4.1 Input and Output Files

The input and output files required for the stand-alone version of **MAC/GMC** and **FEAMAC** are specified in different ways. The differences between these two methods of running **MAC/GMC** are detailed below.

4.1.1 MAC stand-alone

Since **MAC/GMC** reads from standard in and writes to standard out, the name of the files used as input and output are left to the user's discretion, i.e., will have no impact on the performance of the **MAC/GMC** code. As will be explained in the following sections, the user will also need to specify the names of the **PATRAN** output files and the files containing the X-Y data for plotting.

Note that a "debug" file may be generated depending on the "PRINT" level the user specifies in the input file (see **4.2.2**). This file will have the name "mac_debug".

Input	**Output**
infilename	*outfilename* debug file (mac_debug) **PATRAN** files (optional, see section **4.2.9**) plot data files (see section **4.2.16 and 4.2.17**) damage files (optional, see section **4.2.6**) yield surface files (optional, see section **4.2.7**)

4.1.2 FEAMAC

Almost all of the capabilities of **MAC/GMC** are available through the user defined material facilities of the **ABAQUS** finite element code. Access to **MAC/GMC** from within **ABAQUS** is outlined here. Currently, **MAC/GMC,** can only be access using 3D elements; however, **FEAMAC** has only been specifically tested using the C3D8 element within **ABAQUS.** Note any options not available or modified for finite element implementation are denoted subsequently by the following notation, for example:

☞ **FEAMAC Note:** *Not required by FEAMAC*

An example containing an **ABAQUS** input deck, required FORTRAN subroutines, and **FEAMAC** input files are given in **Example O**.

ABAQUS Input Deck:

To utilize **FEAMAC**, the **ABAQUS** input deck must contain the "*USER

SUBROUTINE", "*****USER DEFINED FIELD","*****INITIAL CONDITIONS, TYPE=FIELD" ,"*****EXPANSION" (if conducting nonisothermal analysis) and "*****USER MATERIAL" card. Where the USER SUBROUTINE card identifies the name of the file containing the FORTRAN subroutines required to access **MAC/GMC** and the "USER MATERIAL" card instructs **ABAQUS** to obtain the materials stress/strain behavior from the supplied subroutines. The name of the user material (defined in the **ABAQUS** input deck), converted to upper case, will be appended with the file extension that has been defined in the user supplied FORTRAN routines, consequently this name must be used as the **MAC/GMC** input filename for the given material. Note: the material name is limited by **ABAQUS** to be 8 characters or less. See the **ABAQUS** Users' Manual for a complete descriptions of these cards

ABAQUS FORTRAN Subroutine Files:

The name of this file is to be provided on the "*USER SUBROUTINE" card inside the **ABAQUS** input deck. These subroutines will be executed for every integration point of every element in the finite element model associated with a given material group. The required subroutine are listed in **Example O**. Only a portion of the FEAMAC_INIT routine should be edited by the user. This portion is bounded by the comments;

```
C *** BEGIN USER EDITS ***
```
and
```
C *** END OF USER EDITS ***
```

The seven variables within this section that can be edited are as follows:

PATH:	CHARACTER*80 Path to working directory
EXTENSION:	CHARACTER*80 File name extension to be appended to material name to form the name of the **MAC/GMC** input deck.
DFNAME:	CHARATER*80 Name of diagnostic file.
NPEL:	INTEGER Number of integration points whose data is to be plotted.
N_PEN(N):	INTEGER ARRAY (N:1->NPEL) Element numbers whose data is to be plotted.
N_PIN(N):	INTEGER ARRAY (N:1->NPEL) Integration numbers whose data is to be plotted.

☞ **Note**: N_PEN() and N_PIN() pairs define locations of data extraction to be utilized with the *MACRO or *MICRO **MAC/GMC** options

AB_PRINT: INTEGER
Diagnostic print level for **FEAMAC**.

0 - NONE
1 - UMAT: STRESS, STRAIN, INCREMENTAL AND EXECUTION TRACING INFORMATION
3 - FEAMAC_PRE: STRESS, STRAIN, INCREMENTAL AND EXECUTION TRACING INFORMATION
4 - TEMPERATURE INFORMATION
5 - STATE INFORMATION BEFORE AND AFTER CALL TO FEAMAC
25 - CONSTITUTIVE MATERIAL PROPERITES

☞ **Note**: Higher values of AB_PRINT provide all information provided by lower values plus additional information as described above.

FEAMAC Input Deck:

The **FEAMAC** input deck can be identical to the stand-alone **MAC/GMC** input deck as discussed in section **4.2**. The associated file name is obtained by converting the material property name, defined within the **ABAQUS** input deck, to uppercase and appending the file extension specified in the FEAMAC_INIT subroutine described previously. Since the load history definition and time integration are dictated by the associated **ABAQUS,** input deck, information in the **MAC/GMC** deck that relates to these issues will be ignored. All entries in section **4.2** that describe **MAC/GMC** options should be considered to be required by **FEAMAC** unless denoted otherwise.

☞ **Note:** Multiple **FEAMAC** input decks can be specific per **ABAQUS** run, so that multiple material systems can be analyzed within a given structure. Utilization of this option will allow the most efficient execution of large problems. For example, if one knows that only a small portion of the problem will be inelastic then define the zone of elements that may go inelastic by a given material group and the remainder by another. In this way maximum speed can be achieved, as a purely isothermal, elastic, problem with no micro fields being printed executes the fastest.

FEAMAC Output Files:

Most diagnostic messages from **FEAMAC** appear in the **ABAQUS** msg file. Files containing X-Y plotting information are generated for each N_PEN/N_PIN pair. The names for these files correspond to those defined in the **MAC/GMC** deck except that the N_PEN and N_PIN number are append to the name.

Example:
If in the FEAMAC_INIT subroutine the user specified;

```
NPEL = 1
N_PEN(1) = 18
N_PIN(1) = 7
```

and in the **FEAMAC** deck one specified;

```
*CURVE
  NP=2 %
*MACRO
  NT=1
  NC=1 X=1 Y=7 NAM=plot_file_sve %
```

FEAMAC would provide the stress and strain data in the 11-direction for integration point 7 of element number 18 in a file named

"PLOT_FILE_SVE.18.7.feamacro.data".

☞ **Note:** see section **4.2.17** regarding curve data input

.