7.4.1 Test case 5. Hydrostatic tension test and Tension test on an 8 node brick element

Aim: Evaluate the behaviour of the implemented material law in a test case documented in the literature, like the case of of the Hydrostatic tension in a brick element. In this case a tension is applied to 3 faces of the element while the other ones have certain restrictions.

Expected result: represent the solution given from the result of equations that describe the specific problem presented in [Aravas, 1987] and [Aru, 2015]

Command used to run the program In main folder you find the folder 3_UMAT that contains the following files, one is the the GTN subroutine and the other is a UMAT for the Rousselier model taken from the Thesis of [Aru, 2015]. This second UMAT is a model that has certain coincidence with the GTN damage model and its implementation also followed the article of [Aravas, 1987].

```
UMAT_GTN_W.f
UMAT_Rousse.f
```

In main folder you find the folder 2_Test_Abaqus_input_files\Inp_Rousselier that contains the following files

```
testR_te_vol.inp
testR_te_vol2.inp
testR_ht_vol.inp
```

In the folder 2_Test_Abaqus_input_files\Inp_GTN that contains the following file

```
test_ht_vol.inp
```

To run the tests, copy the files to a location where you can use Abaqus configurated for use of subroutines. In the terminal, in the case of the abaqus configurate in the cae-pool TUBAF, use the following lines

```
abq2020 interactive job=testR_te_vol.inp user=UMAT_Rousse.f verbose=2
```

Exit from the execution mode and run the next test, follow this procedure until run all the test

```
abq2020 interactive job=testR_te_vol2.inp user=UMAT_Rousse.f verbose=2

abq2020 interactive job=testR_ht_vol.inp user=UMAT_Rousse.f verbose=2

abq2020 interactive job=test_ht_vol.inp user=UMAT_GTN_W.f verbose=2
```

In the folder where was executed the Abaqus for each test case should be generated the a file with the extension .feh with the same name of the input files used. Copy these files to the folder 2_Test_Abaqus_input_files\Results_abaqus.

Finally in the folder 2_Test_Abaqus_input_files is the file

```
fi_R_testR_ht_vol.py
```

Run this file to process the information and plot the result Finally in the folder 2_Test_Abaqus_input_files is the file fi_R_testR_ht_vol.py. The figures are generated in the folder 2_Test_Abaqus_input\Fig_test_ht_vol

```
$ python3 fi_R_testR_ht_vol.py
```

Obtained result When the GTN UMAT subroutine was used in abaqus, as expected from the 1D material point tension test, the result is that this only runs some steps, when enters to the plastic zone the criterions of convergence are not fulfilled, or is reported excessive distortion of certain nodes.

For the case of the Rousselier UMAT this run the tension test, figures 18 to 20, but does not run the hydrostatic test, the reference point are not showed since this correspond to the GTN model and do not agree with this model. These result are presented with the intention to show that it has been taken time during the project to work in the postprocessing of the result but since the GTN UMAT still has problems in the plastic zone it was not possible present its the results. In the figure 18 are plotted 2 tension test rotated 90 degrees showing that the behavior is isotropic.

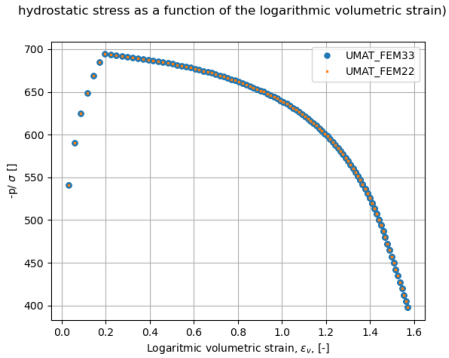

Figure 18: Stress in the tension direction as function of true strain in a Tension test in a C3D8 element, Rousselier UMAT

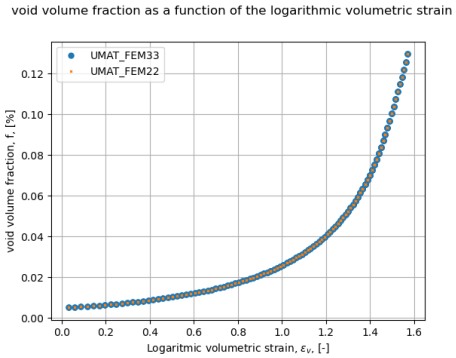

Figure 19: void volume fraction as function of true strain in a Tension test in a C3D8 element,, Rousselier UMAT

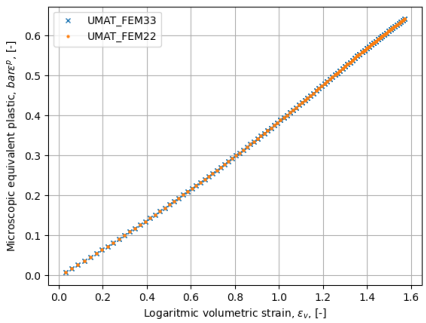

Figure 20: microscopic equivalent stress as function of true strain in a Tension test in a C3D8 element, Rousselier UMAT