



## Creation of a data file to facilitate the postprocessing of elbow element results: FELBOW

This example illustrates the use of a Fortran program to read selected element integration point records from an Abaqus results file to facilitate the postprocessing of elbow element results.  $X$ - $Y$  data are created that are suitable for use with the  $X$ - $Y$  plotting capability in Abaqus/CAE.

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 data for elbow elements from an Abaqus results file to visualize one of the following:

1. Variation of a variable along a line of elbow elements,
2. Variation of a variable around the circumference of a given elbow element, or
3. Ovalization of a given elbow element.

An ASCII file containing  $X$ - $Y$  data is created that can be read into Abaqus/CAE for visualization purposes.

To execute option 1, the elbow elements must be numbered such that they increase monotonically within the range of elements considered; all elements in the desired range must be elbow elements.  $X$ - $Y$  data will be created with the  $X$ -data being the distance along the line of elbow elements, measured along the elbow centerline and the  $Y$ -data being the variable value. The user must ensure that the integration point coordinates (COORD) are written to the results file if either option 2 or 3 is needed. For option 2  $X$ -data are the distance around the circumference of the elbow element, measured along the middle surface, and  $Y$ -data are the variable value. For option 3 the  $X$ - $Y$  data are the current coordinates of the middle-surface integration points around the circumference of the elbow element, projected to a local coordinate system in the plane of the deformed cross-section. The origin of the local system coincides with the center of the cross-section; the plane of the deformed cross-section is defined as the plane that contains the center of the cross-section and integration points 1 and 2.

## Programming details

The user is prompted for the name of the results file (assumed to be binary) and the postprocessing option (1, 2, or 3). The user is then prompted for additional information depending on the option that was chosen; this information includes

- The range of element numbers (options 2 and 3 require only a single element number),
- The section point number (options 1 and 2 only),
- The integration point number (option 1 only),
- The element variable (options 1 and 2 only),
- The component of the variable (as defined in [Results File](#), options 1 and 2 only),
- The step number, and
- The increment number.

The data are processed in a double DO-loop over all records, via a call to `DBFILE`. The desired data are stored in variable `VAR`; the integration point coordinates are stored in `COORDS`. The program checks to make sure the requested data are available in the results file. An error is issued if the user tries to process data that are not found in the results file.

## Program compilation and linking

Before program execution, compile and link the Fortran program by using the **abaqus make** utility:

```
abaqus make job=felbow
```

Repeat this command until all Fortran errors are corrected. After successful compilation, the program's object code is linked automatically with the Abaqus object codes stored in the shared program library and interface library 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 executing the program, run an analysis that creates a results file containing the appropriate output. This analysis includes, for example, output for the elements in a given range and the integration point coordinates of the elements. When the program is executed using the command `abaqus felbow`, the prompt

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

will appear. Enter the name of the results file to define `FNAME`. The user is then prompted for other information, such as the desired postprocessing option, element number, etc. The program processes the data and produces a file named `output.dat` that contains the information required to visualize the elbow element results.

## Results and discussion

[Elastic-plastic collapse of a thin-walled elbow under in-plane bending and internal pressure](#) contains several figures created with the aid of this program. The output agrees with the expected results.

## Input files

### [felbow.f](#)

Postprocessing program.