Demonstration Problems

To demonstrate the various features of this user-defined material model implemented as a UMAT subroutine within ABAQUS/Standard, a two-dimensional rectangular panel, shown in Figure 3, subjected to uniform extension is solved using a single 4-node flat quadrilateral finite element. A right-hand coordinate system is used with positive z directed out of the paper and toward the reader. This problem simulates a uniaxial response. The dimensions of the panel are 4-inches by 8-inches and a total thickness of 0.2 inches. The loading is an applied edge displacement in the long (positive x) direction of the panel. The boundary conditions provide restraint from rigid body motion and permit contraction in the shorter (y) dimension to prevent the development of stresses transverse to the loading direction (*i.e.,* develop a uniaxial stress

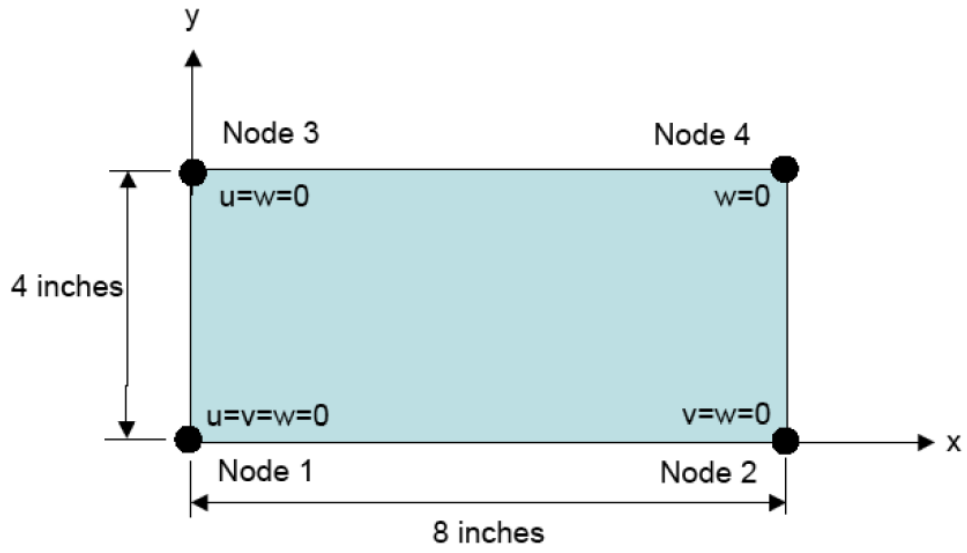

Figure 3. Geometry and boundary conditions for the demonstration problem.

Appendix D – Representative ABAQUS/Standard Input File for the Demonstration Problems

The following records define the ABAQUS/Standard finite element model for the demonstration problem used to illustrate this UMAT subroutine's capabilities and features. It involves a single 4-node shell element with boundary conditions and loading resulting in a uniaxial uniform stress condition.

```
*HEADING
**
*NODE
       1,            0.,            0.
       2,            0.,           4.0
       3,            8.,            0.
       4,            8.,           4.0
**
*ELEMENT, TYPE=S4R, ELSET=MODEL
       1,       1,       3,       4,     2
**
**  DEMOMAT1 = linear elastic brittle
**  DEMOMAT2 = nonlinear elastic brittle
**
*SHELL SECTION, COMPOSITE, ELSET=MODEL
     0.1,    1,     DEMOMAT2,     0.0
     0.1,    1,     DEMOMAT2,     0.0
*TRANSVERSE SHEAR STIFFNESS
1.0E6, 1.0E6
**
** ================================================================================================
** ================================================================================================
** UMAT Property Data Definitions
** props(1-8):E11t,E22t,E33t,E11c,E22c,E33c,G12,G13,
** props(9-16):G23,nu12,nu13,nu23, Xt, Yt, Zt, Xc,
** props(17-24):Yc,Zc,S12,S13,S23,Eps11T,Eps22T,Eps33T,
** props(25-32):Eps11C,Eps22C,Eps33C,Gam12,Gam13,Gam23,Eps11Tmx,Eps22Tmx,
** props(33-40):Eps33Tmx,Eps11Cmx,Eps22Cmx,Eps33Cmx,Gam12mx,Gam13mx,Gam23mx,GIc,
** props(41-48):FPZ,SlimT,SlimC,SlimS,weibull(1),weibull(2),weibull(3),weibull(4),
** props(49-55):weibull(5),weibull(6),Dgrd(1),Dgrd(2),Dgrd(3),RECURS,PDA
** ================================================================================================
** Demo problem material definition #1, total thickness =  0.20 inches
**   - Linear elastic brittle
*MATERIAL, NAME=DEMOMAT1
*USER MATERIAL,  CONSTANTS=55
 2.000E+06, 2.000E+06, 2.000E+06, 4.500E+06, 4.500E+06, 4.500E+06, 0.800E+06, 0.800E+06,
 0.800E+06, 2.500E-01, 2.500E-01, 2.500E-01, 7.000E+03, 7.000E+03, 7.000E+03, 1.800E+04,
 1.800E+04, 1.800E+04, 6.200E+03, 6.200E+03, 6.200E+03, 3.500E-03, 3.500E-03, 3.500E-03,
 4.000E-03, 4.000E-03, 4.000E-03, 7.750E-03, 7.750E-03, 7.750E-03, 0.200E-01, 0.200E-01,
 0.200E-01, 0.200E-01, 0.200E-01, 1.000E-01, 1.000E-01, 1.000E-01, 1.000E-01, 3.000E+01,
 2.000E-01, 0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00,
 0.000E+00, 0.000E+00, 1.000E-06, 1.000E-06, 1.000E-06, 1.000E+00, 1.000E+00
*DEPVAR
   8
** ================================================================================================
** Demo problem material definition #2, total thickness =  0.20 inches
**   - Nonlinear elastic brittle
*MATERIAL, NAME=DEMOMAT2
*USER MATERIAL,  CONSTANTS=55
 2.000E+06, 2.000E+06, 2.000E+06, 4.500E+06, 4.500E+06, 4.500E+06, 0.800E+06, 0.800E+06,
 0.800E+06, 2.500E-01, 2.500E-01, 2.500E-01, 7.000E+03, 7.000E+03, 7.000E+03, 1.800E+04,
 1.800E+04, 1.800E+04, 6.200E+03, 6.200E+03, 6.200E+03, 6.000E-03, 6.000E-03, 6.000E-03,
 5.800E-03, 5.800E-03, 5.800E-03, 7.750E-03, 7.750E-03, 7.750E-03, 0.200E-01, 0.200E-01,
 0.200E-01, 0.200E-01, 0.200E-01, 1.000E-01, 1.000E-01, 1.000E-01, 1.000E-01, 3.000E+01,
 2.000E-01, 0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00, 0.000E+00,
 0.000E+00, 0.000E+00, 1.000E-06, 1.000E-06, 1.000E-06, 1.000E+00, 2.000E+00
```

```
*DEPVAR
  8
** ==========================================================================================
**
**  Solution step 1
**
*STEP,NLGEOM, INC=10000
*STATIC
0.01, 1.0, 1.0e-04, 0.01
*CONTROLS, PARAMETERS= TIME INCREMENTATION
 10,10,10,10,10,, , 10, 10
**
**  Boundary conditions
**
*NSET, NSET=UVWDIS
       1,
*NSET, NSET=UWDIS
       2,
*NSET, NSET=DIS, GENERATE
       3,       4,       1
**
** uvwdis
**
*BOUNDARY, OP=MOD
UVWDIS, 1,,          0.
UVWDIS, 2,,          0.
UVWDIS, 3,,          0.
**
** uwdis
**
*BOUNDARY, OP=MOD
UWDIS, 1,,          0.
UWDIS, 3,,          0.
**
** dis - imposed displacement
**
*BOUNDARY, OP=MOD
DIS, 1,,        0.080
**
*OUTPUT, FIELD, FREQ=1
*NODE OUTPUT
 U,
*ELEMENT OUTPUT
 S,
 E,
 SDV,
*OUTPUT, HISTORY, FREQ=1
*NODE OUTPUT, NSET=DIS
 U,
 RF,
*ELEMENT OUTPUT, ELSET=MODEL
 E,
 S,
 SDV,
*NODE FILE, FREQ=1
U,
**
*EL FILE, FREQ=1
E,
S,
*END STEP
```