



## Calculation of principal stresses and strains and their directions: FPRIN

This example illustrates the use of a Fortran program to read stress and strain records from an Abaqus results file and to calculate principal stress and strain values and their directions.

This page discusses:

- [General description](#)
- [Programming details](#)
- [Program compilation and linking](#)
- [Program execution](#)
- [Results and discussion](#)
- [Input files](#)

**Products:** Abaqus/Standard

### General description

This program shows how to retrieve integration point and nodal averaged stress and strain components from an Abaqus results file and then compute principal values and directions using the Abaqus utility routine `SPRIND`. Usage of this utility routine is documented in the input file provided for this problem, and further details about the interface to this subroutine are discussed in [Obtaining Stress Invariants, Principal Stress/Strain Values and Directions, and Rotating Tensors in an Abaqus/Standard Analysis](#). The results file created by the `FJOIN` program in [Joining data from multiple results files and converting file format: FJOIN](#) is used here to verify that the records that have been put together are retrievable. The previously generated results file was named `fjoinxxx.fin`. To use it as an input file for postprocessing program `FPRIN`, the file extension must be changed. This program will assume that the results file has the default `.fil` extension, which corresponds to Fortran unit 8.

### Programming details

The user should first review the general discussion on programming concepts and Abaqus Fortran interfaces in [About user postprocessing of Abaqus results files](#) and the detailed discussion of postprocessing given in [File Output Format](#).

When running program `FPRIN` (this program is named `fprin.f` on the Abaqus release media), the user will be prompted for the file name that initializes `FNAME`. Other variables, such as `LOUTF`, `NRU`, `LRUNIT(1,NRU)`, and `LRUNIT(2,NRU)`, are initialized inside the program. `INITPF` and `DBRNU` are then called to complete the necessary initializations and file connections. Data processing starts with a double `DO`-loop over all the records to be read, one-by-one, via a call to `DBFILE`. Each record is identified by its record key, which is stored in the second entry of the record. When records 1922

and 2000 are processed by program `FPRIN`, the heading and the current step and increment numbers are written out so as to provide a way to recognize the beginning of data in each analysis. Record type 1 is then examined to determine the output location of stress and strain, the number of direct and shear stress and strain components, and either the element number or the node number for which the records are written. The stress and strain records (11 and 21, respectively) will be filtered out for processing by the Abaqus utility routine `SPRIND`. When a stress or strain record is passed into `SPRIND`, principal stresses or strains and the corresponding principal directions are calculated and returned in an unsorted order.

### Program compilation and linking

Before program execution, the Fortran program has to be compiled and linked. Both operations, as well as the inclusion of the `aba_param.inc` file, are performed by a single execution of the **abaqus make** utility:

```
abaqus make job=fprin
```

This may have to be repeated until all Fortran errors are corrected. After successful compilation, the program's object code is automatically linked with the Abaqus object codes stored in the shared program library and interface library in order to build the executable program. Refer to [System customization parameters](#) to see which compile and link commands are used for a particular computer.

### Program execution

Before the program is executed, a results file must have been created. In this example the results file `fjoinxxx.fin` created by the `FJOIN` program discussed in [Joining data from multiple results files and converting file format: FJOIN](#) is used. This file must be renamed to `fjoinxxx.fil` since Fortran unit 8 (which is associated with the `.fil` file extension) is used in the program to read the file. When the program is executed using the command `abaqus fprin`, the prompt

```
Enter the name of the input file (w/o .fil):
```

will appear. Enter `fjoinxxx` to define `FNAME`. The program processes the data and produces a file named `pvalue.dat`, which contains information about principal stresses and strains and their directions.

### Results and discussion

The computed principal stress and strain values and their directions are tabulated below.

Analysis File	Principal Component	Stress × 10 <sup>5</sup>	Strain × 10 <sup>-3</sup>	Dir-1	Dir-2	Dir-3
<a href="#">fjoin002.inp</a>	1	10.714	25.0	1.0	0.0	0.0
	2	10.714	25.0	0.0	1.0	0.0

Analysis File	Principal Component	Stress × 10 <sup>5</sup>	Strain × 10 <sup>-3</sup>	Dir-1	Dir-2	Dir-3
	3	0.0	0.0	0.0	0.0	1.0
<a href="#">fjoin003.inp</a>	1	-2.8846	-12.5	0.707	-0.707	0.0
	2	2.8846	12.5	0.707	0.707	0.0
	3	0.0	0.0	0.0	0.0	1.0
<a href="#">fjoin004.inp</a>	1	0.0	-7.5	1.0	0.0	0.0
	2	7.5	25.0	0.0	1.0	0.0
	3	0.0	0.0	0.0	0.0	1.0

Input files

[fprin.f](#)

Postprocessing program.