IN-34-CR
64278
P_23

Final Technical Report

PROGRESSIVE FAILURE ANALYSIS OF FIBROUS COMPOSITE MATERIALS AND STRUCTURES

by

Yehia A. Bahei–El–Din

Department of Civil and Environmental Engineering
Rensselaer Polytechnic Institute, Troy, New York 12180–3590
& Structural Engineering Department, Cairo University, Giza, Egypt

Submitted to

National Aeronautics and Space Administration
Langley Research Center
Hampton, VA 23665–5225

Contract Number NAG–1–1274

December 1990

TABLE OF CONTENTS

ABSTRACT

This report contains a brief description of the modifications implemented in the PAFAC finite element program for simulation of progressive failure in fibrous composite materials and structures. Details of the memory allocation, input data, and the new subroutines are given. Also, built–in failure criteria for homogeneous and fibrous composite materials are described.

1. INTRODUCTION

It is well known that initiation of local failure in fibrous composites may occur at an overall load which is only a fraction of the ultimate strength. For example, Johnson et. al. (1983) detected first fiber failure in center notched 0° $B/A\ell$ composite specimens at about 60% of the ultimate load. Furthermore, cracks may be caused in the matrix by thermal stresses generated during fabrication or after sustained cyclic plastic straining (Dvorak and Johnson 1980). Such localized damage states cause reduction of the composites overall stiffness, but will not significantly affect the overall strength of a given structure until large scale damage develop in the system. The failure process is therefore progressive and must be considered in any realistic model. In particular, finite element procedures which are capable of performing progressive failure analyses are very useful in predicting failure of structures with complex geometry and nonlinear constitutive behavior.

Several years ago, the finite element program PAC78 (Bahei–El–Din 1979, Bahei–El–Din et al. 1981) was developed to perform elastic–plastic analysis of fibrous composite structures. Constitutive equations of a unidirectionally reinforced composite material were derived from the vanishing fiber diameter model (Bahei–El–Din 1979, Dvorak and Bahei–El–Din 1982). The program was later modified under NASA Langley sponsorship to perform plastic and failure analysis of composites. This resulted in the PAFAC program (Bigelow and Bahei–El–Din 1983) which is currently in use at NASA Langley and other government research laboratories and centers. The primary objective at that time was to implement a failure criterion based on the fiber stresses in the finite element routine and detect the first fiber failure in a given structure under mechanical loads. The PAFAC program has been used at NASA Langley for stress analysis of various problems in metal matrix composites and laminates (Bigelow 1989, Johnson and Bigelow 1989, Johnson et al. 1983).

The objective of the current project is to develop and implement in the PAFAC program the logic required to perform progressive nonlinear/failure analysis for fibrous media. The changes in the old PAFAC program, and the failure criteria used in the analysis are described in this report. First, we give an overview of the PAFAC program. Next, the logic implemented in the program to simulate failure of elements is described. Finally, the memory allocation, input data, and the new user–defined subroutines added to the program are given.

2. OVERVIEW OF THE PAFAC FINITE ELEMENT PROGRAM

PAFAC is a general purpose three–dimensional finite element program. It can be used for linear elastic analysis or nonlinear elastic–plastic analysis. Two material types are acceptable to the program, homogeneous isotropic materials, and fiber–reinforced composite materials. The composite material model is a three–dimensional continuum model with unidirectional elastic fibers and an elastic–plastic matrix. The matrix could be nonhardening or exhibiting kinematic hardening behavior. The homogeneous material model is a three–dimensional continuum model exhibiting kinematic hardening behavior. For either material model, the Mises yield criterion and the hardening rule of Prager with Ziegler's modification are employed. A restart capability is built in the program which allows analysis of the problems with previous loading history. Loads handles by the PAFAC program are proportional mechanical loads with any loading, unloading, and reloading sequence.

The PAFAC program does not include any pre– or post–processors. The users, however, may prepare their own subroutines for automatic generation of nodal coordinates, element properties and connectivity, boundary conditions, and yield and hardening information for all material types.

The three–dimensional solid element (eight–node hexahedron) is the only type that the PAFAC program uses to span the material volume. By means of the deflection boundary conditions, any type of deformation may be simulated, for example, beams, plates, plane strain, etc. Governing equations for the finite element solution are generated in the context of the displacement method of analysis. Linear equilibrium equations are solved by Cholesky method. Nonlinearities are handled by a Newton–Raphson type iterative scheme. In its present form, the PAFAC program predicts initial failure loads in fibrous composites using a quadratic failure criterion for the fiber phase function of the axial stress and the longitudinal shear stress.

Description of the input data and the output of the PAFAC program can be found in the user's manual prepared by Bigelow and Bahei–El–Din [1983]. Solutions for a variety of laminated plate problems can be found in Bigelow (1989), Johnson and Bigelow (1989), and Johnson et al. (1983).

3. DESCRIPTION OF THE MODIFIED PAFAC PROGRAM

3.1 Progressive Failure Analysis

In the modified PAFAC program, element failure is examined upon completion of a loading step. The user can choose to examine failure using failure criteria defined by the program, or specify the failure criteria by supplying two subroutines, UFAIL to describe the failure criteria and check element failure, and DFAIL to provide the input data required by the user–defined failure criterion. The form of the failure criteria specified by the program depends on the type of material. For isotropic materials, the following failure envelope is specified in the stress space:

$$f = \left(\frac{\overline{\sigma}_n}{\sigma^u} \right)^2 - \left(\frac{\overline{\sigma}_s}{\tau^u} \right)^2 - 1.0 = 0 \ , \tag{1}$$

where

$$\overline{\sigma}_n = \sigma_{11}^2 + \sigma_{22}^2 + \sigma_{33}^2 - \sigma_{11}\sigma_{22} - \sigma_{22}\sigma_{33} - \sigma_{33}\sigma_{11} \ , \tag{2}$$

$$\overline{\sigma}_s = 3\ (\sigma_{12}^2 + \sigma_{13}^2 + \sigma_{23}^2) \ , \tag{3}$$

σ_{ij} is the average stress in an element, and σ^u, τ^u are the strength of the material under normal stress and shear stress, respectively. The failure surface given by eqs. (1)–(3) is reminiscent of the Mises yield surface but with different strength under normal stresses and shear stresses.

For fibrous composite materials, the user can choose among two failure criteria available in the PAFAC program, one is based on the overall stress, the other is based on the fiber stress. In either case, only the axial normal stress and the longitudinal shear stress components appear in the failure criteria. Specifically, the failure envelope is given by

$$f = \left(\frac{\sigma_{33}}{\sigma^u} \right)^2 + \left(\frac{\sigma_{13}}{\tau^u} \right)^2 + \left(\frac{\sigma_{23}}{\tau^u} \right)^2 - 1.0 = 0 \ , \tag{4}$$

or

$$f = \left(\frac{\overset{f}{\sigma}_{33}}{\sigma^{uf}} \right)^2 + \left(\frac{\overset{f}{\sigma}_{13}}{\tau^{uf}} \right)^2 + \left(\frac{\overset{f}{\sigma}_{23}}{\tau^{uf}} \right)^2 - 1.0 = 0 \ . \tag{4}$$

Here, σ_{ij} is the overall stress and $\overset{f}{\sigma}_{ij}$ is the fiber stress specified in a Cartesian coordinate system x_k, $k = 1,2,3$, such that the x_3–axis coincides with the fiber axial direction, and x_1x_2 coincides with the transverse plane. The symbols σ^u, σ^u, denote overall strength of the fibrous composite under axial normal stress and longitudinal shear stress, respectively. Similarly, σ^{uf}, τ^{uf} denote the fiber strength under axial normal stress and longitudinal shear stress, respectively.

When element failure is detected by the program, the forces equilibrating the stresses supported by the failed element are retrieved from the common array, and their norm, RNORM, is computed as the sum of the absolute value of the individual components. This force vector is re–applied to a new mesh in which the failed element is removed, and a new overall stiffness matrix is formed. The force vector, however, is applied in small increments, such that the norm of each increment does not exceed the norm, PNORMB, of the overall load increment applied to the structure prior to the element failure. Specifically, the program scales down the residual load vector computed at element failure by the ratio PNORMB/RNORM if it is less than unity, otherwise, the residual load vector is applied back to the modified mesh as one load increment. During redistribution of the residual load vector found for a failed element, the overall loads are held constant. If in this process more elements fail, the residual load vectors corresponding to the stresses in the failed elements are computed and the overall residual load vector is updated. After redistribution of the total stress supported by the failed elements, a new overall load increment is applied and the process for evaluation of the failure criterion in the element and redistribution of the stresses, if necessary, is repeated.

3.2 Modifications to Memory Allocation and Input Data

Minor modifications are made in the memory allocation and input data. Table 1 shows the common map of the general constants and arrays in the common block of the modified PAFAC program. The input data for a standard job are given in Tables 2–4. For a restart job, the input data are given in Tables 5,6.

3.3 New User–Defined Subroutines

Two user–defined subroutines are added to the new PAFAC program, UFAIL and DFAIL, which specify the failure criterion and read the failure parameters. Listing of the DFAIL subroutine is given in Fig. 1, and listing of the UFAIL subroutine is given in Fig. 2.

ACKNOWLEDGEMENT

The principal investigator is indebted to Dr. Catherine A. Bigelow, project monitor, and Dr. W. Steven Johnson for their encouragement and useful technical suggestions.

REFERENCES

Bahei–El–Din, Y.A., 1979 "Plasticity Analysis of Metal Matrix Composite Laminates," Ph.D. Thesis, Duke University.

Bahei–El–Din, Y.A., Dvorak, G.J. and Utku, S., 1981 "Finite Element Analysis of Elastic–Plastic Fibrous Composite Structures," *Computers and Structures*, Vol. 13, pp. 321–330.

Bigelow, C.A. 1989 "Analysis of Notched Metal Matrix Composites Under Tension Loading," *Metal Matrix Composites: Testing, Analysis, and Failure Modes*, ASTM STP 1032, edited by W.S. Johnson, American Society for Testing and Materials, Philadelphia, pp. 130–147.

Bigelow, C.A. and Bahei–El–Din, Y.A., 1983 "Plastic and Failure Analysis of Composites (PAFAC)," Program No. LAR–13183, COSMIC, University of Georgia.

Dvorak, G.J. and Bahei–El–Din, Y.A., 1982 "Plasticity Analysis of Fibrous Composites," *Journal of Applied Mechanics*, Vol. 49, pp. 327–335.

Dvorak G.J. and Johnson W.S., 1980 "Fatigue of Metal Matrix Composites," *International Journal of Fracture*, Vol. 16, p. 585.

Johnson, W.S. and Bigelow, C.A., 1989 "Elastic–Plastic Stress Concentrations Around Notches in Continuous Fiber Reinforced Metal Matrix Composites," *Test Methods and Design Allowables for Fiber Composites*, ASTM STP 1003, C.C. Chamis, ed., American Society for Testing and Materials, Philadelphia, pp. 261–275.

Johnson, W.S., Bigelow, C.A. and Bahei–El–Din, Y.A., 1983 "Experimental and Analytical Investigation of the Fracture Processes of Boron/Aluminum Laminates Containing Notches," NASA Technical Paper 2187.

Table 1 Common map of the general constants and arrays in the common block of the modified PAFAC program

Location in Common	Symbol	Description
1	IN	Total number of nodal points
2	IBN	Total number of input units for deflection boundary conditions (dbc)
3	IT	Total number of elements
4	IP	Total number of concentrated loads
5	IPRS	Number of pressure units
6	ITYPE	Indicator for material type
7	IMAT	Number of material types
8	INX	Number of last link to be executed
9	IMFI	Number of fiber angle types
10–17	N_i	Labels of current element vertices
18	M	Label of current element
19	ITY	Common size in words
20	ITEM	Temperature change type number
21	IMET	Material type number
22	ISUM	Number of reduced equilibrium equations
23	ISHUF	Node relabelling indicator
24	IORD	Number of words for reduced stiffness matrix
25	IORD1	IORD+1
26	ACEL	Body force per unit volume
27	INP	Indicator for output level
28	IPBG	Integer constant for element load vector
29	IPEN	Integer constant for element load vector
30	CONS	Constant for element load vector
31	IU	Pointer for diagonal element count array
32–34	G1,G2,G3	Direction cosines of acceleration vector
35–42	J1,J2,..J8	Pointer for J1W, J2W,N1,N2,..N6 arrays
43		not used
44	IBB	Pointer for IBB array
45	IBO	Pointer for IBO array
46	IID	Pointer for material constant array
47	IIA	Pointer for thermal coefficient array
48	IDT	Pointer for temperature change array
49	ICFI	Pointer for fiber angle array
50	IXX	Pointer for x–coordinate array
51	IYY	Pointer for y–coordinate array
52	IZZ	Pointer for z–coordinate array
53	IIC	Pointer for constant array of dbc units
54	IDEF	Pointer for unknown deflection array
55	IST	Pointer for overall reduced stiffness matrix
56	IIS	Pointer for element stiffness matrix
57	IERR	Error indicator
58	DT	Value of temperature change of element
59–61	AL1,AL2, AL3	Coefficients of thermal expansion of element referred to material axes

Table 1 Common map of the general constants and arrays in the common block of the modified PAFAC program (Cont.)

Location in Common	Symbol	Description
62–82	$D21_i$	Material matrix of an element
83–106	P_i	Load vector of an element
107–130	UV_i	Deflections of an element due to temperature change
131–138	X_i	Overall x–coordinates of element's vertices
139–146	Y_i	Overall y–coordinates of element's vertices
147–154	Z_i	Overall z–coordinates of element's vertices
155–161	XD_i	$X_i–X_1$
162–168	YD_i	$Y_i–Y_1$
169–175	ZD_i	$Z_i–Z_1$
176	IMES	Indicator for element descriptors input
177	IBUN	Indicator for boundary conditions input
178	ICOR	Indicator for nodal coordinates input
179	IPIR	Unit number for restart jobs input data
180	PRES	Pressure value for an element
181	IPR	Pointer for pressure array
182	ITAS	Unit number for intermediate storage
183	JMFI	Fiber angle type number
184	JMMZ	Pointer for dbc constant array
185	JMMY	Pointer for dbc second pair array
186	JMMX	Pointer for dbc first pair array
187	JPRS	Pressure type number
188	J10	Pointer for N_8 array
189	J9	Pointer for N_7 array
190	ISDT	Number of temperature change types
191–199		General usage area
200–325		User's area
326	IO	Formatted input unit number
327	MO	Formatted output unit number
328	MAXI	Substep printout indicator
329	MAXSI	Maximum number of substeps
330	NPRIN	Printout indicator for stress link
331	NSTAV	Strain computation method indicator
332	ER1	Convergence criterion for substep iteration
333	KHARD	Strain–hardening indicator
334	IHARD	Indicator for hardening information input
335	NAA	Count of vectors for saving element's response
336	LHD	Pointer for material plastic properties array
337	IELAS	Indicator: 0≡elastic element; 1≡otherwise
338–358	LLL_i	Pointer of NAA arrays
359	ITER	Step counter
360	ITERS	Substep counter
361	PNORMB	Norm of overall load vector
362	PNORM	Norm of residual load vector
363	IDIV	Indicator for subdivision of hexahedrons
365	CUMS	Cumulative load factor of current step
366	DESC	Incremental load factor of current step

Table 1 Common map of the general constants and arrays in the common block of the modified PAFAC program (Cont.)

Location in Common	Symbol	Description
367	LLD	Pointer for load factor–step number array
368	LCDP	Number of data points in LLD array
369	MPR	Internal indicator for stress link printout
370	JJ	Indicator: 0≡elastic–plastic; 1≡elastic
371	IPOT	Unit number of plotting file
372	YJ2	Effective stress
373	H	Plastic tangent modulus
374	ITAX	Unit number for cumulative data
375	ZZ	Scale factor of the step
376	IAMB	Ambient temperature of the structure
377	IYIT	Indicator for isothermal materials
378	PSCAL	Storage for original scale factor
379	ISECT	Internal indicator for isothermal processes
380	IYEL	Indicator for generation of stiffness matrix
381	KTERS	Internal storage for ITERS
382	QNORMB	Internal storage for PNORMB
383	D	Internal storage
384	IDD	Job identification
385	IRR	Job counter
386	NI	Count of elements in plastic zone
387	ICOUNT	Internal storage
388	ISCAL	Indicator for scaling in a restart job
389	IFLAG*	Indicator: 0≡regular load increment; >0 failure increment
390	IFAL*	Failure criterion indicator
391	NOFAL*	Pointer for element failure indicators
392	IIF*	Pointer for parameters of failure criterion

* New item.

Table 2 Input items of a standard PAC91 job

Input Item No.	Conditions	Input Item Contents	Format
1		$(B_i$, i=1,20), This line contains any alphanumeric message except the first four columns that should not contain the literal '????'	20A4
2		IN,IT,ITYPE,IBN,IP,IPRS,IMAT, ISDT,IMFI,LCDP,INX,INP,ISHUF, ICOR,IBUN,IMES,IPIR,KHARD,IHARD, IYIT,ITAS,IAMB,NAA,IFAL,G1,G2,G3, ACEL (See Table 3 for details)	2I4,I1,3I4,4I2, 11I1,I5,2I2,6x, 3F5.4,E10.3
3	ITYPE=0	$(i,E_i,\nu_i,\alpha_i$, i=1,IMAT)	(3(I2,3E8.2))
	ITYPE=1	$(i,v_{fi},E_{mi},\nu_{mi},E_{fi},\nu_{fi}$, i=1,IMAT)	(I2,F5.3,4E12.5)
4	KHARD=1 1≤IHARD≤9	$(i,j,(\bar{\epsilon}^{p}_{k},\bar{\sigma}_k$, k=1,j), i=1,IMAT), 2≤J≤10*IHARD	(2I3/(10E8.3))
	KHARD=1 IHARD=0	Input should be prepared as required by user's subroutines YHARP,YHAR2P,YHAR4P	
	KHARD=0	No input	
5		$(K_i,C_i$, i=1,LCDP)	(8(I2,E8.2))
6	1≤IPRS≤99	$(i,p_i$, i=1,IPRS)	(8(I2,E8.2))
	IPRS=0	No input	
7	1≤IMFI≤99	$(i,\varphi_i$, i=1,IMFI) (see Bigelow and Bahei–El–Din (1983) for details)	(8(I2,E8.5))
	IMFI=0	No input	
8	ICOR=0	$(j,x_j,y_j,z_j$, j=1,IN)	(2(I4,3E12.4))
	ICOR=1	Input should be prepared as required by user's subroutine CORG	

Table 2 Input items of a standard PAC91 job (cont.)

Input Item No.	Conditions	Input Item Contents	Format
9	IBUN=0	$(i_k,j_k,i'_k,j'_k,a_k,$ k=1,IBN)	(5(I4,I1,I4, I1,F6.0))
	IBUN=1	Input should be prepared as required by user's subroutine BUNG	
10	IMES=0	$(MM_m,J1W_m,J2W_m$,N1,...N8, m=1,IT) MM_m=–(m–(m/1000)*1000) interpreted in Fortran integer arithmetic sense; J1W=100*IMET+JMFI, J2W=100*JPRS+ITEM	(20I4)
	IMES=1	Input should be prepared as required by user's subroutine MESG	
11	ISHUF=0,1	No input	
	ISHUF=2	$(N_i$, i=1,N)	(20I4)
	ISHUF=3	$(N_i,IMAX_i$, i=1,IN)	(20I4)
12	1≤IP≤9999	$(i_l,j_l,P_l$, l=1,IP)	(5(I4,I1,E11.4))
	IP=0	No input	
13*	IFAL=0	Input should be prepared as required by user's subroutine DFAIL	
	IFAL=–1	$(\tau_i^u,\sigma_i^u$, i=1,IMAT)	(2E10.3)
	IFAL=1 ITYPE=1	$(\tau_i^{uf},\sigma_i^{uf}$, i=1,IMAT)	(2E10.3)
14		MAXI,MAXSI,NPRIN,NSTAV,ER1,IDIV, IPOT,ITAX,IYEL,IDD,IRR,TEST (See Table 4 for details)	4I2,E12.5,I1,I2 I1,I4,A4,I2,43X, 'END'

* New item.

Table 3 Description of input item 2 of a standard PAC91 job

Name	Columns	Format	Range	Description
IN	1–4	I4	8–9999	Total number of mesh points
IT	5–8	I4	1–9999	Total number of elements
ITYPE	9	I1	0–1	Material indicator: 0 – homogeneous, isotropic, 1 – fibrous composite
IBN	10–13	I4	3–9999	Total number of deflection boundary condition units
IP	14–17	I4	0–9999	Total number of concentrated load units
IPRS	18–21	I4	0–9999	Total number of different pressures
IMAT	22–23	I2	1–99	Total number of different materials
ISDT	24–25	I2	0	Total number of different temperature changes
IMFI	26–27	I2	0–99	Total number of different fiber angle types: 0 if ITYPE=0; 1–99 if ITYPE=1
LCDP	28–29	I2	1–25	Total number of load factor–step number points
INX	30	I1	1–4	Number of last link to be executed
INP	31	I1	0–2	Printout indicator: 0 – minimum output; 1 – intermediate output; 2 – detailed output
ISHUF	32	I1	0–3	Relabelling indicator: 0 – no relabelling; 1 – iterate to relabel without reading data; 2 – read data and iterate to relabel; 3 – relabel as shown on data lines
ICOR	33	I1	0–1	Indicator for nodal coordinate generation: 0 – read from data lines; 1 – subroutine CORG
IBUN	34	I1	0–1	Indicator for deflection boundary conditions generation: 0 – read from data lines; 1 – subroutine BUNG
IMES	35	I1	0–1	Indicator for element descriptors generation: 0 – read from data lines; 1 – subroutine MESG
IPIR	36	I1	0–9	Unit number for storage of information necessary for restart jobs
KHARD	37	I1	0–1	Strain hardening indicator: 0 – nonhardening (only if ITYPE = 1); 1 – kinematic hardening
IHARD	38	I1	0–9	Indicator for hardening information: 1–9, piecewise linear; 0 , not piecewise linear or nonhardening

Table 3 Description of input item 2 of a standard PAC91 job (cont.)

Name	Columns	Format	Range	Description
IYIT	39	I1	0	Indicator for yield stress–temperature information
ITAS	40	I1	1–9	Unit number for intermediate storage
IAMB	41–45	I5	0	Ambient temperature of material
NAA	46–47	I2	15,21	Number of arrays for saving element information: 15 if ITYPE=0; 21 if ITYPE=1
IFAL*	48–49	I2	–1,0,1	Failure criterion indicator: –1 : program–defined based on overall stress 0 : user–defined via subroutine UFAIL 1 : program–defined based on fiber stress (only if ITYPE=1)
G1	56–60	F5.4	–1 to 1	X–component of unit acceleration vector
G2	61–65	F5.4	–1 to 1	Y–component of unit acceleration vector
G3	66–70	F5.4	–1 to 1	Z–component of unit acceleration vector
ACEL	71–80	E10.3	any	Magnitude of acceleration vector times unit mass, i.e. unit weight

* New item.

Table 4 Description of input item 14 of a standard PAC91 job and input item 6 of a restart PAC91 job

Name	Columns	Format	Range	Description
MAXI	1–2	I2	0–99	Printout indicator for stress link: > 0 – printout is produced at every step and at every substep; 0 – printout is produced at the steps given in input item 5 and at the end of failure steps
MAXSI	3–4	I2	–9 to 99	Maximum number of substeps; if ≤ 0, 10 substeps are assumed
NPRIN	5–6	I2	0–2	Printout level for stress link: 0 – stresses and forces are printed 1 – stresses, forces, and strains are printed 2 – stresses, forces, strains and displacements are printed
NSTAV	7–8	I2	0–1	Indicator for strain computation from displacements: 0 – best fit strain tensor is obtained 1 – computations by user's subroutine STNA
ER1	9–20	E12.5	any ≥ 0	Convergence tolerance; if 0, 0.01 is assumed
IDIV	21	I1	0–1	Indicator for subdividing hexahedrons: 0 – in two ways; 1 – in one way
IPOT	22–23	I2	0–9	Unit number for storage of information necessary for plotting
ITAX	24	I1	1–9	Unit number for storage of cumulative results
IYEL	25–28	I4	1–9999	Number of elements allowed to yield in a substep without generation of stiffness matrix
IDD	29–32	A4	any	Identification; not blank if IPOT > 0
IRR	33–34	I2	1–99	Sequential job label: 1 for standard job > 1 for restart job
TEST	78–80	'END'		Indicator for end of current job

Table 5 Input items of a restart PAC91 job

Input Item No.	Conditions	Input Item Contents	Format
1		$(B_i$, i=1,20), This line contains any alphanumeric message except the first four columns that should contain the literal '????'	20A4
2		NIP,NIPRS,NINP,NIHARD,JTAS,NIPIR ISCAL,NLCDP (See Table 6 for details)	I4,I2,10X,5I1,I2
3		$(K_i,C_i$, i=1,LCDP)	(8(I2,E8.2))
4	NIPRS=IPRS*, $1 \leq$IPRS≤ 9999	$(i,p_i$, i=1,NIPRS)	(8(I2,E8.2))
	NIPRS≠IPRS or IPRS=0	No input	
5	NIP=IP‡ $1 \leq$IP≤ 9999	$(i_l,j_l,P_l$, l=1,NIP)	(5(I4,I1,E11.4))
	NIP≠IP or IP=0	No input	
6		MAXI,MAXSI,NPRIN,NSTAV,ER1,IDIV, IPOT,ITAX,IYEL,IDD,IRR,TEST (See Table 4 for details)	4I2,E12.5,I1,I2 I1,I4,A4,I2,43X, 'END'

* IPRS = Number of pressure types of the preceding job

‡ IP = Number of concentrated load units of the preceding job

Table 6 Description of input item 2 of a restart PAC91 job

Name	Columns	Format	Range	Description
NIP	1–4	I4	0–9999	Total number of concentrated load units
NIPRS	5–6	I2	0–99	Total number of different pressures
NINP	17	I1	0–2	Printout indicator: 0 – minimum output; 1 – intermediate output; 2 – detailed output
NIHARD	18	I1	0–9	Indicator for hardening information: 1–9, Information of standard job is used; 0 , user's subroutines YHARP, YHAR2P, YHAR4P are used
JTAS	19	I1	1–9	Unit number for restart input information
NIPIR	20	I1	0–9	Unit number for storage of information necessary for succeeding restart job
ISCAL	21	I1	0–1	Scaling indicator: 0 – load factors of input item 3 are not scaled with SCAL* before their use; 1 – load factors of input item 3 are scaled with SCAL before their use
NLCDP	22–23	I2	1–LCDP‡	Total number of load factor–step number points

* SCAL = Load factor for initial yielding computed in the initial standard job

‡ LCDP = Number of data points of load factor–step number in the initial standard job

```
      SUBROUTINE DFAIL

C     To read in user–provided input data related to failure

      IMPLICIT DOUBLE PRECISION (A–H,O–Z)
      COMMON/ONE/IN
      COMMON/TWO/XIN
      DIMENSION  IA(401),AA(401),S(1),IS(1)
      EQUIVALENCE  (IN,IA(1)),(XIN,AA(1))
      EQUIVALENCE  (IA(7),IMAT),(IA(326),IO),(AA(392),IIF)
      EQUIVALENCE  (IA(401),S(1)),(AA(401),IS(1))

C..The user provides here statements which read failure parameters for every material
C..type up to IMAT.  The data are to be stored in array AA in the area specified by the
C..pointer IIF such that all parameters for material type 1 are listed in the first 10
C..positions followed by all parameters of material type 2 in the next 10 positions, etc.
C..The read statement should specify the input unit number as IO.

C..Example, read one material parameter

      DO 10 I=1,IMAT
       IPOINT=IIF+(IMAT–1)*10
       READ (IO,*)  AA(IPOINT+1)
10    CONTINUE

C..End of Example

      RETURN
      END
```

Fig. 1 Listing of user–defined subroutine DFAIL

```
      SUBROUTINE UFAIL(KFAIL)
C     To provide the user's failure criterion and check if it is satisfied for element M
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
      COMMON/ONE/IN
      COMMON/TWO/XIN
      DIMENSION IA(401),AA(401),IS(1),S(1)
      DIMENSION LLL(21)
      EQUIVALENCE (IN,IA(1)),(XIN,AA(1))
      EQUIVALENCE (IA(6),ITYPE),(IA(18),M),(IA(21),IMET),(IA(46),IID)
      EQUIVALENCE (IA(338),LLL(1)),(AA(392),IIF),(IA(401),IS(1)),(AA(401),S(1))
      EQUIVALENCE (AA(236),DIR(1,1)),(AA(274),E66(1,1))
      DIMENSION DIR(3,3),E66(6,6)
      DIMENSION SSO(6),SSL(6),SSM(6),SSF(6)

      KFAIL=-1

C..Overall stresses of the current element are stored in array SSO. For a composite
C..element, the stresses contained in SSO are described in the global coordinate system
C..while those given in array SSL are described in the local coordinate system where
C..direction 3 is the fiber axial direction, and plane 1-2 is the transverse plane. The
C..corresponding matrix and fiber stresses are stored in arrays SSM and SSF,
C..respectively. The stress components are listed in arrays SSO,SSL,SSM,SSF in the
C..order S11,S22,S33,S12,S13,S23.

      DO 10 I=1,6
       SSO(I)=AA(LLL(I)+M)
10    CONTINUE

      IF (ITYPE.GT.0)  THEN
       CALL CODI4
       CALL TRANS4
       DO 30 I=1,6
        SSL(I)=0.0D0
        DO 30 J=1,6
         SSL(I)=SSL(I)+E66(I,J)*SSO(J)
30     CONTINUE
       IIDI=IID+(IMET-1)*5
       VF=AA(IIDI+1)
       VM=1.0D0-VF
       DO 20 I=1,6
        SSM(I)=AA(LLL(I+9)+M)
        SSF(I)=(SSL(I)-VM*SSM(I))/VF
2      CONTINUE
      ENDIF
```

Fig. 2 Listing of user–defined subroutine UFAIL

```
C..This part is prepared by the user to check the failure criterion for the current
C..element. Input data entered by the user in subroutine DFAIL are stored in array AA
C..in the area specified by the pointer IIF in the following order: parameters for
C..material 1 are located in the first 10 positions, and those for material 2 are located in
C..the next 10 positions, etc. If the failure criterion is satisfied, set KFAIL=1.

C..Example, failure occurs when the axial stress in the fiber reaches σ^uf

      IPOINT=IIF+(IMET-1)*10
      SULT=AA(IPOINT+1)
      IF(DABS(SSF(3)).GE.SULT) KFAIL=1

C..End of Example

      RETURN
      END
```

Fig. 2 Listing of user–defined subroutine UFAIL (Cont.)