Instructions for Implementing the PPR Cohesive Model in ABAQUS

Description

The following presents detailed instructions for implementing the PPR potential-based cohesive model into a finite element mesh and running the analysis in ABAQUS via a user-defined element (UEL).

Download the Fortran UEL

In order for a UEL to be imported into ABAQUS, it must be written in a separate Fortran file. The PPR Fortran file is available for download here. A flowchart depicting how the UEL is called in ABAQUS is shown to the right. Comments within the UEL reference equations given in the Educational Paper, which gives more details on their derivations. In addition, a list of the nomenclature that is used within the code is provided in order to help alleviate any confusions bewteen the code and the paper.

Create Input File and Run Abaqus

Before a problem can be run in ABAQUS, a few changes must be made to the input file to incorporate a UEL. First, the cohesive elements must be declared using the `*USER ELEMENT` command, e.g.

```
*USER ELEMENT, TYPE=U1, NODES=4, COORDINATES=2, PROPERTIES=9, VARIABLES=4
1, 2
```

The `TYPE` indicates the name of the element type, `NODES` is the number of nodes, `COORDINATES` is the largest active degree of freedom, `PROPERTIES` is the number of the input parameters (for the PPR choesive model, there are nine), and `VARIABLES` is the number of solution dependent variables. The next line lists the active degrees of freedom, i.e. `1, 2`, which corresponds to the horizontal and vertical displacements.

Next, the cohesive element connectivities must be defined, using the typical `*ELEMENT` command. The `TYPE` used is the same as the one given when the UEL was defined (e.g. `U1`). The element connectivities are then listed in the standard fashion:

```
*ELEMENT, TYPE=U1, ELSET=COH_ELE
101, 1, 2, 3, 4
    ⋮
    ⋮
```

Here, `ELSET` is the name of the element set to which these elements are assigned, which is this case is `COH_ELE`.

After the element connectivities are listed, the input parameters must be defined for the given element set, which are defined as follows,

```
*UEL PROPERTY, ELSET=COH_ELE
100, 200, 4e6, 3e6, 5, 1.6, 0.005, 0.005,
0.01
```

The nine parameters must be listed in the following order: ϕ_n, ϕ_t, σ_{max}, τ_{max}, α, β, λ_n, λ_t, and the thickness along the out of plane-direction. The rest of the input file remains unchanged.

After moving the input file, and PPR UEL into a directory, one can execute an ABAQUS analysis in conjunction with the UEL subroutine through the following command, i.e.

```
abaqus job=input_file_name user=UEL_file_name
```

If using the input file provided, the analysis may take close to an hour. After the analysis has been completed, a series of additional files will have been created in your directory; including, but not limited to:

- *input_file_name.dat*: contains printed error messages, results and other comments
- *input_file_name.log*: contains a very short summary (20 lines) of the starting and stopping steps in the analysis.

The key results can be found in the ".dat" file. In order to produce plots from the results file one can extract the data using a variety of means. If using the files provided, for this specific problem, a Matlab parser has been created: simply called "parser.m".

To use the parser the user need only change the name of the input ".dat" file at the beginning of the code.