN
         CALL FATALERROR(NIO)
         WRITE(NIO, *) ‘TO MANY NON-LINEAR PROPERTIES FOR MATERIAL # ‘,
     &        MN
         WRITE(NIO, *) ‘   # USED      = ‘, NV
         WRITE(NIO, *) ‘   # ALLOCATED = ‘, NVP
         STOP
      ENDIF


************************************************************************
*                        BEGIN USER EDITS                              *
************************************************************************
C      WRITE(6, *) ‘ PV=’, PV
C      WRITE(6, *) ‘ PE=’, PE

c-----------------------------------------------------------------------
c   MATERIALS #1 & #2 ARE ELASTIC
c    (set inelastic strain rates to zero)
c-----------------------------------------------------------------------
      IF ((MN .EQ. 1) .OR. (MN .EQ. 2)) THEN
         DSA(13) = 0
         DSA(14) = 0
         DSA(15) = 0
```

```
            DSA(16) = 0
            DSA(17) = 0
            DSA(18) = 0
            RETURN
         ENDIF

c-------------------------------------------------------------------------
c   MATERIAL #3 --> USE BODNER-PARTOM
c-------------------------------------------------------------------------

c--------------------------------------------------------------------------
c  copy appropriate viscoplastic material constants
c--------------------------------------------------------------------------

      IF( NV .LT. 6 ) THEN
         CALL FATALERROR(NIO)
         WRITE(NIO, *) ' NOT ENOUGH PV SPACE: NV =', NV
         STOP
      ENDIF
      D0 = PV(1)
      Z0 = PV(2)
      Z1 = PV(3)
      BM = PV(4)
      AN = PV(5)
      Q = PV(6)
c-------------------------------------------------------------------------
c  copy stress from [sa] to [ss]
c-------------------------------------------------------------------------
      SS(1) = SA(7)
      SS(2) = SA(8)
      SS(3) = SA(9)
      SS(4) = SA(10)
      SS(5) = SA(11)
      SS(6) = SA(12)
c--------------------------------------------------------------------------
c  compute the deviatoric stress [s] in the subcell
c--------------------------------------------------------------------------
      TEMP = (SS(1) + SS(2) + SS(3)) / 3.0
      S(1) = SS(1) - TEMP
      S(2) = SS(2) - TEMP
      S(3) = SS(3) - TEMP
      S(4) = SS(4)
      S(5) = SS(5)
      S(6) = SS(6)
c--------------------------------------------------------------------------
c--------------------------------------------------------------------------
      AJ2 = 0.5 * (S(1)**2 + S(2)**2 + S(3)**2) + S(4)**2 + S(5)**2 +
     &     S(6)**2
      SQ3AJ = DSQRT( SS(1)**2 + SS(2)**2 + SS(3)**2 + 2 * (SS(4)**2 +
     &     SS(5)**2 + SS(6)**2) )
      SQ2 = 1.414215
      IF (SQ3AJ .EQ. 0.0) THEN
         CALL ZEROR(R, 6)
      ELSE
```

```
         R(1) = SS(1) / SQ3AJ
         R(2) = SS(2) / SQ3AJ
         R(3) = SS(3) / SQ3AJ
         R(4) = SQ2 * SS(4) / SQ3AJ
         R(5) = SQ2 * SS(5) / SQ3AJ
         R(6) = SQ2 * SS(6) / SQ3AJ
      ENDIF
c-------------------------------------------------------------------------
c  if d0=0 then assume elastic and zero-out
c  [dsa(13-30)] (inelastic strain rate and
c  internal variable rates), then return
c-------------------------------------------------------------------------
      IF (D0 .EQ. 0) THEN
         DO 100 JJ = 13, 30
            DSA(JJ) = 0.0
  100    CONTINUE
         RETURN
c-------------------------------------------------------------------------
c  inelastic
c-------------------------------------------------------------------------
      ELSE
         ZEF = Z0 + Q * SA(20) + (1 - Q) * (R(1) * SA(21) + R(2) *
     &        SA(22) + R(3) * SA(23) + R(4) * SA(24) + R(5)
     &        * SA(25) + R(6) * SA(26))

         IF (AJ2 .EQ. 0.0) THEN
            AL = 0.0
         ELSE
            ARG1 = ZEF**2.0 / (3.0 * AJ2)

            IF (ARG1 .GT. 1E6) ARG1 = 1E6
            CON = .5 * (AN + 1.0) / AN
            ARG = CON * (ARG1)**AN

            IF (ARG .GT. 50.0) ARG = 50.0
            AL = D0 / (DEXP(ARG) * DSQRT(AJ2))
         ENDIF
c-------------------------------------------------------------------------
c  inelastic strain rates
c-------------------------------------------------------------------------
         DSA(13) = AL * S(1)
         DSA(14) = AL * S(2)
         DSA(15) = AL * S(3)
         DSA(16) = 2 * AL * S(4)
         DSA(17) = 2 * AL * S(5)
         DSA(18) = 2 * AL * S(6)

c-------------------------------------------------------------------------
c  plastic work rate
c-------------------------------------------------------------------------
         WPD = S(1) * DSA(13) + S(2) * DSA(14) + S(3) * DSA(15) + S(4) *
     &        DSA(16) + S(5) * DSA(17) + S(6) * DSA(18)
c-------------------------------------------------------------------------
c  state variable rates
```

```
c-----------------------------------------------------------------------
         DSA(19) = WPD
         Z0M = BM / Z0
         ZD = Z0M * (Z1 - ZEF) * WPD
         DSA(20) = ZD

         DSA(21) = ZD * R(1)
         DSA(22) = ZD * R(2)
         DSA(23) = ZD * R(3)
         DSA(24) = ZD * R(4)
         DSA(25) = ZD * R(5)
         DSA(26) = ZD * R(6)
      ENDIF


************************************************************************
*                         END USER EDITS                               *
************************************************************************

      RETURN
      END
```

The USRFUN subroutine

The USRFUN subroutine is used here to allow material properties to be entered in functional form instead of having to be linearly interpolated within **MAC/GMC**

```
c######################################################################
      SUBROUTINE USRFUN(MN, TIME, TSTEP, CTEMP, DTEMPR, SA, DSA,
     &     DOLD, PEM, PVM, D, LOCTISO, ALPA, ALPT,
     &     NE, NV, NMTS, NEP, NVP, NSASIZE)

c     purpose:  user subroutine to allow elastic and viscoplastic
c               material properties to be functions of TEMP or
c               field variables.  Used for user defined functional
c               form material properties, that is, when:
c               (mat .eq. 'U') .and. (ifm .eq. 2)
c
c     note:     can be used in conjunction with a provided material
c               constitutive model, or a constitutive model input
c               by the user in USRMAT

      IMPLICIT DOUBLE PRECISION (A - H, O - Z)

      DIMENSION DOLD(6, 6), D(3)

      DIMENSION PEM(NEP, NMTS), PVM(NVP, NMTS)
      DIMENSION ALPA(NMTS), ALPT(NMTS)
      DIMENSION DSA(NSASIZE), SA(NSASIZE)
```

```
**********************************************************************
c  note: 1) in this subroutine, [SA] and [DSA] contain the
c           micro (subcell) quantities for aboudi’s micromechanics model
c
c        2) arrangement of [dsa] & [sa]  arrays:
c           variable            location
c         +-------------------------------------------------------------
c         | strain rate         (1-6)  (contains ENGINEERING shears)
c         |-------------------------------------------------------------
c         | stress rate         (7-12)
c         |-------------------------------------------------------------
c         | inelastic
c         | strain rate         (13-18)  (contains ENGINEERING shears)
c         |-------------------------------------------------------------
c         | 12 “slots”          (19-30)
c         | for state variables
c         |-------------------------------------------------------------
c         | thermal strain rate  (31-36)
c         +-------------------------------------------------------------
**********************************************************************
c NOTE:  quantities in [SA] and [DSA] are SUBCELL quantities - the
c        values on entry are for the first subcell containing material
c        # MN - the values on exit of this subroutine will be applied to
c        ALL SUBCELLS containing material # MN.  It is thus recommended
c        that, if using the field variables, you assign the appropriate
c        material # to ONE SUBCELL ONLY.  Use of [SA] and [DSA] in this
c        context in conjunction with bending in laminate theory will
c        result in erroneous results as field variables become dependent
c        on through-thickness position (while the material # does not).
**********************************************************************
c   on entry:
c      MN           - material number
c      TIME         - current time
c      TSTEP        - current time step
c      CTEMP        - current temperature
c      DTEMPR       - time rate of change of temperature
c      SA           - vector of total (integrated) quantities (see above)
c      DSA          - vector of rate quantities (see above)
c      DOLD(6, 6)  - previous elastic material stiffness matrix
c      PEM(NE, MN) - vector of previous elastic constants for material
c                     # MN (where NE = # of elastic constants --> 9 MAX)
c      PVM(NV, MN) - vector of previous viscoplastic constants for
c                     material # MN
c                     (where NV = # of viscoplastic constants --> 19 MAX)
c
c   expected on exit:
c      PEM(NE, MN) - vector of current elastic constants for material MN
c      PVM(NV, MN) - vector of current viscoplastic constants for
c                     material MN
c      D(3)         - vector of direction cosines
c                    (required for models 3, 7, & 9)
c      LOCTISO      - flag indicating if ANY material exhibits local
c                     transverse isotropy (and global anisotropy)
c                   = 0 - all materials are at most globally transversely
```

```
c                         isotropic (D not used)
c                   = 1 - at least one material is locally transversely
c                         isotropic (D used)
c      ALPA(MN)    - longitudinal cte for material MN
c      ALPT(MN)    - transverse cte for material MN
c           NE     - NUMBER OF ELASTIC PROPERTIES USED
c           NV     - NUMBER OF VISCOPLASTIC PROPERTIES USED
c
************************************************************************


************************************************************************
*                       BEGIN USER EDITS                               *
************************************************************************
C      WRITE(6, *) 'TOP OF USRFUN'

c-----------------------------------------------------------------------
c  MATERIAL # 1:  E = FUNCTION OF STRAIN & PREVIOUS E
c                 FOR USE WITH USER CONSTITUTIVE MODEL
c-----------------------------------------------------------------------
      IF (MN .EQ. 1) THEN

         IF (SA(1) .GT. 0) THEN
            EAOLD = PEM(1, MN)
            EA = 700.E9 - EAOLD * SA(1) * 80.0
         ELSE
            EA = 700.E9
         ENDIF

         ET = EA
         FNA = 0.41
         FNT = 0.41
         GA = EA / (2.0 * (1.0 + FNA))
         ALPA(MN) = 4.5E-6
         ALPT(MN) = 4.5E-6

         NE = 5
         PEM(1, MN) = EA
         PEM(2, MN) = ET
         PEM(3, MN) = FNA
         PEM(4, MN) = FNT
         PEM(5, MN) = GA

c-----------------------------------------------------------------------
c  MATERIAL #  2:  PROPERTIES = INTERPOLATED AT INPUT TEMPERATURES
c                  THIS SUBROUTINE IS NOT CALLED FOR MN #2 SINCE THE
c                  MATERIAL PROPERTIES ARE NOT FUNCTIONAL FORM
c-----------------------------------------------------------------------


c-----------------------------------------------------------------------
c  MATERIAL # 3:  PROPERTIES = LINEAR FUNCTION OF TEMPERATURE
c                 FOR USE WITH USER CONSTITUTIVE MODEL
c-----------------------------------------------------------------------
```

```
      ELSEIF (MN .EQ. 3) THEN

         IF (CTEMP .LT. 21.0) CTEMP = 21.0
         IF (CTEMP .GT. 400.) CTEMP = 400.

C -- ELASTIC
         E = 72.4E9 - 81.53E6 * (CTEMP - 21.)
         FN = 0.33 + 7.916E-5 * (CTEMP - 21.)
         ALP = 22.5E-6 + 3.958E-9 * (CTEMP - 21.)

         NE = 5
         PEM(1, MN) = E
         PEM(2, MN) = E
         PEM(3, MN) = FN
         PEM(4, MN) = FN
         PEM(5, MN) = E / (2. * (1. + FN))
         ALPA(MN) = ALP
         ALPT(MN) = ALP

C -- VISCOPLASTIC
         D0 = 1.E4
         Z0 = 340.E6
         Z1 = 435.E6
         BM = 300.0
         AN = 10.0 - 0.02493 * (CTEMP - 21.)
         Q  = 1.0

         NV = 6
         PVM(1, MN) = D0
         PVM(2, MN) = Z0
         PVM(3, MN) = Z1
         PVM(4, MN) = BM
         PVM(5, MN) = AN
         PVM(6, MN) = Q


      ENDIF

************************************************************************
*                         END USER EDITS                               *
************************************************************************

      RETURN
      END
```

The USRFORMDE subroutine

The USRFORMDE subroutine is used here to form the elastic material stiffness matrix in **MAC/GMC** when a user defined constitutive model is used**.**

```
c########################################################################
      SUBROUTINE USRFORMDE(MN, PEM, PVM, D, LOCTISO, DNEW,
     &     NE, NV, NEP, NVP, NMTS)

c     purpose:  user subroutine to allow formation of material stiffness
c               matrices based on a user constitutive model, which may use
c               arbitrary material properties (used when ncmd = 99)
c

      IMPLICIT DOUBLE PRECISION (A - H, O - Z)

      DIMENSION DNEW(6, 6)
      DIMENSION PEM(NEP, NMTS), PVM(NVP, NMTS)
      DIMENSION D(3)

************************************************************************
c   on entry:
c      MN           - material number
c      PEM(NE, MN) - vector of elastic constants for material
c                     # MN (where NE = # of elastic constants --> 9 MAX)
c      PVM(NV, MN) - vector of viscoplastic constants for material # MN
c                     (where NV = # of viscoplastic constants --> 19 MAX)
c      D(3)         - vector of direction cosines
c                    (required for models 3, 7, & 9)
c      LOCTISO      - flag indicating if ANY material exhibits local
C                     transverse isotropy (and global anisotropy)
c                   = 0 - all materials are at most globally transversely
c                         isotropic (D not used)
c                   = 1 - at least one material is locally transversely
c                         isotropic (D used)
c
c   expected on exit:
c      DNEW(6, 6)  - current elastic material stiffness matrix

************************************************************************
*                       BEGIN USER EDITS                               *
************************************************************************
C      WRITE(6, *) 'TOP OF USRFORMDE'

C NOTE:  In the examples shown here, standard engineering material
C        elastic constants (E, nu) are used.  However, the user is
C        free to employ any material elastic constants with his
C        constitutive model (i.e., bulk modulus, etc.).  Thus, the
C        user must provide the equations to determine the stiffness
C        components required by GMC from his elastic constants.

c-----------------------------------------------------------------------
c  MATERIAL # 1:  E = FUNCTION OF STRAIN & PREVIOUS E
```

```
c                  FOR USE WITH USER CONSTITUTIVE MODEL
c------------------------------------------------------------------------
      IF (MN .EQ. 1) THEN

         EA = PEM(1, MN)
         ET = PEM(2, MN)
         FNA = PEM(3, MN)
         FNT = PEM(4, MN)
         GA = PEM(5, MN)

         DO 100 I = 1, 6
            DO 100 J = 1, 6
               DNEW(I, J) = 0
 100     CONTINUE

         GT = 0.5 * ET / (1 + FNT)
         FK = 0.25 * EA / (0.5 * (1 - FNT) * (EA / ET) - FNA**2)

         DNEW(1, 1) = EA + 4.0 * FK * FNA**2
         DNEW(2, 1) = 2.0 * FK * FNA
         DNEW(3, 1) = 2.0 * FK * FNA

         DNEW(1, 2) = 2.0 * FK * FNA
         DNEW(2, 2) = FK + GT
         DNEW(3, 2) = FK - GT

         DNEW(1, 3) = 2.0 * FK * FNA
         DNEW(2, 3) = FK - GT
         DNEW(3, 3) = FK + GT

         DNEW(4, 4) = GT
         DNEW(5, 5) = GA
         DNEW(6, 6) = GA

c------------------------------------------------------------------------
c  MATERIAL #  2:  PROPERTIES = INTERPOLATED AT TEMPERATURES
c                   FOR USE WITH USER CONSTITUTIVE MODEL
c------------------------------------------------------------------------
      ELSEIF (MN .EQ. 2) THEN

         E = PEM(1, MN)
         FN = PEM(2, MN)
         GA = E / (2. * (1. + FN))

         DO 320 I = 1, 6
            DO 320 J = 1, 6
               DNEW(I, J) = 0
 320     CONTINUE

         GT = 0.5 * E / (1 + FN)
         FK = 0.25 * E / (0.5 * (1 - FN) - FN**2)

         DNEW(1, 1) = E + 4.0 * FK * FN**2
         DNEW(2, 1) = 2.0 * FK * FN
```

```
         DNEW(3, 1) = 2.0 * FK * FN

         DNEW(1, 2) = 2.0 * FK * FN
         DNEW(2, 2) = FK + GT
         DNEW(3, 2) = FK - GT

         DNEW(1, 3) = 2.0 * FK * FN
         DNEW(2, 3) = FK - GT
         DNEW(3, 3) = FK + GT

         DNEW(4, 4) = GT
         DNEW(5, 5) = GA
         DNEW(6, 6) = GA

c------------------------------------------------------------------------
c  MATERIAL # 3:  PROPERTIES = LINEAR FUNCTION OF TEMPERATURE
c                 FOR USE WITH USER CONSTITUTIVE MODEL
c------------------------------------------------------------------------
      ELSEIF (MN .EQ. 3) THEN

         E = PEM(1, MN)
         FN = PEM(3, MN)
         GA = PEM(5, MN)

         DO 300 I = 1, 6
            DO 300 J = 1, 6
               DNEW(I, J) = 0
 300     CONTINUE

         GT = 0.5 * E / (1 + FN)
         FK = 0.25 * E / (0.5 * (1 - FN) - FN**2)

         DNEW(1, 1) = E + 4.0 * FK * FN**2
         DNEW(2, 1) = 2.0 * FK * FN
         DNEW(3, 1) = 2.0 * FK * FN

         DNEW(1, 2) = 2.0 * FK * FN
         DNEW(2, 2) = FK + GT
         DNEW(3, 2) = FK - GT

         DNEW(1, 3) = 2.0 * FK * FN
         DNEW(2, 3) = FK - GT
         DNEW(3, 3) = FK + GT

         DNEW(4, 4) = GT
         DNEW(5, 5) = GA
         DNEW(6, 6) = GA

      ENDIF
************************************************************************
*                         END USER EDITS                               *
************************************************************************
       RETURN
      END
```

The USRCPEVAL subroutine

The USRCPEVAL subroutine is used here to form the time derivative of the material stiffness matrix in **MAC/GMC**

```
c#########################################################################
      SUBROUTINE USRCPEVAL(DSA, SA, MN, TIME, TSTEP, CTEMP, DTEMPR,
     &     DNEW, DOLD, PEM, PVM, D, LOCTISO, ALPA, ALPT, DDOT,
     &     NE, NV, NMTS, NEP, NVP, NSASIZE)


c     purpose:  user subroutine to allow formation of the TIME
c               derivative of the material stiffness matrix.
c               this subroutine is used when:
c               a) material properties are user defined and functional
c                  form.  That is:  (mat .eq. ‘U’) .and. (ifm .eq. 2)
c               b) the constitutive model is user-defined, and the
c                  material properties are not functional form, and
c                  the material properties are temperature-dependent
c                  that is: (ncmd .eq. 99) .and. (ifm .ne. 2) .and.
c                  (ndpt .eq. 2)

      IMPLICIT DOUBLE PRECISION (A - H, O - Z)


      DIMENSION DNEW(6, 6), DOLD(6, 6)
      DIMENSION DDOT(6, 6)

      DIMENSION PEM(NEP, NMTS), PVM(NVP, NMTS)
      DIMENSION ALPA(NMTS), ALPT(NMTS)
      DIMENSION DSA(NSASIZE), SA(NSASIZE)

************************************************************************
c  note: 1) in this subroutine, [SA] and [DSA] contain the
c           micro (subcell) quantities for aboudi’s micromechanics model
c
c        2) arrangement of [dsa] & [sa]  arrays:
c           variable             location
c         +-------------------------------------------------------------
c         | strain rate          (1-6)  (contains ENGINEERING shears)
c         |-------------------------------------------------------------
c         | stress rate          (7-12)
c         |-------------------------------------------------------------
c         | inelastic
c         | strain rate          (13-18)  (contains ENGINEERING shears)
c         |-------------------------------------------------------------
c         | 12 “slots”           (19-30)
c         | for state variables
c         |-------------------------------------------------------------
c         | thermal strain rate  (31-36)
c         +-------------------------------------------------------------
************************************************************************
c   on entry:
c      SA          - vector of total (integrated) quantities (see above)
```

```
c      DSA         - vector of rate quantities (see above)
c      MN          - material number
c      TIME        - current time
c      TSTEP       - current time step
c      CTEMP       - current temperature
c      DTEMPR      - time rate of change of temperature
c      DNEW(6, 6)  - current elastic material stiffness matrix
c      DOLD(6, 6)  - previous elastic material stiffness matrix
c      PEM(NE, MN) - vector of current elastic constants for material MN
c                    (where NE = # of elastic constants --> 9 MAX)
c      PVM(NV, MN) - vector of current viscoplastic constants for
c                    material MN
c                    (where NV = # of elastic constants --> 19 MAX)
c      D(3)        - vector of direction cosines
c                    (required for models 3, 7, & 9) local
c      LOCTISO     - flag indicating if ANY material exhibits
c                    transverse isotropy (and global anisotropy)
c                  = 0 - all materials are at most globally transversely
c                        isotropic (D not used)
c                  = 1 - at least one material is locally transversely
c                        isotropic (D used)
c      ALPA(MN)    - longitudinal cte for material MN
c      ALPT(MN)    - transverse cte for material MN
c
c   expected on exit:
c      DDOT(6, 6)  - derivative with respect to TIME of stiffness matrix
************************************************************************


************************************************************************
*                      BEGIN USER EDITS                                *
************************************************************************

      DIMENSION DHOLD(6, 6, 3), DNEW1(6, 6), PEM1(9, 10), PVM1(19, 10)
      DIMENSION ALPA1(10), ALPT1(10)

C      WRITE(6, *) ‘TOP OF USRCPEVAL’


c-----------------------------------------------------------------------
c  MATERIAL # 1:  E = FUNCTION OF STRAIN & PREVIOUS E
c                 FOR USE WITH USER CONSTITUTIVE MODEL
c-----------------------------------------------------------------------
      IF (MN .EQ. 1) THEN

c --- method one - simply divide change in stiffness components by TSTEP
         DO 200 I = 1, 6
            DO 200 J = 1, 6
               DDOT(I, J) = (DNEW(I, J) - DOLD(I, J)) / TSTEP
  200    CONTINUE

c-----------------------------------------------------------------------
c  MATERIAL #  2:  PROPERTIES = INTERPOLATED AT TEMPERATURES
c                  FOR USE WITH USER CONSTITUTIVE MODEL
```

```
c-----------------------------------------------------------------------
      ELSEIF (MN .EQ. 2) THEN

c --- method 2 - if material props are only function of temperature,
c --- calculate stiffness at  CTEMP - 0.5 & CTEMP + 0.5, difference
c --- equals change per temp, multiply by DTEMPR equals change per time
         IF (CTEMP .LT. 21.0) CTEMP = 21.0
         IF (CTEMP .GT. 400.) CTEMP = 400.

         DO 300 I = 1, 6
            DO 300 J = 1, 6
               DO 300 K = 1, 2
                  DHOLD(I, J, K) = 0
 300     CONTINUE

         DO 420 K = 1, 2

            IF (K .EQ. 1) CTEMP1 = CTEMP - 0.5
            IF (K .EQ. 2) CTEMP1 = CTEMP + 0.5

c --- this subroutine interpolates properties for material MN at
c --- the temperature CTEMP1
            CALL INTTMP(MN, CTEMP1, PEM1, ALPA1, ALPT1, PVM1)

c --- calculate the stiffness components
            CALL USRFORMDE(MN, PEM1, PVM, D, LOCTISO, DNEW1,
     &           NE, NV, NEP, NVP, NMTS)
c --- note: for internal constitutive models (ncmd .ne. 99), use
c --- subroutine FORMDE to calculate stiffness components.
c --- syntax: CALL FORMDE(MN, PEM1, D)

            DO 480 I = 1, 6
               DO 480 J = 1, 6
                  DHOLD (I, J, K) = DNEW1(I, J)
  480       CONTINUE
  420    CONTINUE

         DO 520 I = 1, 6
            DO 520 J = 1, 6
               DDOT(I, J) = (DHOLD(I, J, 2) - DHOLD(I, J, 1)) * DTEMPR
  520    CONTINUE

c-----------------------------------------------------------------------
c  MATERIAL # 3:  PROPERTIES = LINEAR FUNCTION OF TEMPERATURE
c                 FOR USE WITH USER CONSTITUTIVE MODEL
c-----------------------------------------------------------------------
      ELSEIF (MN .EQ. 3) THEN

c --- method 2 - if material props are only function of temperature,
c --- calculate stiffness at  CTEMP - 0.5 & CTEMP + 0.5, difference
c --- equals change per temp, multiply by DTEMPR equals change per time
         IF (CTEMP .LT. 21.0) CTEMP = 21.0
         IF (CTEMP .GT. 400.) CTEMP = 400.
```

```
         DO 330 I = 1, 6
            DO 330 J = 1, 6
               DO 330 K = 1, 2
                  DHOLD(I, J, K) = 0
 330     CONTINUE

         DO 400 K = 1, 2

            IF (K .EQ. 1) CTEMP1 = CTEMP - 0.5
            IF (K .EQ. 2) CTEMP1 = CTEMP + 0.5

c --- determine material props at the appropriate temperature
            CALL USRFUN(MN, TIME, TSTEP, CTEMP1, DTEMPR, SA, DSA,
     &           DOLD, PEM1, PVM1, D, LOCTISO, ALPA1, ALPT1,
     &           NE, NV, NMTS, NEP, NVP, NSASIZE)

c --- calculate the stiffness components
            CALL USRFORMDE(MN, PEM1, PVM, D, LOCTISO, DNEW1,
     &           NE, NV, NEP, NVP, NMTS)
c --- note: for internal constitutive models (ncmd .ne. 99), use
c --- subroutine FORMDE to calculate stiffness components.
c --- syntax: CALL FORMDE(MN, PEM1, D)

            DO 450 I = 1, 6
               DO 450 J = 1, 6
                  DHOLD (I, J, K) = DNEW1(I, J)
  450       CONTINUE
  400    CONTINUE

         DO 500 I = 1, 6
            DO 500 J = 1, 6
               DDOT(I, J) = (DHOLD(I, J, 2) - DHOLD(I, J, 1)) * DTEMPR
  500    CONTINUE


      ENDIF
************************************************************************
*                        END USER EDITS                                *
************************************************************************

      RETURN
      END
```