A Finite Element Program (FECGS): Post-Processor FEPOST

User's Manual

Version 1.0

Guoyu Lin

GKSS-Forschungszentrum Geesthacht GmbH

Max-Planck-Str.

D-21502 Geesthacht, Germany

July, 1997

PREFACE

FECGS is a finite element program system (hereafter, program) which includes an implicit FE code for stress analysis and crack growth simulation using micromechanical models, e.g. the cohesive zone model and the Gurson model, a pre-processor (FEMESH) and a post-processor (FEPOST). The program is specially developed for the purpose of fracture mechanics analysis. However, the applicability of the program is rather general.

The program is for academic and research purpose only. The program is provided "as is" without express or implied warranty. The author assumes no responsibility for any errors that may appear using any part of the program. However, any comments, suggestions and bug reports are appreciated. FEPOST is also able to access ABAQUS result file (.fil).

This manual provides the basic reference document for FEPOST.

Point of contact:
Guoyu Lin
Institute of Materials Research
GKSS Research Center
D-21502 Geesthacht
Germany
Tel: 49-4152-872539
Fax: 49-4152-872534
E-mail: guoyu.lin@gkss.de

Contents

Chapter 1

GENERAL FEATURES

1.1 Introduction

FEPOST is part of finite element program system (FECGS) for the purpose of post-processing the results files. FEPOST is executed uninteractively by providing a file of commands. The present version is available for the IBM workstation RISC 6000. The main input of the program is a file which indicates the options required, and gives the data associated with those options. Current version of FEPOST is also able to process the output file (.fil) from a finite element analysis run by ABAQUS using the commands *NODE FILE, ..., and *EL FILE, ... FEPOST deals only with nodal variables provided by a FE analysis. For the element variables, FEPOST will interpolate the element variables from the gauss points to the element nodes. Thus, the element varibles from an analysis run by ABAQUS should be recorded in the gauss integration points.

Major capabilities of FEPOST include:

- Tabulating results.
  - Tabulating a list of quantities, e.g. displacements at given coordinates, summing the nodal reaction force up for a given array of nodes, averaging stresses and strains over a material volume etc.
  - Displacement along a given curve (referring to displacement profile).
  - Distribution of solution variables along a given curve, for examples, the stresses distribution.
  - Q-stress at a given point or along a curve (a special option for the purpose of fracture mechancs).

  - calculating the volume integration for the cleavage failure probability (a special option for the purpose of fracture mechanics).
  - Quantifying the elastic T-stress (a special option for the purpose of fracture mechanics).

- Generating graphic plots.
  - Plotting finite element meshes.
  - Creating the contour plots.
  - Creating the deformed mesh plots.
  - Generating the plots of plastic zone.

1.2 Command entry

Uninteractive execution of FEPOST involves entry of alphanumerica commands. All entries should be provided in an input file.

The * prompt indicates a command line to be executed, with any parameters and required values. All the commands should be given in capitals. The command and its related parameters should be given in one line.

Parameter values are entered with the "=" sign: for example, VARIABLE=MISES. Some parameters require two or more values. Enter the values grouped by parentheses: for example, COORDINATES=(1.0, 1.0). Command name and parameters may be abbreviated on input - one must give enough character to make the entry unique in context, e.g. VAR → VARIABLES and COORD → COORDINATES.

Some commands require lines of data to be entered. All entries are free format, with comma "," or blanks as the separators. In this manual, the type of data required is indicated by "A" for a character string, "F" for a floating point number, and "I" for an integer.

Any line that begins with ** is treated as a comment line.

All commands and related inputs are reported in the post output file (_pout).

1.3 Definition of the coordinate system

In FEPOST a fixed rectangular Cartesian coordinate system with axes x_i, $i = 1, 2, 3$, is used. system.

1.4 Executing FEPOST

FEPOST is executed by directly entering 'fepost'. The program will automatically looking for the FECGS or ABAQUS results output files (.fil) and displays the files in a list. Enter the corresponding number to indicate files to be processed. Then FEPOST will inquire the default input file with extension .post. If there is no default input file, the program will waiting for an input file.

For example, suposse that there five output files: test1.fil, test2.fil, test3.fil, test4.fil and test5.fil, in the current directory. To execute FEPOST enter:

fepost

The system will respond with:

```
FEPOST Release 1.0

***** Automatic sreaching output files (.fil) *****

The filenames with extension of .fil are:

1 —— test1.fil
2 —— test2.fil
3 —— test3.fil
4 —— test4.fil
5 —— test5.fil

***** More than one files are found by program *****

Enter a file identifier:
```

Enter the file identifier listed, for example, 4. FEPOST will inquire its input file test4.post. If there is no test4.post existed, the program prompts:

```
ENTER FILENAME OF INPUT DATA:
```

and waiting for the FEPOST input file.

Chapter 2

INPUT, OUTPUT AND PLOT FILES

2.1 FEPOST input file

- FECGS/ABAQUS OUTPUT FILES - An output file from an analysis run must be available to provide the model geometry and solution data base from which the FEPOST output will be extracted. By default, this file name will be used as FEPOST output file name.
- FEPOST INPUT FILES - A FEPOST input file should be provided, which involves all the commands, related parameters and the required valuesto be executed.

2.2 FEPOST output file

- ERROR FILE - An error file is generated with FEPOST output file and attached string _perr, if there are any errors during FEPOST is executed.
- FEPOST OUTPUT FILE - Echo the FEPOST commands, the filename is the FEPOST output file name with attached string _pout.
- INFORMATION FILE - Writing the resulted information during executing FEPOST, the filename is the FEPOST output file name with attached string _pinf.
- JOURNAL FILE - Writing the information of execution of FEPOST, for examples, executed subroutines and CPU time needed, the filename is the FEPOST output file name with attached string _pjnl.
- NEUTRAL PLOT FILE - Save the resulted device-independent plot file. The plot file extension is .mpl with a filename matching that of the output file name.

- PATH DATA FILES - The results along a given line through the model using the *PATH COMMAND, for example, the stress distribution along a line. The file name is output file name with attached string _path××, the last two characters are the number of *PATH command in the form of two digits, eg. _path01 for the first *PATH, _path02 for the second *PATH and so on.

- PROFILE FILES - Resulting profile data using *PROFILE command, the filename is the output file name with attached string _ppro××, the last two characters are the number of *PROFILE command in the form of two digits as that define in PATH DATA FILES.

- TABLE RESULTS FILES - A table generated by command *PARAMETER, the filename is FEPOST output file name with attached string _petm01.

2.3 FEPOST plot file, graphic display and translation

FEPOST generates a device-independent graphic file, the neutral plot file. The data format of the neutral file is according to ABAQUS/POST. The neutral plot file (.mpl) is a binary file. It contains plot commands in a device-independent format which can be converted into device specific commands by the device driver. The available device drivers are x11 (displaying in X window from IBM RISC 6000), ps (translating to postscript) and pft (translating to Mac ProFit with a special ProFit program to draw the plot). The FEPOST neutral plot files are also accessible for the ABAQUS device drivers.

2.4 Color definition

Valid color definition in FEPOST using color identifiers range from 0 to 15. The color names associated with these identifiers, and their color spectrum definitions are:

0	Black	0% for all colors
1	White	100% for all colors
2	Grey(gray)	60% for all colors
3	Blue	100% blue
4	Medium blue	100% blue - 60% green
5	Sky blue	100% blue - 80% green
6	Cyan	100% blue - 90% green
7	Aqua	100% green - 80% blue
8	Spring green	100% green - 55% blue
9	Green	100% green
10	Chartreuse	100% green - 70% red
11	Yellow	100% green - 100% red
12	Gold	100% red - 85% green
13	Dark gold	100% red - 70% green
14	Orange	100% red - 50% green
15	Red	100% red

Chapter 3

COMMAND DESCRIPTION

3.1 *CONTOUR: Drawing contour plots

The command *CONTOUR draws the contour plots. The current version is available only for two-dimensional problems.

The following parameters are required:

OUTPUT	Set this parameter equal to MPL for generating a neutral plot file.
VARIABLE	Set this parameter equal to the name of variable to be contoured, e.g. VAR=S22.

The following parameters are optional:

UNDEFORM	Set this parameter equal to YES for drawing the contour in undeformed configuration. The default is OFF.
QUILT	Set QUILT=ON to force a single contour value per element for shaded (filled) contour plots (without interpolation). The default is OFF.
SHADED	Set this parameter equal to YES to draw a shaded (filled) contour plot. The default is NO.
LEVEL	The number of contour levels. The default is 10.
MAGNIFICATION	Magnification factor to be applied to the displacements of the model for display. The default is 0.0 (no magnification).
EL PLOT	Set EL PLOT=NO to suspend drawing the element boundary in the contour plots. By default, EL PLOT=YES.
ELSET	Set this parameter equal to the name of element set in which the contour plot is generated. By default, the entire model is drawn.

Examples:

1. Use the following entry to draw the contour plot in the element set E0000001, contours are plotted in the undeformed model and filled with corresponding colors, element boundaries are not plotted, number of contour levels is 8.

```
*CONTOUR,OUTPUT=MPL, UNDEFORM=YES, EL PLOT=NO, SHADED=YES, LEVEL=8, ELSET=E0000001
```

2. Use the following entry to draw the contour plot in the deformed configuration with actual displacement, contours are filled, element boundaries are not plotted, there are 8 contour levels, contour plot is forced on a single contour value per element.

*CONTOUR,OUTPUT=MPL, QUILT=ON, MAG=1.0, SHADED=YES, LEVEL=8, ELSET=E0000001

3.1.1 *DETAIL: Defining a part of the model

Command *DETAIL, suboption of *CONTOUR, is used to specify a part of the model to be plotted. The *DETAIL specification only affects the current contour plot. By default the entire model is plotted.

The following parameters are optional:

ELSET	Set this parameter equal to a element set name defining the elements to be included in the detail.

Format	**Entry**

The following data line is needed if the ELSET parameter is omitted.

6F	X1, Y1, Z1, X2, Y2, Z2. Here (X1, Y1, Z1) define the coordinates of minimum model space bounds of the part of the model to be plotted, and (X2, Y2, Z2) define the coordinates of maximum model space bounds of the part of the model to be plotted.

Examples:

1. Use the following entry to creat a part of the model defined by element set E0000001.

```
*DETAIL, ELSET=E0000001
```

2. Use the following entries to define a part of the model bounded by (0.0, 0.0, 0.0) and (10.0, 10.0, 0.0).

```
*DETAIL
0.0, 0.0, 0.0, 10.0, 10.0, 0.0
```

3.1.2 *HISTORY: Selecting time increments

*HISTORY, suboption of *CONTOUR, is used to specify the time increments to be plotted. By default, all the time increments recorded are plotted.

Format	**Entry**

The following data lines are needed if the *HISTORY suboption is used:

For step increment base

A, 3I	"STEP" K, INC1, INC2. "STEP" is case sensitive. K - Step number, INC1 - first incremental number, INC2 - last incremental number. FEPOST draws all the increments between INC1 and INC2 at step K.

or, in crack extension base

A, I, nF	"Da" n, Da 1, Da 2, . . . , Da n. Here, "Da" is case sensitive, n is number crack extension to be selected, Da i is ith selected crack extension.

Examples:

1. Use the following entries to draw contour plot in the corresponding time increments: step 1 all increments within increments 5 and 100; step 2 increment 50; and step 3 increment 100.

```
*HISTORY
STEP 1, 5 100
STEP 2, 50 50
STEP 3, 100 100
```

2. Use the following entries to draw contour plot for the time increment corresponding to the crack extensions at, or about at 0.0, 1.0, 2.0, 4.0 and 6.0.

```
*HISTORY
Da 5, 0.0, 1.0, 2.0, 4.0, 6.0
```

3.1.3 *NORMALIZE: Normalizing the coordinates

Command *NORMALIZE, suboption of *CONTOUR command, is used for normalizing the coordinates in the contour plot.

The following parameter is required:

CONSTANT	Set this parameter equal to the value to normalize the coordinates in the model.

Examples:

1. Use the following entry to normalize the coordinates by 100.0.

```
*NORMALIZE, CONSTANT=100.0
```

3.1.4 *LEVEL: Defining the contour level

Command *LEVEL, suboptions of *CONTOUR command, is used to define the contour level of the variable to be contoured. The contour levels are uniformly distributed according to the number of contour levels, or may be entered individually.

By default, the first contour level is taken as minimum value of variable plus 1% of difference between maximum and minimum values, the last contour level is taken as minimum value of variable plus 90% of difference between maximum and minimum values.

The following parameters are optional:

MINIMUM	Set this parameter equal to the value of the minimum contour level in contour plot. The default is the actual minimum value.
MAXIMUM	Set this parameter equal to the value of the maximum contour level in contour plot. The default is the actual maximum value.

Format	**Entry**

The following data line is required if both parameters MINIMUM and MAXIMUM are omitted:

I, nF	n, level 1, level 2, ..., level n. n is the number of contour levels.

Examples:

1. Use the following entry to define the minimum and maximum contour levels.

```
*LEVEL, MIN=100.0, MAX=300.
```

2. Use the following entries to define the contour levels.

```
*LEVEL
5 100.0, 200.0, 300.0, 400.0, 500.0
```

3.1.5 *REF LENGTH: Drawing a reference scale

*REF LENGTH, Suboption of *CONTOUR, is used to draw a reference length along the horizontal direction from left to right in the contour plots.

The following parameters are required:

LENGTH	Set this parameter equal to the reference length (scale).
COORD	Set this parameter equal to the coordinates (X, Y, Z) where the reference length is going to be drawn, e.g. COORD=(X, Y, Z).
UNIT	Set this parameter equal to the definition of the corresponding reference length.

Examples:

1.The following entry is used to draw a reference length of 10 *mm* started at point (10., -1.0, 0.0) and ended at point (20., -1.0, 0.0).

```
*REF LENGTH, LENGTH=10.0, COORD=(10., -1.0, 0.0), UNIT=10.0 mm
```

2.The following entry is used to draw a reference length of 10 *mm* started at point (10., -1.0, 0.0) and ended at point (20., -1.0, 0.0), which is defined as Ro.

```
*REF LENGTH, LENGTH=10.0, COORD=(10., -1.0, 0.0), UNIT= Ro
```

3.1.6 *DRAW CRACK: Drawing current crack

*DRAW CRACK command, suboption of *CONTOUR, draws the current crack in the contour plot. This option is specified for the crack growth simulation using the cohesive zone model. Crack tip is indicated by a line which is perpendicular to the crack surface.

The following parameter is required:

ELSET	Set this parameter equal to the name of the interface element set which is used for the crack growth simulation.

The following parameters are optional:

DIRECTION	Set this parameter equal to the crack growth direction indicated by X (or Y, or Z). By default, DIR=X.
TIPONLY	Set this parameter equal to YES for drawing crack tip only, the defaul is TIPONLY=NO.
COLOR	Set this parameter equal to the color identifier which is used to draw the crack, see section 2.4 for definition of the color identifiers. By default, the color is defined by the program.

Format **Entry**

The following data line is required for this option:

6F	X1, Y1, Z1, X2, Y2, Z2. These six real numbers are used to draw a line from point (X1, Y1,Z1) to point (X2, Y2, Z2) for indicting the crack tip. This line will be moved with crack growth. IF suboption *NORMALIZE is used (Xi, Yi, Zi) shall be the normalized value.

Examples:

1. The following entries are used to draw the crack tip from point (0.0, 0.0, 0.0) to point (0.0, -0.6, 0.0) only.

```
*DRAW CRACK, DIR=Y, ELSET=CZMELE, TIPONLY=YES
0.0, 0.0, 0.0, 0.0, -0.6, 0.0
```

2. The following entries are used to draw the crack and the crack tip. The latter is indicated by a line from point (0.0, 0.0, 0.0) to point (0.0, -0.6, 0.0) and moved with the crack growth. Element set CZMELE is used to draw the actual crack.

```
*DRAW CRACK, DIR=Y, ELSET=CZMELE, COLOR=15
0.0, 0.0, 0.0, 0.0, -0.6, 0.0
```

3.1.7 *DRAW CZM: Drawing the cohesive zone in the contour plot

command *DRAW CZM, suboption of *CONTOUR, draws the currently active cohesive zone in the contour plot. This option is specified for the crack growth simulation using the cohesive zone model.

The following parameter is required:

ELSET	Set this parameter equal to the name of the interface element set which is used for the crack growth simulation.

The following parameter is optional:

COLOR	Set this parameter equal to the color identifier which is used to draw the cohesive zone, see section 2.4 for definition of the color identifiers. By default, the color is defined by the program.

Examples:

1.The following entry is used to draw the current cohesive zone.

```
*DRAW CZM, ELSET=CZMELE, COLOR=4
```

3.2 *CRACK: Calculating the crack extension

The *CRACK command is used to calculate the crack extension from the model based on a given criterion. A reference point is first defined. The crack extension is calculated as the maximum distance from reference point to the point of farest element node of the failed elements. These failed elements should be connected each other, such that they form an open crack.

The following parameters are required:

VARIABLE	Set this parameter equal to name of the variable which is used as the criterion for removing elements, for example, VAR=SDV2. If the value of the variable at any integration point of an element satisfies the given criterion, this element is defined to be failed.
GT	Set this parameter equal to the critical value of the VARIABLE for defining the failure elements. If the variable is greater than this value, the element is failed.

The following parameters are optional:

ELSET	Set this parameter equal to element set name in which the elements may be removed. The default is the whole model.
INPUT	Set INPUT=COORD (reference point for calculating the crack extension is given by the coordinates), INPUT=NODE (reference point is defined as a finite element node). If INPUT parameter is omitted, the reference point is defined by the coordinates (0.0, 0.0, 0.0).

Format **Entry**

The following data line is need if INPUT parameter is specified:

For the case of INPUT=COORD,

3F	X, Y, Z. The coordinates of the reference point.

Or, for the case of INPUT=NODE,

I	NODE. Nodal label of the reference point.

Examples:

1. Use the following entries to calculate crack extension in the element set E0000001. If the variable SDV2 is great than 0.10, this element is failed. The crack extension is calculated as the maximum distance from reference point (0.0, 1.0, 0.0) to the point of farest element node in the failed elements.

```
*CRACK, INPUT=COORD, VARIABLE=SDV2, GT=0.10, ELSET=E0000001
0.0, 1.0, 0.0
```

2. The following entries are used to calculate crack extension in the element set E0000001. If the variable SDV2 is great than 0.10, this element is failed. The crack extension is calculated as the maximum distance from reference point (node 10 in the model) to the point of farest element node in the failed elements.

```
*CRACK, INPUT=NODE, VARIABLE=SDV2, GT=0.10, ELSET=E0000001
10
```

3.3 *DEFORM: Drawing the deformed mesh

The command *DEFORM draws the deformed configuration. In the current version this option is available only for two-dimensional problems.

The following parameter is required:

OUTPUT	Set this parameter equal to MPL for generating a neutral plot file.

The following parameters are optional:

ELSET BOUND	Set this parameter equal to YES for drawing element set boundaries. This parameter is specially for the analysis involving several material regions.
UNDEFORM	Set this parameter equal to YES for drawing also the undeformed mesh. The default is No.
MAGNIFICATION	Set this parameter equal to displacement magnification factor for the deformed shape. The default value is 1.0.

Examples:

1. Use the following entry to draw the deformed and undeformed meshes, the part of the model to be plotted is defined by the element set E0000001.

```
*DEFORM,OUTPUT=MPL, UNDEFORM=YES
```

2. Use the following entry to draw the deformed mesh of the entire model. Element set boundaries are plotted also.

```
*DEFORM,OUTPUT=MPL, ELSET BOUND=YES
```

3.3.1 *DETAIL: Creating a part of the model

*DETAIL, suboption of *DEFORM, is used to specify a part of the model to be plotted. The *DETAIL specification only affects the current deformed plot. By default the entire model is plotted.

The following parameter is optional:

ELSET	Set this parameter equal to a element set name defining the elements to be included in the detail.

Format **Entry**

The following data line is needed if ELSET parameter is omitted.

6F	X1, Y1, Z1, X2, Y2, Z2. Here, (X1, Y1, Z1) define the coordinates of minimum model space bounds of the part of the model to be plotted, and (X2, Y2, Z2) define the coordinates of maximum model space bounds of the part of the model to be plotted.

Examples:

1. Use the following entry to creat a part of the model defined ny element set E0000001.

```
*DETAIL, ELSET=E0000001
```

2. Use the following entries to define a part of the model bounded by (0.0, 0.0, 0.0) and (10.0, 10.0, 0.0).

```
*DETAIL
0.0, 0.0, 0.0, 10.0, 10.0, 0.0
```

3.3.2 *HISTORY: Selecting the time increments

Command *HISTORY, a suboption of *DEFORM, is used to specify the time increments to be plotted. By default, all the time increments recorded are plotted. Referring to section 3.1.2 for the details of explanation of the command, the definition of parameters and its required values and the examples.

3.3.3 *COLOR TABLE: Defining a color table

Command *COLOR TABLE, a suboption of *DEFORM, is used to specify a color table. In particularly, if the model contains several mesh areas corresponding to different materials, one may wish to plot the different material regions based on a predefined color table.

Format	**Entry**

The following data line is needed for *COLOR TABLE command:

(n+1)I	n, color 1, color 2, . . . , color n. n is number of colors, and color i is the color identifier defined in the section 2.4.

Examples:

1.The following entries are used to specify a color table of 6 colors with the color identifiers, 4, 6, 8, 10, 12 and 15. The color identifiers have been defined in section 2.4.

```
*COLOR TABLE
6, 4, 6, 8, 10, 12, 15
```

3.4 *EL REMOVE: Removing the element from the model

The *EL REMOVE is used to remove the elements from the model based on a given criterion. The command is specially for the application of the local approach based fracture mechanics analysis.

The following parameters are required:

VARIABLE	Set this parameter equal to identifier of the variables which is used as the criterion for removing elements. If a variable in any integration point of an element satisfies the given criterion, this element is removed and it will not appear in, for examples, the deformed mesh plot and the contour plot.
GT	Set this parameter equal to a critical value of the VARIABLE for removing the element, if the defined variable is greater than this values, the element will be removed.

The following parameter is optional:

ELSET	Set this parameter equal to element set name in which the elements may be removed. The default is the whole model.

Examples:

1. Use the following entry to conduct element remove in the element set E0000001. If the variable SDV2 in any gauss point of an element is greater than 0.10, this element will be removed.

```
*EL REMOVE, VARIABLE=SDV2, GT=0.10, ELSET=E0000001
```

2. Use the following entry to conduct element removing in the whole model.

```
*EL REMOVE, VARIABLE=SDV2, GT=0.10
```

3.5 *END POST: End command input

This command is used to end the command input in FEPOST.

3.6 *INT VAR: Defining the variables for interpolation

The command *INT VAR defines the variables to be interpolated from element gauss integration points to the element nodes, so that the commands, such as *CONTOUR, *PATH ..., can be conducted.

Format	**Entry**

The following data line is needed for command *INT VARIABLE:

A	Given a list of variables to be interpolated, the variables are separated by comma (,) or blanks.

Examples:

1. The following entries are used to conduct interpolation of the variables S11, S22, S12 and SDV1.

```
*INT VAR
S11, S22, S12, SDV1
```

3.7 *NORMALIZE: Normalizing the stresses and strains

The *NORMALIZE is used to normalize the stresses and strains created by the FE analysis.

The following parameter is required:

SIGMA	Set this parameter equal to the reference value for normalizing the stresses.

The following parameters are optional:

YOUNG	Set this parameter equal to Young's modulus.
EPS	Set this parameter equal to reference strain for normalizing the strains. The default value is SIGMA/YOUNG if YOUNG is given, otherwise 1.0.
ELSET	Set this parameter equal to element set name in which the stresses and strains will be normalized. The default is the whole model.

Examples:

1. Use the following entry to normalize the stresses and strains in the element set E0000001. The stresses are normalized by 300.0 and the strains are normalized by 0.00333 (=300/90000.0).

```
*NORMALIZE,SIGMA=300.0, YOUNG=90000.0, ELSET=E0000001
```

2. Use the following entry to normalize the stresses and strains in the whole model. The stresses are normalized by 300.0 and the strains are normalized by 0.02.

```
*NORMALIZE,SIGMA=300.0, EPS=0.02
```

3. Use the following entries to normalize the stresses in the element sets E0000001, E0000002 and E0000003. The stresses are normalized by 300.0 in the element set E0000001, 400.0 in the element set E0000002 and 500.0 in the element set E0000003.

```
*NORMALIZE,SIGMA=300.0, ELSET=E0000001
*NORMALIZE,SIGMA=400.0, ELSET=E0000002
*NORMALIZE,SIGMA=500.0, ELSET=E0000003
```

3.8 *OUTPUT: Defining the output file name

The *OUTPUT command defines a FEPOST output filename. The default is the output filename from FECGS/ABAQUS without extension .fil.

The following parameteris required:

FILE	Set this parameter equal to the name of the FEPOST output file.

Examples:

1. The following entry defines a FEPOST output filename - blabla.

```
*OUTPUT,FILE=blabla
```

3.9 *PARAMETER: Generating a tabular list

The *PARAMETER is used to generate a table of variables for the time increments recorded in the FE analysis.
Following commands are suboption of *PARAMETER:

*PDIS
*PFORCE
*SIGav
*EPSav
*SUM EVOL
*Jn××_××
*TIME LOAD
*T-STRESS
*Q-STRESS
*VARIABLE (S11, S22,. . . , SDV1, . . .)
*V AVERAGE

Those command lines shall be entered after *PARAMETER.

3.9.1 *PDIS: Calculating displacement at a given point

Command *PDIS, suboption of *PARAMETER command, for evaluating the displacement at a defined degree of freedom for a given point.

The following parameters are required:

INPUT	Set INPUT=COORD (point for calculating the displacement is given by the coordinates), INPUT=NODE (point is defined as a finite element node)
NAME	Set this parameter equal to the name of the displacement, for example, NAME=CTOD.

Format **Entry**

If INPUT=COORD the following data lines are required:

I, 3F	Degree of freedom, X,Y,Z.

Repeat the above line as often as needed to give the more points.

If INPUT=NODE the following data lines are required:
DOFN, NODE Label
...

2I	Degree of freedom, Nodal label.

Repeat the above line as often as needed to give the more points.

Examples:

1. Use the following entries to calculate the nodal displacement U1 at the point (100.0, 100.0, 0.0) and U2 at the point (0.0, 100.0, 0.0).

```
*PDIS, INPUT=COORD, NAME=LOAD
1,100.0, 100.0, 0.0
2,0.0, 100.0, 0.0
```

2. Use the following entries to calculate the nodal displacement U1 for node 100, U2 for node 100 and U2 for node 200.

```
*PDIS, INPUT=NODE, NAME=LOAD
1,100
2,100
2,200
```

3.9.2 *PFORCE: Calculating reaction force

Command *PFORCE, suboption of *PARAMETER command, is used to sum the nodal reaction force at a given degree of freedom for a defined group of nodes.

The following parameters are required:

DOFN	Set this parameter equal to the degree of freedom.
NAME	Set this parameter equal to the name of the resulted reaction force to be given.
INPUT	Set this parameter equal to POLYGON or NODE for defining a node group.

Format **Entry**

The following data line is required:

If INPUT=POLYGON enter the following data line to define a group of nodes,

F	X1,Y1,Z1, X2, Y2, Z2. Defining the coordinates of minimum model space bounds (X1, Y1, Z1), and the coordinates of maximum model space bounds (X2, Y2, Z2). All the nodes within the defined region are included in the group.

If INPUT=NODE enter the following data line to define a group of nodes,

(n+1)I	n, Node 1, node 2, . . . , node n. n is the number of nodes to be included, node i is the nodal label.

Examples:

1. Use the following entries to calculate the nodal reaction force in the Y-direction resulted from the node group wthin the given polygon. The calculated reaction force is named LOAD.

```
*PFORCE, DOFN=2, INPUT=POLYGON, NAME=LOAD
0.0, 0.0, 0.0, 100, 100.0, 0.0
```

2. The following entries are used to sum the nodal reaction force in the Y-direction from 5 nodes (100, 200, 400, 600, 800). The calculated reaction force is named LOAD.

```
*PFORCE, DOFN=2, INPUT=NODE, NAME=LOAD
5,100, 200, 400, 600, 800
```

3.9.3 *SIGav: Calculating volume average stresses

Command *SIGav, a suboption of *PARAMETER command, is used for calculating the volume average stresses over a given material volume.

The following parameters are required:

VARIABLE	Set this parameter equal to the name of the variable (stress components) to be averaged. A list of available variables are S11, S22, S33, S12, S13, S23.
NAME	Set this parameter equal to the name of the calculated value.

Examples:

1. Use the following entries to calculate the average stresses of S11 and S22 over the entire model, the resulted values are named S11av and S22av.

```
*SIGav,VARIABLE=S11,NAME=S11av
*SIGav,VARIABLE=S22,NAME=S22av
```

3.9.4 *EPSav: Calculating volume average strain

Command *EPSav, a suboption of *PARAMETER command, is used for calculating the volume average strains over a given material volume.

The following parameters are required:

VARIABLE	Set this parameter equal to the name of the variable (strain) to be averaged. A list of available variables are E11, E22, E33, E12, E13, E23.
NAME	Set this parameter equal to the name of the calculated value.

Examples:

1. Use the following entries to calculate the average strains of E11 and E22 over the entire model.

```
*EPSav,VARIABLE=E11,NAME=E11av
*EPSav,VARIABLE=E22,NAME=E22av
```

3.9.5 *SUM EVOL: Calculating or summing the material volume

Command *SUM EVOL, a suboption of *PARAMETER command, is used for calculating or summing the material volume defined by element sets.

The following parameters are optional:

ELSET	Set this parameter equal to the name of the element set in which material volume is calculated. By default, FEPOST calculates the whole material volume.
DEFORM	Set this parameter equal to YES if material volume has to calculate in the deformed configuration.
RECORD	Set this parameter equal to YES if element volume has been recorded. Otherwise FEPOST will calculate element volume by itself.
NAME	Set this parameter equal to the name of the calculated value to be appear in the tabular list. The default is NAME=TEVOL.

Format **Entry**

The following data line is required if material volume include several element sets:

nA	ELSET1, ELSET2, ... ELSETn. Enter all the element set names. Each line can have maximum 8 element sets.

Repeat the above line as often as needed to enter more element sets.

Examples:

1. Use the following entry to calculate the material volume of the element set E1.

```
*SUM EVOL,ELSET=E1,NAME=EVOL1, DEFORM=YES
```

2. Use the following entry to calculate the whole material volume.

```
*SUM EVOL, RECORD=YES
```

3. Use the following entry to calculate the whole material volume.

```
*SUM EVOL,NAME=EVOL
```

4. Use the following entries to calculate the material volume consisting of the element set E1 and E2.

```
*SUM EVOL,NAME=EVOL12
E1, E2
```

3.9.6 *Jn××_××: Selecting the J-integral

Command *Jn××_××, suboption of *PARAMETER command, is used to select the J-integral at the given position and contour for a 3D fracture analysis and a given contour for 2D analysis.

The first two characters ×× are used to define the position of the J-integral to be selected in a 3D analysis, and the second ×× are used to define the contour number of the J-integral to be selected at that position.

For the 2D analysis the command is *Jn××, or *Jn01_××, and ×× indicates the contour number of J-integral to be selected.

Examples:

1. Use the following entries to select the J-integral in a 3D analysis.

```
*Jn01_10
*Jn05_10
```

1. Use the following entry to select the contour 10 J-integral in a 2D analysis.

```
*Jn10
```

3.9.7 *TIME LOAD: Convert time to related applied load or deformation

*TIME LOAD, suboption of *PARAMETER command, is used to convert the total time in a FE analysis to the corresponding applied force, displacement etc.

The following parameter isoptional:

NAME	Set this parameter equal to the name of the time load to be given.

Format **Entry**

The following data line is required:

A, I, F	”STEP” K AMAG. String ”STEP” is case sensitive, K is the STEP number, AMAG is the magnitude of applied value in the time load step K.

Repeat the above line as often as needed to give the more step.

Examples:

1. Use the following entries to convert the total time to the applied load.

```
*TIME LOAD, NAME=FORCE
STEP 1, 100.0
STEP 2, 400.0
STEP 3, 500.0
```

3.9.8 *T-STRESS: Evaluating the T-stress

*T-STRESS, suboption of *PARAMETER command, is used for evaluating the elastic T-stress. The details of the formulation for calculating the T-stress is given in section 5.1. This option works for both homogeneous material and bimaterial interfacial crack.

The following parameters are required:

RADIUS	Set this parameter equal to radius where T-stress is going to be calculated.
START ANGLE	Set this parameter equal to an angle in degree (0 - 360°).
END ANGLE	Set this parameter equal to an angle in degree (0 - 360°). Radius, start angle and end angle define a part of circle to evaluate the T-stress.
NUM	Number of points to be used to calculate the T-stress.
KI	Set this parameter equal to Jn××.

Format **Entry**

The following data line is required:

4F	E1, Po1, E2, Po2. Ei and Poi are a pair of Young's modulus and Poisson's ratio used for calculating the K from J-integral. For the homogeneous material, only first pair is needed, whereas, four numbers are need for a bimaterial interface crack.

Examples:

1. Use the following entry to conduct T-stress calculation along a given circle of radius 1.0 from 0° to 90°.

```
*T-STRESS, RADIUS=1.0, START ANGLE=0, END ANGLE=90
90000.0, 0.3, 90000.0, 0.3
```

3.9.9 *Q-STRESS: Evaluating the Q-stress at a given point

*Q-STRESS, suboption of *PARAMETER command, is used for evaluating the Q-stress at a given point defined by coordinates (X,Y, Z). **This option is currently not yet available**. The details of the formulation for calculating the Q-stress is given in section 5.2.

The following parameters are required:

COORD	Set this parameter equal to the normalized coordinates (X,Y,Z) where Q-stress to be calculated, for example, COORD=(0.0, 2.0, 0.0)
J NORMALIZE	Set this parameter equal to the selected J-integral for normalizing the coordinates, referring section 3.9.6 for the selection of J-integral.
YIELD	Set this parameter equal to the reference stress, σ_o, for normalizing the coordinates, J/σ_o.

Examples:

1. Use the following entry to conduct Q-stress calculation at a normalized point (2.0, 0.0, 0.0).

```
*Q-STRESS, COORD=(2.0, 0.0, 0.0), YIELD=300.0, J NOR=Jn10
```

3.9.10 *VARIABLE (S11, S22, SDV1 . . .): Calculating the solution variables

*VARIABLE, Suboption of *PARAMETER command, is used for evaluating the value of this variable at given point. The variable can be any of stress components, strain components and state solution variables from FECGS or ABAQUS. The available variables are listed in section 4.1. For example, at a given coordinates (X,Y,Z), or, a given gauss intergration point.

The following parameters are required:

INPUT	Set INPUT=COORD (point for calculating the variable is given by the coordinates), INPUT=GAUSS (point is defined as a element gauss point)
NAME	Set this parameter equal to the name of the quantity to be calculated.

Format **Entry**

If INPUT=COORD the following data line is required:

3F	X, Y, Z. The coordinates where VARIABLE is calculated.

Repeat the above line as often as needed to give the more points.

Format **Entry**

If INPUT=GAUSS the following data line is required:

2I	Enter element number, Gauss integration number

Repeat the above line as often as needed to give the more points.

Examples:

1. Use the following entries to calculate the stress component S11 for the points given by the coordinates (100.0, 100.0, 0.0) and (0.0, 100.0, 0.0).

```
*S11, INPUT=COORD, NAME=S11
100.0, 100.0, 0.0
```

```
0.0, 100.0, 0.0
```

2. Use the following entries to calculate the stress component S11 at gauss points 1 and 2 of element 100.

```
*S11, INPUT=GAUSS, NAME=S11
100,1
100,2
```

3.10 *PATH: Calculating the results along a given curve

The *PATH command is used to evaluate the distribution of the required variables along a given line or curve.

The following parameters are optional:

ELSET	Set this parameter equal to the element set name in which the distribution of given variables is calculated. The default is the whole model.
OUTPUT	Set this parameter equal to HISTORY, the distribution of a given variable along the given curve for the selected time increments is written in a result table. By default, FEPOST writes the results of the selected variables for each time increment selected.

The following suboption commands are required:

*VARIABLE
*SEGMENT

3.10.1 *VARIABLE: Selecting output variables

*VARIABLE suboption of *PATH command defines the solution variables, such as the stresses and strains, to be calculated.

Format	Entry

The following data line is required:

A	Give a list of variable names to be calculated. If parameter OUTPUT=HISTORY is given in *PATH, enter only one variable name.

Examples:

1. Use the following entries to define a list of variables.

```
*VARIABLE
S11,S22,S12, E11, E22, E12, MISES
```

3.10.2 *SEGMENT: Defining a curve

Command *SEGMENT, suboption of *PATH command, defines the curves in which the distribution of the listed variables is calculated.

The following parameters are optional:

NUM	Number of segments to be divided. By default, NUM=20.
TYPE	Set this parameter equal to curve type. TYPE=LINE(default) defining a line between two given points, TYPE=ARC defining an elliptical curve, TYPE=NODE defining a list of nodes.
SCALE	Set this parameter equal to a real number, q, to define a power law bias, by default, q=1.0. The segment lengths are scaled by $1, q, q^2, \ldots$.
distribution	

Format **Entry**

The following data line is required if TYPE=LINE:

6F	X1, Y1, Z1, X2, Y2, Z2. (X1, Y1, Z1) define point 1 and (X2, Y2, Z2) define point 2.

The following data lines are required if TYPE=ARC:

7F	X0, Y0, Z0, A0, C0, T1, T2. Among those, (X0, Y0, Z0) defines center of ellipse, A0 and C0 define the two semi-axes of ellipse along X- and Y-axes in the local coordinate system, respectively, T1 and T2 are angles in degree (0° - 360°) of counterclockwise to define a part of ellipse.
9F	n11, n12, n13, n21, n22, n23, n31, n32, n33. Defining the local coordinate system.

The following data line is required if TYPE=NODE:

(n+1)I	n, node 1, node 2, ..., node n. n is the total number of nodes, node i is the nodal label.

Examples:

1. Use the following entries to define a line from point (X1, Y1, Z1) to (X2, Y2, Z2) along which the distribution of selected variables is calculated.

```
* *SEGMENT, NUMBER=40, SCALE=1.2, TYPE=LINE
```

```
0.0, 0.0, 0.0, 10.0, 0.0, 0.0
```

2. Use the following entries to define a elliptic curve centered at point (0.0, 0.0, 0.0), the two semi-axes of ellipse are 15.0 and 10.0, respectively.

```
*SEGMENT, NUMBER=40, SCALE=1.2, TYPE=ARC
0.0, 0.0, 0.0, 15.0, 10.0, 0.0, 90.0
1.0, 0.0, 0.0, 0.0, 1.0, 0.0, 0.0, 0.0, 1.0
```

3. Use the following entries to define a line formed by the nodes 1, 2, 3, 4, 5, 6, 7, 8, 9, 10.

```
*SEGMENT, TYPE=NODE
10, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10
```

3.10.3 *NORMALIZE: Normalizing the coordinates

*NORMALIZE is Suboption of *PATH command for normalizing the coordinates.

The following parameters are required:

TYPE	Set TYPE=CONSTANT - the coordinates are normalized by a constant, TYPE=Jn×× - the coordinates are normalized by Jn××/σ_o,TYPE=Jav - the coordinates are normalized by average J Jav/σ_o, TYPE=Knii - the coordinates are normalized by (Kn××/σ_o)^2, referring to the section 3.9.6 for the definition of Jn××.

Format **Entry**

If INPUT=CONSTANT the following data line is required:

F	Constant.

Examples:

1. Use the following entry to normalize the coordinates by Jn11/σ_o.

```
*NORMALIZE, TYPE=Jn11
```

2. Use the following entry to normalize the coordinates by a constant 100.0.

```
*NORMALIZE, TYPE=CONSTANT
100.0
```

3.10.4 *HISTORY: Selecting time increments

Command *HISTORY, suboption of *PATH, is used to specify a time step to be plotted. By default, all the time increment recorded are plotted.

Format	**Entry**

The following data lines are needed if *HISTORY command is used:

A, 2I	”STEP” K, INC. String ”STEP” is case sensitive, K is the step number, INC is the increment number. IF INC=0, all the increments in the step K recorded are included. IF K=0, all the steps recorded are selected.

Repeat the above line as often as needed to give the more time increments.

Examples:

1. Use the following entries to select the corresponding time increments.

```
*HISTORY
STEP 1, 5
STEP 2, 50
STEP 3, 100
```

3.11 *PLOT MESH: Drawing the FE mesh

The *PLOT MESH command is used to draw the finite element mesh. In the current version, this option is only available for drawing 2D meshes.

The following parameter is required:

OUTPUT	Set this parameter equal to MPL for generating the neutral plot file

The following options may be needed if one wants to draw the different element sets in different colors.

*ELSET

Format **Entry**

The following data line is needed if *ELSET option is required:

A	The element set names to be drawn, the different element sets are separated by comma (,) or blanks.

The following option and data lines are need if one wants to draw the different element sets in different colors.

*COLOR TABLE

Format **Entry**

The following data line is need if *COLOR TABLE option is required:

(n+1)I	n, color 1, color 2, . . . , color n.

Examples:

1. Use the following entry to draw the FE mesh ofthe whole model.

```
*PLOT MESH, OUTPUT=MPL
```

2. Use the following entries to draw the FE mesh from the element sets: E0000001, E0000002, E0000003.

```
*PLOT MESH, OUTPUT=MPL
*ELSET
E0000001, E0000002, E0000003
*COLOR TABLE
3, 3, 8, 16
```

3.12 *Q-STRESS: Evaluating the Q-stress for a given curve

*Q-STRESS command is used for evaluating the Q-stress distribution along a given line or curve. This option is currently not yet available. Section 5.2 gives the details of the formulation for calculating the Q-stress.

3.13 *W-STRESS: Calculating the volume integration

*W-STRESS command is used to calculate the volume integration for the cleavage failure probability using weakest link statistics and three-parameters Weibull distribution model. This option is currently not yet available.

Chapter 4

LIST OF VARIABLES AND ELEMENT TYPES

4.1 Variable identifiers

The following list of nodal variables are available from the output files:

Ui	i-component of displacement (i $\leq$ 3)
RFi	i-component of nodal reaction force (i $\leq$ 3).

The following list of element variables are available from the output files:

Sij	ij-component of stress (i, j $\leq$ 3)
Eij	ij-component of strain (i, j $\leq$ 3)
PEij	ij-component of plastic strain (i, j $\leq$ 3)
MISES	Mises equivalent stress
PRESS	Hydrostatic presure stress, defined as -$\frac{1}{3}trac\sigma$ or -$\frac{1}{3}\sigma_{kk}$
SPi	ith principal stress, SP1 $\geq$ SP2 $\geq$ SP3
SDVi	ith state variable component
PEEQ	Equivalent plastic strain

The following list of the element variables have been defined by FEPOST:

H	Triaxiality parameter, defined as σ_H/σ_e and $\sigma_H = \frac{1}{3}trac(\sigma)$

4.2 Element lists

The following list of element types are available from the output files:

Plane strain elements:

CPE3, CPE4, CPE4R, CPE6, CPE8, CPE8R, CPE9, CPE9R, the last two elements are FECGS elements.

Plane stress elements:

CPS3, CPS4, CPS4R, CPS6, CPS8, CPS8R, CPS9, CPS9R, the last two elements are FECGS elements.

Axisymmetric elements:

CAX3, CAX4, CAX4R, CAX6, CAX8, CAX8R, CAX9, CAX9R, the last two elements are FECGS elements.

3D solid elements:

C3D4, C3D6, C3D8, C3D8R, C3D12, C3D15, C3D20, C3D20R, C3D27, C3D27R.

Chapter 5

QUANTITY DEFINITIONS

5.1 The T-stress

The definition of the T-stress can be written:

$$T = \sigma_{xx}^{spec} - \frac{K_I}{\sqrt{2\pi r}} f_{xx}(\theta). \tag{5.1}$$

where σ_{xx}^{spec} is the x-direction normal stress in a near crack tip region of an actual specimen and loading. The second term on the right hand side of Eq. (5.1) is the first singular stress term in the Williams eigen-expansion.

In FEPOST, the T-stress is calculated as follows:

$$T = \frac{1}{n} \sum_{i=1}^{i=n} T^i = \sum_{i=1}^{i=n} \left[{}^i\sigma_{xx}^{spec}(r^i, \theta = \pi) - \frac{K_I}{\sqrt{2\pi r^i}} f_{xx}(\theta^i) \right]. \tag{5.2}$$

Alternatively, T^i can also be obtained by,

$$T^i = [{}^i\sigma_{xx}^{spec} - {}^i\sigma_{yy}^{spec}] - \frac{K_I}{\sqrt{2\pi r^i}} [f_{xx}(\theta^i) - f_{yy}(\theta^i)] \tag{5.3}$$

5.2 The Q-stress

The definition of the Q-stress is given as follows:

$$Q = \frac{\sigma_{yy}^{spec} - \sigma_{yy}^{ref}}{\sigma_o}. \tag{5.4}$$

5.3 The Weakest link model

Here, only the formulations used are listed, the detailed explanation and interpretation are not documented. The weakest link statistics adopted is:

$$\delta\phi = 1 - \exp\left[-\delta V \int^{\sigma} g(S)dS\right]. \tag{5.5}$$

The total failure probability is thus,

$$\phi = 1 - \exp\left(\int^{V}\left[-\delta V \int^{\sigma} g(S)dS\right]\right). \tag{5.6}$$

Using three-parameter Weibull distribution,

$$\int_{S_u}^{\sigma} g(S)dS = \left(\frac{\sigma - S_u}{S_o}\right)^m fN, \tag{5.7}$$

then one has,

$$\phi = 1 - \exp\left[\int^{V}\left(\frac{\sigma - S_u}{S_o}\right)^m fNdV\right], \tag{5.8}$$

or,

$$ln(1-\phi) = -fN\left(\frac{\sigma_o}{S_o}\right)^m \int^{V}\left(\frac{\sigma - S_u}{\sigma_o}\right)^m dV. \tag{5.9}$$

FEPOST calculates the volume integration shown in the right hand side of the above equations (5.8) and (5.9).