

ARMY RESEARCH LABORATORY



LAMPAT and LAMPATNL User's Manual

by Jeffrey M. Staniszewski and Travis A. Bogetti

ARL-TR-6188

September 2012

NOTICES

Disclaimers

The findings in this report are not to be construed as an official Department of the Army position unless so designated by other authorized documents.

Citation of manufacturer's or trade names does not constitute an official endorsement or approval of the use thereof.

Destroy this report when it is no longer needed. Do not return it to the originator.

Army Research Laboratory

Aberdeen Proving Ground, MD 21005-5066

ARL-TR-6188

September 2012

LAMPAT and LAMPATNL User's Manual

Jeffrey M. Staniszewski and Travis A. Bogetti
Weapons and Materials Research Directorate, ARL

REPORT DOCUMENTATION PAGE				Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing the burden, to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number. PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.					
1. REPORT DATE (DD-MM-YYYY) September 2012		2. REPORT TYPE Final		3. DATES COVERED (From - To) September 2008–October 2011	
4. TITLE AND SUBTITLE LAMPAT and LAMPATNL User's Manual				5a. CONTRACT NUMBER 1120-1120-99	
				5b. GRANT NUMBER	
				5c. PROGRAM ELEMENT NUMBER	
6. AUTHOR(S) Jeffrey M. Staniszewski and Travis A. Bogetti				5d. PROJECT NUMBER H42	
				5e. TASK NUMBER	
				5f. WORK UNIT NUMBER	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) U.S. Army Research Laboratory ATTN: RDRL-WMM-B Aberdeen Proving Ground, MD 21005-5066				8. PERFORMING ORGANIZATION REPORT NUMBER ARL-TR-6188	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)				10. SPONSOR/MONITOR'S ACRONYM(S)	
				11. SPONSOR/MONITOR'S REPORT NUMBER(S)	
12. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release; distribution is unlimited.					
13. SUPPLEMENTARY NOTES					
14. ABSTRACT LAMPAT and LAMPATNL are design tools developed for analyzing thick-section composite structures with progressive failure and material nonlinearity. These tools are implemented as subroutines in the finite element software ABAQUS. This user's manual provides information on the proper use of these subroutines, providing specific details on their setup and output. This manual is the primary documentation for users who wish to use LAMPAT or LAMPATNL to model thick-section composites.					
15. SUBJECT TERMS composite laminate, progressive ply failure, multi-scale, nonlinear response, composite design, thick section					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT UU	18. NUMBER OF PAGES 26	19a. NAME OF RESPONSIBLE PERSON Travis A. Bogetti
a. REPORT Unclassified	b. ABSTRACT Unclassified	c. THIS PAGE Unclassified			19b. TELEPHONE NUMBER (Include area code) 302-831-6324

Contents

List of Figures	v
List of Tables	v
1. Introduction	1
1.1 Description of Output Parameters for LAMPATNL.....	2
1.2 Description of Output Parameters for LAMPAT	5
2. Material Database Format	5
2.1 Header	6
2.2 Region Data.....	7
2.3 Material/Ply Properties.....	7
3. LAMPAT (UARM)	8
3.1 Material Definition.....	8
3.2 Step and Field Output Requests	9
3.3 Element Compatibility	9
3.4 Special Considerations	9
4. LAMPATNL (UMAT)	9
4.1 Material Definition.....	10
4.2 Step and Field Output Requests	10
4.3 Element Compatibility	10
4.4 Special Considerations	10
5. LAMPAT (VUSDFLD)	11
5.1 Material Definition.....	11
5.2 Step and Field Output Requests	12
5.3 Element Compatibility	12
5.4 Special Considerations	12

6. LAMPATNL (VUMAT)	12
6.1 Material Definition	12
6.2 Step and Field Output Requests	13
6.3 Element Compatibility	13
6.4 Special Considerations	13
 Appendix. Example Material Database	 15
 Distribution List	 18

List of Figures

Figure 1. Relation of global coordinates to laminate and ply coordinates.....	4
Figure 2. Example of material database format.	6

List of Tables

Table 1. Selection of software product and subroutine.....	1
Table 2. List of state dependent parameters.....	2

INTENTIONALLY LEFT BLANK.

1. Introduction

The LAMPAT and LAMPATNL user's manual details how to properly use the available LAMPAT and LAMPATNL subroutines, including formatting the material database file, specifics on the material definition, step and output setup, special considerations, and element compatibility for each subroutine. This manual assumes that the user can do all the other additional steps necessary to create the model, including creating the part, creating the material, creating and assigning the section, meshing the part, creating an assembly and steps, applying loads and boundary conditions, and creating and running a job.

Prior to running LAMPAT or LAMPATNL, one must first answer two questions: (1) Do I want to run linear, uncoupled, progressive failure analysis or nonlinear, coupled, progressive failure analysis and (2) is the model I am going to be running a static analysis or a dynamic analysis?

The answer to the first question determines whether to run LAMPAT (linear) or LAMPATNL (nonlinear). The answer to the second question leads the user either to UVARM/UMAT (static) or VUSDFLD/VUMAT (dynamic). Table 1 illustrates the choices and their corresponding subroutines. The chosen LAMPAT or LAMPATNL subroutine must be linked to the job created for the model either through the General tab of the Edit Job dialog box in Abaqus/CAE or the command line with user=(subroutine filename).

Table 1. Selection of software product and subroutine.

	Static Analysis With Abaqus/Standard	Dynamic Analysis With Abaqus/Explicit
Linear, uncoupled progressive failure	Title: LAMPAT	Title: LAMPAT
	Subroutine: UVARM	Subroutine: VUSDFLD
Nonlinear, uncoupled progressive failure	Title: LAMPATNL	Title: LAMPATNL
	Subroutine: UMAT	Subroutine: VUMAT

A detailed report on LAMPAT, including theoretical background and example cases, are provided by Bogetti et al¹. A detailed report on LAMPATNL, with theoretical background, details on implementation, a design methodology that can be used with LAMPATNL, and two example cases are provided in Staniszewski². A full summary and description of the output parameters created by LAMPAT and LAMPATNL are presented in those references, and provided here for completeness.

¹ Bogetti, T. A.; Hoppel, C.P.R.; Burns, B. P. *LAMPAT: A Software Tool for Analyzing and Designing Thick. Laminated Composite Structures*; ARL-TR-0890; U.S. Army Research Laboratory: Aberdeen Proving Ground, MD, September 1995.

² Staniszewski, J. M, *An Improved Design Methodology for Modeling Thick-section Composite Structures Using a Multi-scale Approach*; Master's Thesis, University of Delaware, July 2010.

1.1 Description of Output Parameters for LAMPATNL

A full list and brief description of each of the 16 output parameters of LAMPATNL is provided in table 2. These history-dependent parameters are calculated and stored throughout the incremental solution.

Table 2. List of state dependent parameters.

Abaqus	Parameter Name	Description
SDV1	CXXR	C_{XX} Ratio, X stiffness
SDV2	CYYR	C_{YY} Ratio, Y stiffness
SDV3	CZZR	C_{ZZ} Ratio, Z stiffness
SDV4	CYZR	C_{YZ} Ratio, YZ stiffness
SDV5	CXZR	C_{XZ} Ratio, XZ stiffness
SDV6	CXYR	C_{XY} Ratio, XY stiffness
SDV7	NOF	Number of failures
SDV8	CMODE	Current critical failure mode
SDV9	FMODE	Previous failed mode
SDV10	CPLY	Current critical ply number
SDV11	FPLY	Previous failed ply number
SDV12	SF	Safety factor
SDV13	SUMCIJ	Sum of SDV1 through SDV6
SDV14	PRODCIJ	Produce of SDV1 through SDV6
SDV15	MINCIJ	Minimum value of SDV1 through SDV6
SDV16	MINCIJN	C_{ij} number (1 to 6) of SDV15

The first six parameters are stiffness ratios evaluated directly from the elemental (global) stiffness matrix. At the beginning of the finite element analysis, the values on the diagonal of the stiffness matrix, C_{ij} , are stored. These values change throughout the analysis due to material nonlinearity and progressive ply failure. The stiffness matrix is updated and the current values occupying the diagonal are divided by the original values to determine the stiffness ratio for that direction. Equation 1 shows the calculation of the CXX stiffness ratio CXXR, with the calculation of the five other ratios following similarly.

$$CXXR = \frac{C_{XX}^c}{C_{XX}^i} \quad (1)$$

In equation 1, the c term is the current value and the i term is the initial value. For this analysis, CXX refers to value in the (1, 1) position of the global C_{ij} stiffness matrix, CYY in the (2, 2), CZZ in the (3, 3), CYZ in the (4, 4), CXZ in the (5, 5), and CXY in the (6, 6). The values of the stiffness ratios range from 1 to 0, where a value of 1 indicates no stiffness loss from the original stiffness and 0 indicates complete loss of stiffness of the element in the direction.

The parameter for the number of failures, NOF or SDV7, records the number of ply failures that have occurred in the element. For example, if a (0°/90°)s laminate had transverse tensile failure

in the 90° plies and longitudinal tensile failure in the 0° plies the number of failures would be 4. This value is a result of symmetry with a single failure recorded in the two 90° plies and another single failure recorded in the two 0° plies. If failure has not occurred in the element, NOF will be at zero. This parameter does not indicate the effect of ply failure on the stiffness of the sample but is a count of the number of failure allowables exceeded in the element up to this point.

The critical mode CMODE, previous failed mode FMODE, critical ply CPLY, previous failed ply FPLY, and safety factor SF are important parameters used to determine the progressive ply failure history in LAMPATNL. The safety factor is defined as the ratio of the principal material direction strain allowable to the current ply level strain, as shown below:

$$SF_1 = \frac{Y1T}{\varepsilon_1} \quad \text{if } \varepsilon_1 > 0 \quad (2)$$

$$SF_2 = \left| \frac{Y1C}{\varepsilon_1} \right| \quad \text{if } \varepsilon_1 < 0 \quad (3)$$

$$SF_3 = \frac{Y2T}{\varepsilon_2} \quad \text{if } \varepsilon_2 > 0 \quad (4)$$

$$SF_4 = \left| \frac{Y2C}{\varepsilon_2} \right| \quad \text{if } \varepsilon_2 < 0 \quad (5)$$

$$SF_5 = \frac{Y3T}{\varepsilon_3} \quad \text{if } \varepsilon_3 > 0 \quad (6)$$

$$SF_6 = \left| \frac{Y3C}{\varepsilon_3} \right| \quad \text{if } \varepsilon_3 < 0 \quad (7)$$

$$SF_7 = \left| \frac{Y23}{\varepsilon_4} \right| \quad (8)$$

$$SF_8 = \left| \frac{Y13}{\varepsilon_5} \right| \quad (9)$$

$$SF_9 = \left| \frac{Y12}{\varepsilon_6} \right| \quad (10)$$

where Y1T, Y1C, Y2T, Y2C, Y3T, Y3C, Y23, Y13, and Y12 are the maximum strain allowables. These calculations are done for all N plies in the laminate. The lowest value out of the 9N safety factors is recorded and displayed as the SF output parameter. The ply with the

lowest safety factor becomes the critical ply CPLY and the mode in which this safety factor has occurred is recorded as the critical mode CMODE. The values of CPLY range from 1 to N. The values of CMODE correspond to the number of the lowest safety factor, as defined in equations 2 to 10, and range from 1 to 9. Critical or failure mode 1 refers to 1-direction fiber tension, mode 2 is 1-direction fiber compression, mode 3 is 2-direction matrix tension, mode 4 is 2-direction matrix compression, mode 5 is 3-direction matrix tension, mode 6 is 3-direction matrix compression, mode 7 is interlaminar (23) shear, mode 8 is interlaminar (13) shear, and mode 9 is in-plane (12) shear. The failure modes refer to the ply-level coordinate system that is related to the laminate and global systems, as shown in figure 1.

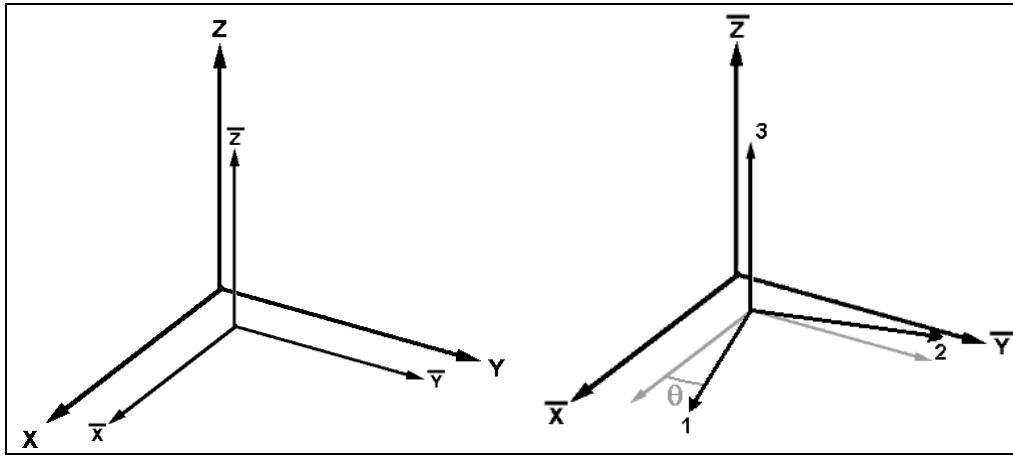


Figure 1. Relation of global coordinates to laminate and ply coordinates

When a strain in a ply has reached its allowable (safety factor=1), the failed ply and direction are discounted and excluded from the safety factor calculation. The critical ply at this point is recorded as the previous failed ply number FPLY and the critical mode becomes the previous failed mode FMODE. These values are initially at zero and will take on the range of values of CPLY and CMODE if ply failure occurs in the element. If failure is recorded, the FPLY and FMODE parameters will portray the progression of failure in the analysis.

The SUMCIJ parameter is simply the sum of the CXXR, CYXR, CZZR, CYZR, CXZR, and CXYR stiffness ratio parameters and can range from 6 to 0. The PRODCIJ parameter is the product of these stiffness ratios and ranges from 1 to 0. The MINCIJ parameter records the minimum value of the six stiffness ratios and also ranges from 1 to 0. The MINCIJN parameter indicates which stiffness ratio is recorded as the minimum for the element. If CXXR is the minimum stiffness ratio, MINCIJN is 1, for CYXR it is 2, for CZZR it is 3, for CYZR it is 4, for CXZR it is 5, and for CXYR it is 6. If the minimum stiffness ratio for the element is greater than 80%, no value for MINCIJN is recorded and this parameter is set to zero. Some output parameters are not used in the following case studies but may have more important uses in different analyses.

1.2 Description of Output Parameters for LAMPAT

The LAMPAT subroutines produce only 3 output parameters, safety factor (SF or SDV12 from above), critical mode (CMODE or SDV8), critical ply (CPLY or SDV10). Although these parameters are named similarly to the nonlinear parameters, please note the differences stated below.

For the linear case (LAMPAT), safety factor is defined as the ultimate load that a laminate could sustain, based on the allowable amount of progressive failure, divided by the applied loading on the laminate. The safety factor is therefore a scalar multiple of the applied loading. The process of determining this value starts by calculating the ratio of the strain allowable to ply strain for each ply and each direction (see equations 2–10). The lowest ratio is called the safety factor for that iteration, and the ply and direction in which the lowest ratio occurs is called the critical ply and critical mode, respectively. The ply and direction are next flagged for failure and the stiffness in the critical direction of that ply is discounted. New linear “smeared” properties for the laminate are then calculated and the process repeats. A new safety factor, critical mode, and critical ply are calculated and recorded for the next iteration, the plies are discounted, and “smeared” properties are updated. Iterations continue until the structural integrity of the laminate has been compromised. At the end of the analysis, the highest safety factor of all the iterations is determined to be last ply failure and the mode and ply of that highest safety factor are reported as critical mode and ply for the element.

The process for determining the safety factor in nonlinear, progressive failure analysis (LAMPATNL) is not as straightforward. First, stress and strains in nonlinear analysis cannot be linearly scaled to larger values because this analysis uses the incremental approach for determining stress from strain. Secondly, the progressive nature of the analysis requires that plies must exceed their strain allowable to be flagged for failure. In the linear analysis, the lowest ratio of the strain failure allowable to the ply strain is designated for failure and the ply is discounted. In nonlinear, progressive failure analysis, the safety factor is defined as the lowest ratio of failure allowable to ply strain at the current level and the critical ply and mode are the ply and direction of this lowest ratio. This determines which ply and direction are closest to failure, not which ply and direction would cause last ply failure. Because of these differences, it is not appropriate to directly compare values of the variables calculated in LAMPATNL to those calculated in the LAMPAT.

2. Material Database Format

All subroutines of LAMPAT and LAMPATNL (UVARM, VUSDFLD, UMAT, VUMAT) require and read a material database file that contains the region and ply material data used for simulating composite structures. Whether linear or nonlinear, static or dynamic, all the subroutines

read data from one database format. The details of how to create and properly format the database are given below and an example database is provided in the appendix of this manual.

The database file is broken into three separate areas: the header, region data, and material/ply properties (figure 2). The order and composition of these areas are important in ensuring the correct functioning of the subroutines. Each of these areas are discussed in detail.

Header	2				!	NUMBER OF REGIONS IN DATABASE		
	2				!	NUMBER OF MATERIALS IN DATABASE		
Region Data	1.0,	3.0,	10.0,	1.0	!	REG_ID,	FAIL_CRT,	ITERS, ---
	0.0,	0.0,	0.0,	0.0	!	BETA,	PHI,	SI, ---
	4.0,	2.0,	0.0,	0.0	!	NPLY,	---	---
	1.0,	0.05,	0.0,	-0.0001	!	MAT_ID,	THICKNESS,	ANGLE, ---
	1.0,	0.05,	90.0,	-0.0001	!	MAT_ID,	THICKNESS,	ANGLE, ---
	1.0,	0.05,	90.0,	-0.0001	!	MAT_ID,	THICKNESS,	ANGLE, ---
	1.0,	0.05,	0.0,	-0.0001	!	MAT_ID,	THICKNESS,	ANGLE, ---
	2.0,	3.0,	10.0,	1.0	!	REG_ID,	FAIL_CRT,	ITERS, ---
	0.0,	0.0,	0.0,	0.0	!	BETA,	PHI,	SI, ---
	2.0,	2.0,	0.0,	0.0	!	NPLY,	---	---
Material/Ply Properties	2.0,	0.05,	0.0,	-0.0001	!	MAT_ID,	THICKNESS,	ANGLE, ---
	2.0,	0.05,	0.0,	-0.0001	!	MAT_ID,	THICKNESS,	ANGLE, ---
	1.0,	2.0,	6.800E-02	!	MAT_ID,	MAT_TYPE,	MAT_DENS	
	6.397E+06,	1.010E+06,	1.010E+06	!	E1,	E2,	E3	
	1.000E+09,	9.929E+03,	9.929E+03	!	SIG_0-1,	SIG_0-2,	SIG_0-3	
	1.000E+01,	3.600E+00,	3.600E+00	!	N-1,	N-2,	N-3	
	3.840E-01,	2.540E-01,	2.540E-01	!	NU23,	NU13,	NU12	
	3.650E+05,	4.190E+05,	4.190E+05	!	G23,	G13,	G12	
	1.000E+09,	4.464E+03,	4.464E+03	!	SIG_0-23,	SIG_0-13,	SIG_0-12	
	1.000E+01,	2.370E+00,	2.370E+00	!	N-23,	N-13,	N-12	
	5.393E-06,	4.091E-05,	4.091E-05	!	A1,	A2,	A3	
	0.0,	0.0,	0.0	!	YLDSTRS,	---	---	... Von Mises (FAIL_CRT=1.0)
	0.0,	0.0,	0.0	!	---	---	---	
	0.0,	0.0,	0.0	!	---	---	---	
	0.0,	0.0,	0.0	!	---	---	---	
	0.0,	0.0,	0.0	!	---	---	---	
	1.750E+05,	3.000E+03,	3.000E+03	!	X1T,	X2T,	X3T	... Max Stress (FAIL_CRT=2.0)
	5.200E+04,	2.000E+04,	2.000E+04	!	X1C,	X2C,	X3C	
	5.000E+03,	1.100E+04,	1.100E+04	!	X23,	X13,	X12	
	0.0,	0.0,	0.0	!	---	---	---	
	0.0,	0.0,	0.0	!	---	---	---	
	2.736E-02,	2.970E-03,	2.970E-03	!	Y1T,	Y2T,	Y3T	... Max Strain (FAIL_CRT=3.0)
	8.129E-03,	1.980E-02,	1.980E-02	!	Y1C,	Y2C,	Y3C	
	1.370E-02,	2.625E-02,	2.625E-02	!	Y23,	Y13,	Y12	
	0.0,	0.0,	0.0	!	---	---	---	
	0.0,	0.0,	0.0	!	---	---	---	
	420.000E+03,	8.700E+03,	1.0	!	XLT,	XTT,	HPD	... Hydrostatic (FAIL_CRT=4.0)
	175.000E+03,	100.00E+03,	20.300E+03	!	XLG,	XTG,	S	
	2.0,	2.0,	0.0	!	ML1,	ML2,	LTP	
	1.0,	1.0,	0.0	!	MT1,	MT2,	TTP	
	1.0,	1.0,	0.0	!	MS1,	MS2,	STP	
	420.000E+03,	175.00E+03,	0.0	!	X1T,	X1C,	---	... Tsai-wu (FAIL_CRT=5.0)
	8.700E+03,	30.000E+03,	0.0	!	X2T,	X2C,	---	
	20.000E+03,	1.1915E-09,	-1.916E-09	!	S12,	F12,	F23	
	0.0,	0.0,	0.0	!	---	---	---	
	0.0,	0.0,	0.0	!	---	---	---	
	2.230E+07,	1.220E+06,	1.220E+06	!	E1,	E2,	E3	... Christensen (FAIL_CRT=6.0)
	0.450E+00,	0.330E+00,	0.330E+00	!	NU23,	NU13,	NU12	
	0.700E+06,	0.700E+06,	0.700E+06	!	G23,	G13,	G12	
	1.507E-02,	0.1027913,	0.0	!	K,	ALPHA,	---	
	1.88E-02,	7.8E-03,	0.0	!	Y1T,	Y1C,	---	
	452.55,	101910,	0.0	!	A1,	A11,	FE3	... Feng (FAIL_CRT=7.0)
	1189.1,	1189.1,	0.0	!	A2,	A4,	FE6	
	-37.42,	4819.74,	0.0	!	A5,	A55,	FE9	
	0.0,	0.0,	0.0	!	FE10,	FE11,	FE12	
	0.0,	0.0,	0.0	!	FE13,	FE14,	FE15	
	420.000E+03,	8.700E+03,	8.700E+03	!	X1T,	X2T,	X3T	... Hashin (FAIL_CRT=8.0)
	175.000E+03,	30.000E+03,	30.000E+03	!	X1C,	X2C,	X3C	
	20.000E+03,	20.000E+03,	20.000E+03	!	X23,	X13,	X12	
	0.0,	0.0,	0.0	!	HA10,	HA11,	HA12	
	0.0,	0.0,	0.0	!	HA13,	HA14,	HA15	

Figure 2. Example of material database format.

2.1 Header

The header consists of two lines of one number on each line, with the first number being the number of regions listed in this database file and the second number being the number of materials contained in the database file. If these numbers do not correspond correctly to what is in the database file, there will be errors in the material properties and/or layups when the code runs and will result in the simulation not running properly.

2.2 Region Data

The region data is where the layups of the composite that are going to be simulated are specified. Each set of region data should contain $3+N$ lines of four numbers separated by commas, where N is the number of plies that are going to be used in the layup. Generally, the fourth column in this region is not used, although numbers should be in this column (this column was used for thermal effects, something not yet integrated into the current code). The first line should contain the region number, failure criteria that is to be used for this region, and the number of iterations that will be evaluated (this is only used for LAMPAT analysis and is typically set to 10). The second line contains the beta, phi, and si that can be used to rotate the region to the proper orientation when compared to the coordinate system in the finite element model. The third line contains the number of plies for this layup and two values that are currently not used. It is important that the number of plies indicated on this line match the number in this region, as not matching these will probably lead to errors in the simulation. The next N number of lines should contain the number of the material used for the ply, the thickness of the ply, and the orientation angle of the ply.

For example, the first region shown in figure 2 contains a $(0^\circ/90^\circ)$ symmetric layup of the Material1 over four plies, which is evaluated using failure criteria 3. This region is not rotated in comparison to the finite element (FE) coordinate system ($\text{beta}=\text{phi}=\text{si}=0$). For LAMPAT analysis, 10 iterations of failure are evaluated (unless, of course, it is stopped for some reason before the 10th iteration).

2.3 Material/Ply Properties

The final area of the material database file contains the material or ply properties. This area consists of 49 lines of data, with 3 numbers separated by commas per line. The first 9 lines contain the properties of the material, while the last 40 lines contain the failure allowables for the 8 different failure criteria (8 blocks of 5 lines each).

The first line contains the number of the material, the material type (not used), and the density of the material (used only in the LAMPAT Preprocessor). The second line contains the Young's modulus (E) in the 1, 2, and 3 directions. The third line contains the sigma-0 (stress asymptote) parameters for the 1, 2, and 3 directions used in the Ramberg-Osgood equation to model nonlinearity. The fourth line is the N (shape factor) parameters for the 1, 2, and 3 directions used in the Ramberg-Osgood equation to model nonlinearity. When running the LAMPAT (linear) analysis lines 3 and 4 (along with the shape factors and sigma-0 for the shear modulus) are ignored. The fifth line contains the Poisson's ratios ν_{23} , ν_{13} , and ν_{12} , in that order. The sixth line contains the shear modulus (G) in the 23, 13, and 12 directions. The seventh line contains the sigma-0 (stress asymptote) parameters for the 23, 13, and 12 directions used in the Ramberg-Osgood equation to model nonlinearity. The eighth line is the N (shape factor) parameters for the 23, 13, and 12 directions used in the Ramberg-Osgood equation to model nonlinearity. The ninth line contains the thermal expansion coefficients alpha 1, 2, and 3 (these material properties are not currently used in the analysis).

Considerations for the failure allowables of this material should be examined on a criterion-by-criterion basis. The two most commonly used failure criteria are the maximum stress and maximum strain failure criteria and they are discussed here. The maximum stress failure allowable starts at line 15 (lines 10 to 14 are for the Von Mises failure allowables). On this line, the tensile failure stresses for the 1, 2, and 3 directions should be listed. Line 16 contains the compressive failure stresses for the 1, 2, and 3 directions. Line 17 should contain the shear failure stresses for the 23, 13, and 12 directions. Lines 18 and 19 are not used but should be left as zeros. The maximum strain failure allowable starts at line 20, which should contain the tensile failure strains for the 1, 2, and 3 directions. Line 21 contains the compressive failure strains for the 1, 2, and 3 directions. Line 22 should contain the shear failure strains for the 23, 13, and 12 directions. Lines 23 and 24 are not used but should be left as zeros. Any other failure allowables for the other failure criteria should be investigated by the user to ensure proper functioning of the failure theory.

3. LAMPAT (UVARM)

The procedure and special considerations for setting up and running a linear, uncoupled, progressive failure analysis on a static simulation using LAMPAT are discussed in this section. For each subroutine, specifics on the material definition, additions to the field output requests, and element compatibility are discussed. Also, modifications to the subroutine increase the variable definitions to accommodate larger mesh sizes (if required) are detailed.

As with all subroutines discussed in this manual, the correct path of the database file used for the model must be specified for the correct functioning of LAMPAT. To do this, the user must modify the following line in the subroutine: `OPEN(UNIT=15,FILE='(path)',STATUS='OLD')`, where (path) is replaced with the path and name of the database file. For example, the database file used for the current simulation is called `dbase.txt` and is located in the root directory of the C drive so the line would become `OPEN(UNIT=15,FILE='C:\dbase.txt',STATUS='OLD')`.

3.1 Material Definition

When creating materials for use with LAMPAT, one of the most important steps is naming the material correctly. The naming convention required for these materials is `REGION#`, where the # is replaced by the number of the region in the database to which this material refers. For example, if the model has three separate composite layups in it, the database should have three regions and the materials created in the model should be named `REGION1`, `REGION2`, and `REGION3` (region is not required to be caps). This naming convention is required because the subroutine will identify the region number for analysis based upon the material name.

There are two required options that need to be selected when defining the material for this case. These options are Elastic (typically engineering constants, though it is not required) and User

Output Variables. The elastic engineering constants is defined by selecting Mechanical->Elastic->Type: Engineering constants in the Edit Material dialog box of Abaqus/CAE. To determine the engineering constants of each region, the LAMPAT Preprocessor must be run with the database file that will be used in the simulation. A material “card” will be output by the preprocessor and these values must be input into the engineering constants dialog box in the order presented. The User Output Variable option is selected through the General menu in the Edit Material dialog box. The number of user-defined variables at each material point must be set to 3.

3.2 Step and Field Output Requests

In creating the step for this analysis (typically a Static, General step), it is recommended that the initial increment size be decreased from 1 to 0.1 (under the Incrementation tab). This will help the model solve without having to cut back the step size and also create intermediate points in the analysis for the plots of safety factor, critical mode, and critical ply. For large problems, the automatic incrementation would probably cut back the step size anyway.

To ensure that the output variables of LAMPAT are recorded, the field output request must be modified to include the UVARM. To add this, select the UVARM checkbox under the State/Field/User/Time section of the Edit Field Output Request dialog box. The output parameters produced by this subroutine are: UVARM1 = safety factor, UVARM2 = critical mode, UVARM3 = critical ply. See section 1.2, description of output parameters for LAMPAT, for an explanation of these values.

3.3 Element Compatibility

This subroutine has been tested and verified to be working with both full and reduced integration elements of plane stress (CPS4 and CPS4R) and three-dimensional (3D) stress (C3D8 and C3D8R). Because of the way the subroutine reads in stresses, the UVARM should be compatible with any element that returns the stress in to the GETVRM utility routine in the form S11, S22, S33, S12, S13, S23.

3.4 Special Considerations

The linear, uncoupled progressive failure simulated in LAMPAT does not require the tracking of failure history of the element and integration point. Because of this, there is no need to modify the subroutine to accommodate a large number of elements.

4. LAMPATNL (UMAT)

The procedure and special considerations for setting up and running a nonlinear, coupled, progressive failure analysis on a static simulation using LAMPATNL are discussed in this section. As with all subroutines discussed in this manual, the correct path of the database file used for the

model must be specified for the correct functioning of LAMPATNL. To do this, the user must modify the following line in the subroutine: `OPEN(UNIT=16,FILE=(path)',STATUS='OLD')`, where (path) is replaced with the path and name of the database file. For example, the database file used for the current simulation is called `dbase.txt` and is located in the root directory of the C drive so the line would become `OPEN(UNIT=16,FILE='C:\dbase.txt',STATUS='OLD')`.

4.1 Material Definition

There are two required options that need to be selected when defining the material for this case. These options are User Material and Depvar, both selected through the General menu in the Edit Material dialog box. For the User Material, the first (and only) mechanical constant is required to be the number of the region in the database that corresponds to the composite desired to be modeled with this material (1 for region 1, 2 for region 2, etc.). For the Depvar option, the number of solution-dependent state variables must be set to 16.

4.2 Step and Field Output Requests

In creating the step for this analysis (typically, a Static, General step), it is important to switch the Nlgeom from Off (default) to On. It is also strongly recommended to increase the maximum number of increments to 500, decrease the initial increment size from 1 to 0.01, and decrease the maximum increment size also from 1 to 0.01 (under the Incrementation tab). Because of the piecewise linear procedure used to model material nonlinearity, it is important to keep the increment size small to ensure the desired nonlinearity is modeled.

To ensure that the output variables of LAMPATNL are recorded, the field output request must be modified to include the solution dependent variables (SDV). To add this, select the SDV checkbox under the State/Field/User/Time section of the Edit Field Output Request dialog box. There are 16 output parameters produced by this subroutine, SDV1 through SDV16. See section 1.1, description of output parameters for LAMPATNL, for an explanation of these values.

4.3 Element Compatibility

This subroutine has been tested and verified to be working with both full and reduced integration elements of plane stress (CPS4 and CPS4R) and 3D stress (C3D8 and C3D8R). When using reduced integration elements with the UMAT, it is required to specify an hourglass stiffness for the elements. For solid elements, it is recommended that the hourglass stiffness be equal to $0.005 \cdot G$, where G is the shear modulus of the material (for more detailed information on selecting an hourglass stiffness, see section 21.1.4-Section Controls of the ABAQUS Analysis User's Manual for Version 6.7).

4.4 Special Considerations

The nonlinear, coupled progressive failure simulated in LAMPATNL requires the tracking of failure history of the element and integration point. There are several arrays of variables in the subroutine that are sized based upon the maximum number of elements in the model. Because

the UMAT does not have access to the total number of elements in a given model, a sufficiently large number is chosen to predefine these arrays (currently, this number is set to 9,000). If there are more than 9000 elements in your model, you must increase this parameter in order for the UMAT to run as expected. To do this, simply modify the line `PARAMETER (INEL=9000)` by increasing the number. Setting this value to reflect the relative size of your analysis ensures that enough variables are predefined, but setting it too large (for example, 30,000 used in an 1000 element model) may decrease the performance of the code and increase the load on the memory of the computer.

5. LAMPAT (VUSDFLD)

The procedure and special considerations for setting up and running a linear, uncoupled, progressive failure analysis on a dynamic simulation using LAMPAT are discussed in this section. As with all subroutines discussed in this manual, the correct path of the database file used for the model must be specified for the correct functioning of LAMPAT. To do this, the user must modify the following line in the subroutine: `OPEN(UNIT=17,FILE='(path)',STATUS='OLD')`, where (path) is replaced with the path and name of the database file. For example, the database file used for the current simulation is called `dbase.txt` and is located in the root directory of the C drive so the line would become `OPEN(UNIT=17,FILE='C:\dbase.txt',STATUS='OLD')`.

5.1 Material Definition

When creating materials for use with LAMPAT, one of the most important steps is naming the material correctly. The naming convention required for these materials is `REGION#`, where the # is replaced by the number of the region in the database to which this material refers. For example, if the model has three separate composite layups in it, the database should have three regions and the materials created in the model should be named `REGION1`, `REGION2`, and `REGION3` (region is not required to be caps). This naming convention is required because the subroutine will identify the region number for analysis based upon the material name.

There are four required options that need to be selected when defining the material for this case. These options are Elastic (typically engineering constants, though it is not required), Density, Depvar, and User Defined Field. The elastic engineering constants is defined by selecting Mechanical->Elastic->Type: Engineering constants in Abaqus/CAE. To determine the engineering constants of each region, the LAMPAT Preprocessor must be run with the database file that will be used in the simulation. A material “card” will be output by the preprocessor and these values must be input into the engineering constants dialog box in the order presented. The Depvar option is selected through the General menu in the edit material dialog box. The number of user-defined variables at each material point must be set to 3. The Density option is also selected through the General menu in the edit material dialog box and should be set to the mass

density of the composite. Finally, the User Defined Field option must be selected through the General menu in the edit material dialog box and there is no input for this.

5.2 Step and Field Output Requests

The typical step type for this analysis is a Dynamic, Explicit step. The default settings for this step are sufficient to the needs of running LAMPAT. To ensure that the output variables of LAMPAT are recorded, the field output request must be modified to include the SDV. To add this, select the SDV checkbox under the State/Field/User/Time section of the Edit Field Output Request dialog box. The output parameters produced by this subroutine are SDV1 = safety factor, SDV2 = critical mode, SDV3 = critical ply. See section 1.2, description of output parameters for LAMPAT, for an explanation of these values.

5.3 Element Compatibility

This subroutine has been tested and verified to be working with both full and reduced integration elements of 3D stress (C3D8 and C3D8R). Because of the way the subroutine reads in stresses, the VUSDFLD should be compatible with any element that returns the stress in to the VGETVRM utility routine in the form S11, S22, S33, S12, S13, and S23.

5.4 Special Considerations

The linear, uncoupled progressive failure simulated in LAMPAT does not require the tracking of failure history of the element and integration point. Because of this, there is no need to modify the subroutine to accommodate a large number of elements.

6. LAMPATNL (VUMAT)

The procedure and special considerations for setting up and running a nonlinear, coupled, progressive failure analysis on a dynamic simulation using LAMPATNL are discussed in this section. As with all subroutines discussed in this manual, the correct path of the database file used for the model must be specified for the correct functioning of LAMPATNL. To do this, the user must modify the following line in the subroutine: `OPEN(UNIT=16,FILE='(path)',STATUS='OLD')`, where (path) is replaced with the path and name of the database file. For example, the database file used for the current simulation is called `dbase.txt` and is located in the root directory of the C drive so the line would become `OPEN(UNIT=16,FILE='C:\dbase.txt',STATUS='OLD')`.

6.1 Material Definition

There are three required options that need to be selected when defining the material for this case. These options are User Material, Density, and Depvar, all selected through the General menu in the Edit Material dialog box. For the User Material, the first (and only) mechanical constant is required to be the number of the region in the database that corresponds to the composite desired

to be modeled with this material. For the Depvar option, the number of solution-dependent state variables must be set to 16. The Density option should be set to the mass density of the composite.

6.2 Step and Field Output Requests

The typical step type for this analysis is a Dynamic, Explicit step. The default settings for this step are sufficient to the needs of running LAMPATNL. Because of the small increment size of explicit analyses, the default setup is sufficient to simulate the piecewise linear procedure used to model material nonlinearity.

To ensure that the output variables of LAMPATNL are recorded, the field output request must be modified to include the SDV. To add this, select the SDV checkbox under the State/Field/User/Time section of the Edit Field Output Request dialog box. There are 16 output parameters produced by this subroutine, SDV1 through SDV16. See section 1.1, description of output parameters for LAMPATNL, for an explanation of these values.

6.3 Element Compatibility

This subroutine has been tested and verified to be working with both fully integrated and reduced integration elements of plane stress (CPS4 and CPS4R) and 3D stress (C3D8 and C3D8R).

When using reduced integration elements with the VUMAT, it is required to specify an hourglass stiffness for the elements. For solid elements, it is recommended that the hourglass stiffness be equal to $0.005 \cdot G$, where G is the shear modulus of the material (for more detailed information on selecting an hourglass stiffness (see section 21.1.4-Section Controls of the ABAQUS Analysis User's Manual for Version 6.7)).

6.4 Special Considerations

The nonlinear, coupled progressive failure simulated in LAMPATNL requires the tracking of failure history of the element and integration point. There are several arrays of variables in the subroutine that are sized based upon the maximum number of elements in the model. Because the VUMAT does not have access to element number and integration point number, specific considerations needed to be taken. Abaqus breaks up each material into blocks of 136 elements to be evaluated in each call to the subroutine. Abaqus does not provide unique identifiers for specific elements and integration points, which presents a problem when tracking failure history. The most recent solution in an attempt to address this problem continuously updates a unique number that is reset with each new increment irrespective of element or integration point.

For the user, this section is important when accounting for a large amount of elements. Currently, the VUMAT should handle 1250 C3D8 or 2500 CPS4 fully integrated elements and 10,000 reduced integration elements. Increasing this number depends upon whether you are using reduced integration or not. For reduced integration, simply change the line PARAMETER (INEL=10000) to a number greater than the amount of elements in your model. For full

integration, take the number of elements in your model and multiply by 8 for the 3D case (4 for the two-dimensional [2D] case). Next, set the line `PARAMETER (INEL=10000)` to something greater than this resulting number. For example, the model being run has 13,000 C3D8 elements. Multiplying 13,000 by 8 gives 104,000 (choose 105,000, as the number greater). So the line should be `PARAMETER (INPT=105000)`.

Appendix. Example Material Database

This appendix appears in its original form, without editorial change.

2				!	NUMBER OF REGIONS IN DATABASE		
2				!	NUMBER OF MATERIALS IN DATABASE		
1.0,	3.0,	10.0,	1.0	!	REG_ID,	FAIL_CRT,	ITERS, ---
0.0,	0.0,	0.0,	0.0	!	BETA,	PHI,	SI, ---
4.0,	2.0,	0.0,	0.0	!	NPLY,	---	---
1.0,	0.05,	0.0,	-0.0001	!	MAT_ID,	THICKNESS,	ANGLE, ---
1.0,	0.05,	90.0,	-0.0001	!	MAT_ID,	THICKNESS,	ANGLE, ---
1.0,	0.05,	90.0,	-0.0001	!	MAT_ID,	THICKNESS,	ANGLE, ---
1.0,	0.05,	0.0,	-0.0001	!	MAT_ID,	THICKNESS,	ANGLE, ---
2.0,	3.0,	10.0,	1.0	!	REG_ID,	FAIL_CRT,	ITERS, ---
0.0,	0.0,	0.0,	0.0	!	BETA,	PHI,	SI, ---
2.0,	2.0,	0.0,	0.0	!	NPLY,	---	---
2.0,	0.05,	0.0,	-0.0001	!	MAT_ID,	THICKNESS,	ANGLE, ---
2.0,	0.05,	0.0,	-0.0001	!	MAT_ID,	THICKNESS,	ANGLE, ---
1.0,	2.0,	6.800E-02	!	MAT_ID,	MAT_TYPE,	MAT_DENS	
6.397E+06,	1.010E+06,	1.010E+06	!	E1,	E2,	E3	
1.000E+09,	9.929E+03,	9.929E+03	!	SIG_0-1,	SIG_0-2,	SIG_0-3	
1.000E+01,	3.600E+00,	3.600E+00	!	N-1,	N-2,	N-3	
3.840E-01,	2.540E-01,	2.540E-01	!	NU23,	NU13,	NU12	
3.650E+05,	4.190E+05,	4.190E+05	!	G23,	G13,	G12	
1.000E+09,	4.464E+03,	4.464E+03	!	SIG_0-23,	SIG_0-13,	SIG_0-12	
1.000E+01,	2.370E+00,	2.370E+00	!	N-23,	N-13,	N-12	
5.393E-06,	4.091E-05,	4.091E-05	!	A1,	A2,	A3	
0.0,	0.0,	0.0	!	YLDSTRS,	---	---	... Von Mises
0.0,	0.0,	0.0	!	---	---	---	
0.0,	0.0,	0.0	!	---	---	---	
0.0,	0.0,	0.0	!	---	---	---	
0.0,	0.0,	0.0	!	---	---	---	
1.750E+05,	3.000E+03,	3.000E+03	!	X1T,	X2T,	X3T	... Max Stress
5.200E+04,	2.000E+04,	2.000E+04	!	X1C,	X2C,	X3C	
5.000E+03,	1.100E+04,	1.100E+04	!	X23,	X13,	X12	
0.0,	0.0,	0.0	!	---	---	---	
0.0,	0.0,	0.0	!	---	---	---	
2.736E-02,	2.970E-03,	2.970E-03	!	Y1T,	Y2T,	Y3T	... Max Strain
8.129E-03,	1.980E-02,	1.980E-02	!	Y1C,	Y2C,	Y3C	
1.370E-02,	2.625E-02,	2.625E-02	!	Y23,	Y13,	Y12	
0.0,	0.0,	0.0	!	---	---	---	
0.0,	0.0,	0.0	!	---	---	---	
420.000E+03,	8.700E+03,	1.0	!	XLT,	XTT,	HPD	... Hydrostatic
175.000E+03,	100.00E+03,	20.300E+03	!	XLC,	XTC,	S	
2.0,	2.0,	0.0	!	ML1,	ML2,	LTP	
1.0,	1.0,	0.0	!	MT1,	MT2,	TTP	
1.0,	1.0,	0.0	!	MS1,	MS2,	STP	
420.000E+03,	175.00E+03,	0.0	!	X1T,	X1C,	---	... Tsai-Wu
8.700E+03,	30.000E+03,	0.0	!	X2T,	X2C,	---	
20.000E+03,	1.1915e-09,	-1.916e-09	!	S12,	F12,	F23	
0.0,	0.0,	0.0	!	---	---	---	
0.0,	0.0,	0.0	!	---	---	---	
2.230E+07,	1.220E+06,	1.220E+06	!	E1,	E2,	E3	... Christensen
0.450E+00,	0.330E+00,	0.330E+00	!	NU23,	NU13,	NU12	
0.700E+06,	0.700E+06,	0.700E+06	!	G23,	G13,	G12	
1.507E-02,	0.1027913,	0.0	!	K,	ALPHA,	---	
1.88E-02,	7.8E-03,	0.0	!	Y1T,	Y1C,	---	
452.55,	101910,	0.0	!	A1,	A11,	FE3	... Feng
1189.1,	1189.1,	0.0	!	A2,	A4,	FE6	
-37.42,	4819.74,	0.0	!	A5,	A55,	FE9	
0.0,	0.0,	0.0	!	FE10,	FE11,	FE12	
0.0,	0.0,	0.0	!	FE13,	FE14,	FE15	
420.000E+03,	8.700E+03,	8.700E+03	!	X1T,	X2T,	X3T	... Hashin
175.000E+03,	30.000E+03,	30.000E+03	!	X1C,	X2C,	X3C	
20.000E+03,	20.000E+03,	20.000E+03	!	X23,	X13,	X12	
0.0,	0.0,	0.0	!	HA10,	HA11,	HA12	
0.0,	0.0,	0.0	!	HA13,	HA14,	HA15	
2.0,	2.0,	5.400E-02	!	MAT_ID,	MAT_TYPE,	MAT_DENS	
1.932E+07,	8.470E+05,	8.470E+05	!	E1,	E2,	E3	
1.000E+09,	1.815E+04,	1.815E+04	!	SIG_0-1,	SIG_0-2,	SIG_0-3	

1.000E+01,	3.600E+00,	3.600E+00	!	N-1,	N-2,	N-3	
3.960E-01,	2.390E-01,	2.390E-01	!	NU23,	NU13,	NU12	
3.030E+05,	2.390E+05,	2.390E+05	!	G23,	G13,	G12	
1.000E+09,	3.805E+03,	3.805E+03	!	SIG_O-23,	SIG_O-13,	SIG_O-12	
1.000E+01,	2.020E+00,	2.020E+00	!	N-23,	N-13,	N-12	
3.064E-07,	4.316E-05,	4.316E-05	!	A1,	A2,	A3	
0.0,	0.0,	0.0	!	YLDSTRS,	---	---	... Von Mises
0.0,	0.0,	0.0	!	---	---	---	
0.0,	0.0,	0.0	!	---	---	---	
0.0,	0.0,	0.0	!	---	---	---	
0.0,	0.0,	0.0	!	---	---	---	
4.200E+05,	8.700E+03,	8.700E+03	!	X1T,	X2T,	X3T	... Max Stress
1.750E+05,	3.000E+04,	3.000E+04	!	X1C,	X2C,	X3C	
2.030E+04,	2.030E+04,	2.030E+04	!	X23,	X13,	X12	
0.0,	0.0,	0.0	!	---	---	---	
0.0,	0.0,	0.0	!	---	---	---	
2.174E-02,	1.027E-02,	1.027E-02	!	Y1T,	Y2T,	Y3T	... Max Strain
9.058E-03,	3.542E-02,	3.542E-02	!	Y1C,	Y2C,	Y3C	
6.700E-02,	8.494E-02,	8.494E-02	!	Y23,	Y13,	Y12	
0.0,	0.0,	0.0	!	---	---	---	
0.0,	0.0,	0.0	!	---	---	---	
420.000E+03,	8.700E+03,	1.0	!	XLT,	XTT,	HPD	... Hydrostatic
175.000E+03,	100.00E+03,	20.300E+03	!	XLC,	XTC,	S	
2.0,	2.0,	0.0	!	ML1,	ML2,	LTP	
1.0,	1.0,	0.0	!	MT1,	MT2,	TTP	
1.0,	1.0,	0.0	!	MS1,	MS2,	STP	
420.000E+03,	175.00E+03,	0.0	!	X1T,	X1C,	---	... Tsai-Wu
8.700E+03,	30.000E+03,	0.0	!	X2T,	X2C,	---	
20.000E+03,	1.1915E-09,	-1.916E-09	!	S12,	F12,	F23	
0.0,	0.0,	0.0	!	---	---	---	
0.0,	0.0,	0.0	!	---	---	---	
2.230E+07,	1.220E+06,	1.220E+06	!	E1,	E2,	E3	... Christensen
0.450E+00,	0.330E+00,	0.330E+00	!	NU23,	NU13,	NU12	
0.700E+06,	0.700E+06,	0.700E+06	!	G23,	G13,	G12	
1.507E-02,	0.1027913,	0.0	!	K,	ALPHA,	---	
1.88E-02,	7.8E-03,	0.0	!	Y1T,	Y1C,	---	
452.55,	101910,	0.0	!	A1,	A11,	FE3	... Feng
1189.1,	1189.1,	0.0	!	A2,	A4,	FE6	
-37.42,	4819.74,	0.0	!	A5,	A55,	FE9	
0.0,	0.0,	0.0	!	FE10,	FE11,	FE12	
0.0,	0.0,	0.0	!	FE13,	FE14,	FE15	
420.000E+03,	8.700E+03,	8.700E+03	!	X1T,	X2T,	X3T	... Hashin
175.000E+03,	30.000E+03,	30.000E+03	!	X1C,	X2C,	X3C	
20.000E+03,	20.000E+03,	20.000E+03	!	X23,	X13,	X12	
0.0,	0.0,	0.0	!	HA10,	HA11,	HA12	
0.0,	0.0,	0.0	!	HA13,	HA14,	HA15	

NO. OF
COPIES ORGANIZATION

1 ELEC	ADMNSTR DEFNS TECHL INFO CTR ATTN DTIC OCP 8725 JOHN J KINGMAN RD STE 0944 FT BELVOIR VA 22060-6218
1	US ARMY RSRCH DEV AND ENGRG CMND ARMAMENT RSRCH DEV & ENGRG CTR ARMAMENT ENGRG & TECHN LGY CTR ATTN AMSRD AAR AEF T J MATTS BLDG 305 ABERDEEN PROVING GROUND MD 21005-5001
1	US ARMY INFO SYS ENGRG CMND ATTN AMSEL IE TD A RIVERA FT HUACHUCA AZ 85613-5300
1	US GOVERNMENT PRINT OFF DEPOSITORY RECEIVING SECTION ATTN MAIL STOP IDAD J TATE 732 NORTH CAPITOL ST NW WASHINGTON DC 20402
1	US ARMY RSRCH LAB ATTN RDRL WML G J BENDER ABERDEEN PROVING GROUND MD 21005
2	US ARMY RSRCH LAB ATTN RDRL WML G W DRYSDALE ATTN RDRL WMM A E WETZEL BLDG 4600 ABERDEEN PROVING GROUND MD 21005
3	US ARMY RSRCH LAB ATTN RDRL WMM A J SANDS ATTN RDRL WMM A J TZENG ATTN RDRL WMM A R EMERSON ABERDEEN PROVING GROUND MD 21005
1	US ARMY RSRCH LAB ATTN RDRL WMM B B LOVE BLDG 393 ABERDEEN PROVING GROUND MD 21005

NO. OF
COPIES ORGANIZATION

3	US ARMY RSRCH LAB ATTN RDRL WMM B G GAZONAS ATTN RDRL WMM B J STANISZEWSKI ATTN RDRL WMM B M VANLANDINGHAM BLDG 4600 ABERDEEN PROVING GROUND MD 21005
1	US ARMY RSRCH LAB ATTN RDRL WMM B S BOYD ABERDEEN PROVING GROUND MD 21005
2	US ARMY RSRCH LAB ATTN RDRL WMM B T BOGETTI ATTN RDRL WMM J ZABINSKI BLDG 4600 ABERDEEN PROVING GROUND MD 21005
2	US ARMY RSRCH LAB ATTN RDRL WMP B C HOPPEL ATTN RDRL WMP B D POWELL BLDG 390 ABERDEEN PROVING GROUND MD 21005
1	US ARMY RSRCH LAB ATTN RDRL WML H T G BROWN BLDG 4600 ABERDEEN PROVING GROUND MD 21005-5066
1	US ARMY RSRCH LAB ATTN RDRL WML G M A MINNICINO BLDG 4600 ABERDEEN PROVING GROUND MD 21005-5069
3	US ARMY RSRCH LAB ATTN IMAL HRA MAIL & RECORDS MGMT ATTN RDRL CIO LL TECHL LIB ATTN RDRL CIO LT TECHL PUB ADELPHI MD 20783-1197

TOTAL: 24 (1 ELEC, 23 HCS)