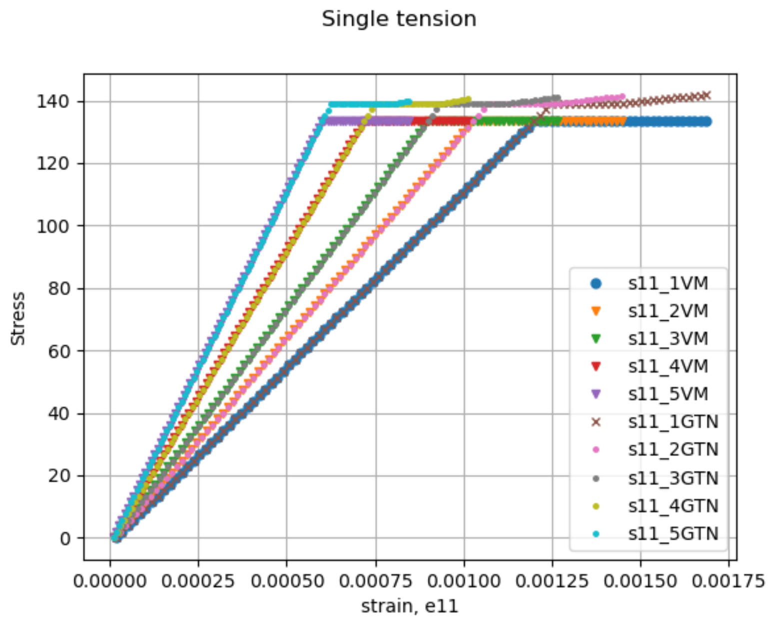

Figure 14: Comparation GTN and VM material laws in the elastic region changing the Young's modulus

7.3 Test case 3:material point Isochoric tension

Aim: In this test is evaluated the performance of algorithm at the material point level, especially the plastic return, when a controlled $\mathcal{E}_{11}$, $\mathcal{E}_{22}$, $\mathcal{E}_{33}$, strain is applied to the element.

Expected result: Find that the ISV variable, void volume fraction, microscopic plastic strain increases its values through the time (strain). Also observe that the code, especially in the plastic zone converges

Command used to run the program:For this test you find in the main folder the following files. The parameters used in this test are the kind of analysis performed 3D or plain stress, the parameter that change this option is set in each of the following files. File to run the test, these files are based on the notes of the course Plasticity WS2020 TUBAF

```
test_isochoric_tension3D.for
test_isochoric_tensionPS.for
```

In the folder 5_Isochoric_test/ you find the files to plot the results

```
fi_test_MatLaw3D.py
fi_test_MatLawPS.py
```

To run the test, compile the files in the main folder and run the generated executable file. It takes around 10-15 minutes to generate the results. The parameters for this test are the standard parameters as are indicated in the section and are set in the file to run the test.

```
$ gfortran tensor_ope_module.for material_law_GTN.for test_isochoric_tension3D.for -llapack
$ ./a.out
$ gfortran tensor_ope_module.for material_law_GTN.for test_isochoric_tensionPS.for -llapack
$ ./a.out
```

The csv files with the results are generated in the folder 5_Isochoric_test/. Now, change to that the folder and run the python files

```
$ cd 5_Isochoric_test
$ fi_test_MatLaw3D.py
$ fi_test_MatLawPS.py
```

The figures are generated in the folders 1_Fig_3D and 2_Fig_PS

Obtained result: In the first tests that where done to the implementation there was an instability that causes that when the stress entered the plastic range in the next step returned to the elastic zone and started again to the elastic zone doing again elastic increments. This problem was due to the fact that some of the derivatives used to find the correction to the volumetric plastic strain an deviatoric plastic strain increments are not continuous functions, therefore a wrong setting of the initial iteration point cause this problem.

In the figure 15 is presented the result until an strain $\mathcal{E}_{11}$ of 0.0016. The code is stable but the behavior finalized the elastic zone is not expected. The behavior of porosity, plastic train and microscopic equivalent plastic strain are increase as is expected, in the first steps of the implementation there where cases where one of these variables became negative, which is physically impossible.

In the figure 16 there is the response for the plain stress case in the elastic range, when the stress the plastic range there is an inestability.

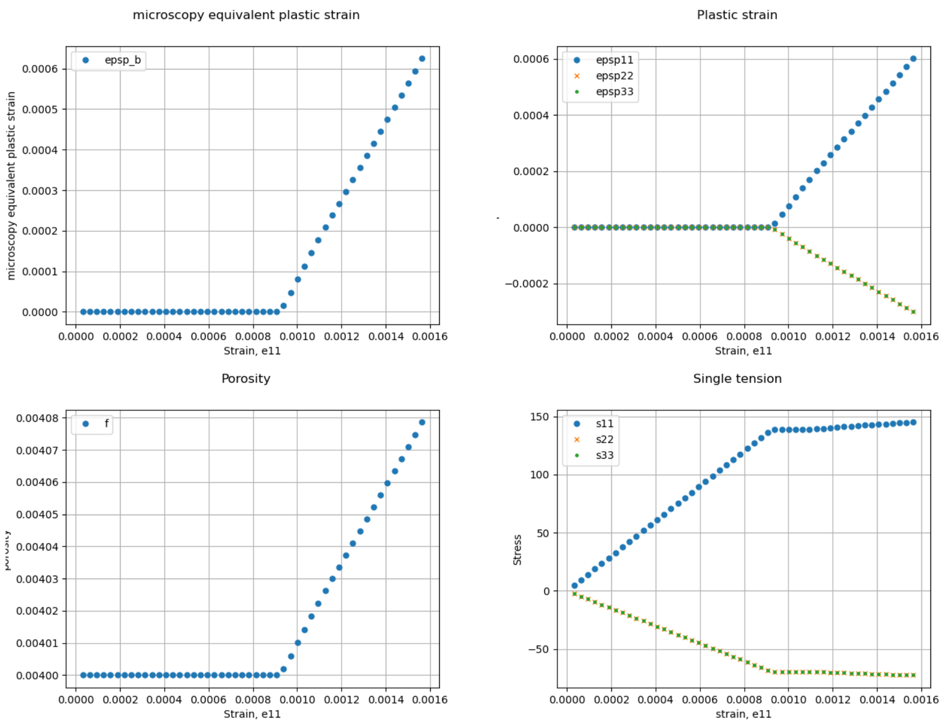

Figure 15: Comparation GTN and VM material laws in the elastic region changing the yield point

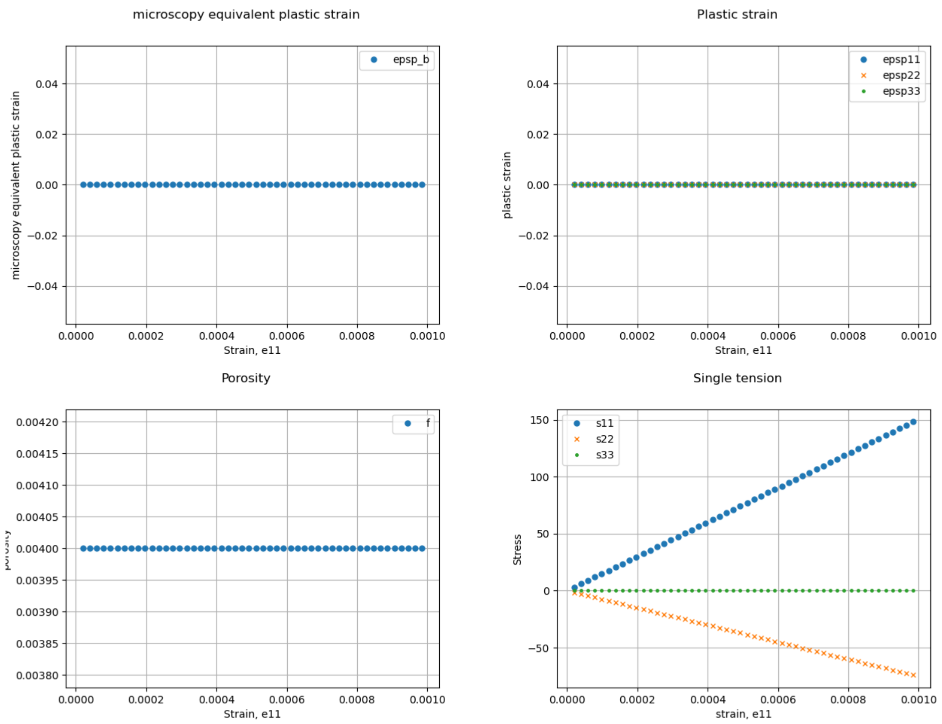

Figure 16: Comparation GTN and VM material laws in the elastic region changing the yield point

7.4 Test case 4:1D material point tension test

Aim: In this test is evaluated the performance of algorithm at the material pointlevel, especially the algortithmic tangent stiffness tensor. When a 1D strain controlled deforation is applied the non-axial component must be zero. In this test the ATS of the nonaxial components is used to to solve the strains.

Expected result: Find that the code is stable and that the algoritmic tangent stiffness drive to a solution of the strain.

Command used to run the program: In the main folder you find the file to run the test. The parameter in this test is the type of material law which is set in the following file as descried in the manula section the program was developed with the von Mises and the GTN options, the VM is only used for test cases.

In the main folder you find File to run the test, this file is based on the notes of the course Plasticity WS2020 TUBAF

```
test_1D_Strain_drive.for
```

Additionally in the folder 6_1D_Stra_dri/ you find the file to plot the result

```
fi_test_strain_drive.py
```

To run the test, compile the file in the main folder and run the generated executable file.

```
$ gfortran tensor_ope_module.for material_law_GTN.for test_elast_be.for  -llapack
$ ./a.out

$ gfortran tensor_ope_module.for material_law_GTN.for test_1D_Strain_drive.for -llapack
$ ./a.out
```