APPENDIX K: Interactive Material Model Driver

INTRODUCTION

The interactive material model driver in LS-DYNA allows calculation of the material constitutive response to a specified strain path. Since the constitutive model subroutines in LS-DYNA are directly called by this driver, the behavior of the constitutive model is precisely that which can be expected in actual applications. In the current implementation the constitutive subroutines for both shell elements and solid elements can be examined.

INPUT DEFINITION

The material model driver is invoked when no *NODE or *ELEMENT commands are present in a standard LS-DYNA input file. The number of material model definitions should be set to one, the number of load curves should be nine, and the termination time to the desired length of the driver run. The complete state dump interval as given in *DATABASE_BINARY_D3PLOT serves as the time step to be used in the material model driver run. Plotting information is saved in core for the interactive plotting phase.

The input deck typically consists only of *KEYWORD, *DATABASE_BINARY_D3PLOT, *CONTROL_TERMINATION, one each of *PART/*MAT/*SECTION, and nine load curves (*DEFINE_CURVE) describing the strain path. These nine curves define the time history of the displacement gradient components shown in Table 61-1.

The velocity gradient matrix, L_{ij}, is approximated by taking the time derivative of the components in Table 61-1. If these components are considered to form a tensor S_{ij} , then

$$L_{ij}(t) = \frac{S_{ij}(t) - S_{ij}(t_{k-1})}{(t - t_k)}$$

and the strain rate tensor is defined as

$$d_{ij} = \frac{L_{ij} + L_{ij}^t}{2}$$

and the spin tensor as

$$\omega_{ij} = \frac{L_{ij} - L_{ij}^t}{2}$$

Load Curve Number	Component Definition
1	$\frac{\partial u}{\partial x}$
2	$\frac{\partial v}{\partial y}$
3	$\frac{\partial w}{\partial z}$
4	$\frac{\partial u}{\partial y}$
5	$\frac{\partial v}{\partial x}$
6	$\frac{\partial u}{\partial z}$
7	$\frac{\partial w}{\partial x}$
8	$\frac{\partial v}{\partial z}$
9	$\frac{\partial w}{\partial y}$

Table 61-1. Load Curve Definitions versus Time

INTERACTIVE DRIVER COMMANDS

After reading the input file and completing the calculations, LS-DYNA gives a command prompt to the terminal. A summary of the available interactive commands is given below. An on-line help package is available by typing HELP. Only available in Unix and Linux.

ACCL	Scale all abscissa data by f. Default is f = 1.
ASET amin omax	Set min and max values on abscissa to amin and amax, respectively. If amin = amax = 0, scaling is automatic.
CHGL n	Change label for component n. LS-DYNA prompts for new label.

CONTINUE	Re-analyze material model.
CROSS c_1 c_2	Plot component c_1 versus c_2.
ECOMP	Display component numbers on the graphics display: EQ.1: x-stress, EQ.2: y-stress, EQ.3: z-stress, EQ.4: xy-stress, EQ.5: yz-stress, EQ.6: zx-stress, EQ.7: effective plastic strain, EQ.8: pressure, EQ.9: von Mises (effective) stress, EQ.10: 1st principal deviatoric stress, EQ.11: 2nd principal deviatoric stress, EQ.12: 3rd principal deviatoric stress, EQ.13: maximum shear stress, EQ.14: 1st principal stress, EQ.15: 2nd principal stress, EQ.16: 3rd principal stress, EQ.17: ln (v/v0), EQ.18: relative volume, EQ.19: v0/v - 1.0, EQ.20: 1st history variable, EQ.21: 2nd history variable. Adding 100 or 400 to component numbers 1-16 yields strains and strain rates, respectively.
FILE name	Change pampers filename to name for printing.
GRID	Graphics displays will be overlaid by a grid of orthogonal lines.
NOGRID	Graphics displays will not be overlaid by a grid of orthogonal lines.
OSCL	Scale all ordinate data by f. Default is f = 1.
OSET omin omax	Set min and max values on ordinate to omin and omax, respectively. If omin = omax = 0, scaling is automatic.

PRINT	Print plotted time history data into file "pampers." Only data plotted after this command is printed. File name can be changed with the "file" command.
QUIT, END, T	Exit the material model driver program.
RDLC m n r_1 z_1 ... r_n z_n	Redefine load curve m using n coordinate pairs (r_1,z_1) (r_2,z_2),...(r_n,z_n).
TIME c	Plot component c versus time.
TV n	Use terminal output device type n. LS-DYNA provides a list of available devices.

Presently, the material model drive is implemented for solid and shell element material models. The driver does not yet support material models for beam elements.

Use the keyword *CONTROL_MPP_MATERIAL_MODEL_DRIVER and run the input deck only on one processor if a distributed memory executable (MPP) is used.

Beginning with Release R11, it is possible to use the material model driver in batch mode. That means if a file called "mmd.bat" exists in the working directory, then the commands contained therein get executed sequentially without opening an interactive command prompt. Supported commands are print, cross, time, and quit. Example for contents of "mmd.bat" to get effective stress versus plastic strain in a file called "sigeps" and effective stress versus time in "sigtim" is given here (read format '(a8,2x,2e10.0)'):

```
print
sigeps
cross              9         7
print
sigtim
time               9
quit
```