Validation case

This documentation presents a homogeneous test case which is used to validate the Direct FE^2 input files generated by the Python scripts presented in the repository. It includes the details of both the Direct FE^2 and reference DNS models as well as a comparison of their results.

Direct FE^2 model

The Direct FE^2 model is based on the 2D demonstration example, 'Demo_2D_Quad-Lin_Quad-Lin.zip', presented in the repository, with some modifications. On the microscale, both the matrix and inclusion phases of the RVE are assigned the material properties of epoxy to form a homogeneous RVE, as shown in Figure 1, while the mesh is preserved.

```
** Section: Solid_Epoxy
*Solid Section, elset=_PickedSet3, material=Epoxy
1.,
** Section: Solid_Carbon
*Solid Section, elset=_PickedSet4, material=Epoxy
1.,
```

Figure 1 Modifications made to the microscale input file

On the macroscale, the prescribed displacement imposed on the loaded end of the cantilever beam is extended to 30mm and the load step is broken down into increments of 0.1, to better capture the material nonlinearity of epoxy.

```
** STEP: Step-1
**
*Step, name=Step-1, nlgeom=NO
*Static
0.1, 1., 1e-05, 0.1
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Displacement/Rotation
*Boundary
_PickedSet4, 1, 1
_PickedSet4, 2, 2
** Name: BC-2 Type: Displacement/Rotation
*Boundary
_PickedSet5, 2, 2, 30.
```

Figure 2 Modifications made to the macroscale input file

The 2D Python script is then executed with these updated macroscale and RVE input files to generate the corresponding Direct FE^2 input file.

DNS model

The input file for the reference DNS model is a direct copy of the modified macroscale input file above. The macroscale mesh is then assigned the material properties of epoxy by directly editing the macroscale input file, as shown in Figure 3.

```
** Section: Solid_Placeholder
*Solid Section, elset=_PickedSet3, material=Epoxy
1.,
                              ⋮
** MATERIALS
**
*Material, name=Epoxy
*Elastic
3500., 0.34
*Plastic
 30.,     0.
 35., 0.0005
 40., 0.0015
 45.,  0.003
          ⋮
```

Figure 3 Modifications made to the macroscale input file to obtain the DNS input file

Results

The results from both models are then compared.

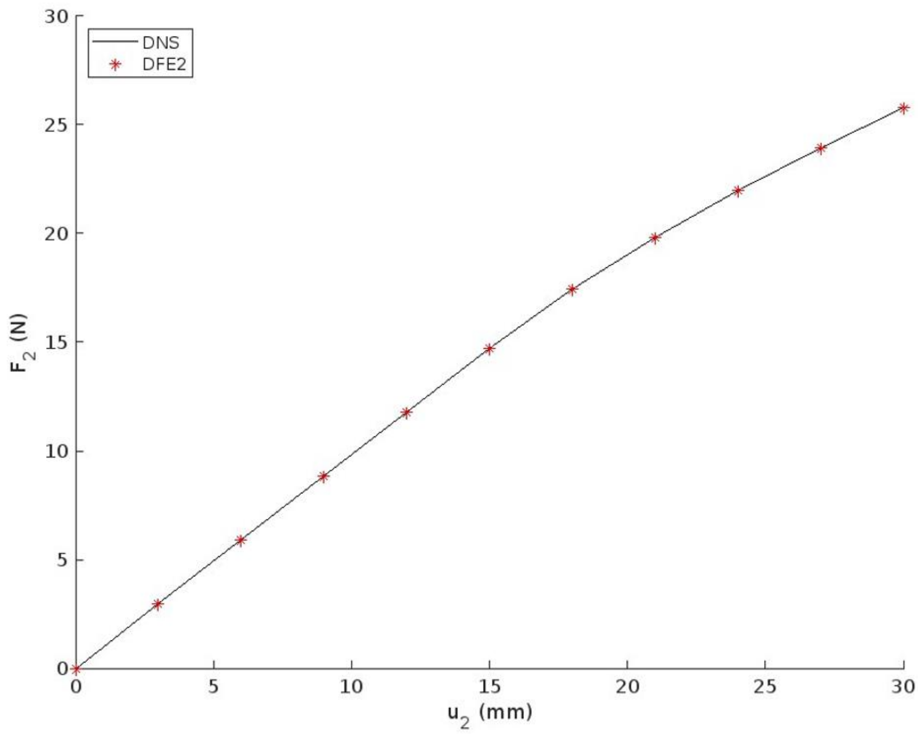

Figure 4 Force-displacement plots for the homogeneous cantilever beam from both models

The force-displacement plots obtained from both models overlap each other exactly, as seen in Figure 4. Furthermore, the deformed shape, axial strains as well as the equivalent plastic strain contours from both models match, as seen in Figure 5 and Figure 6. It is noted that the contour limits need to be adjusted during visualization such that they are consistent between the Direct FE^2 and DNS models for a better comparison.

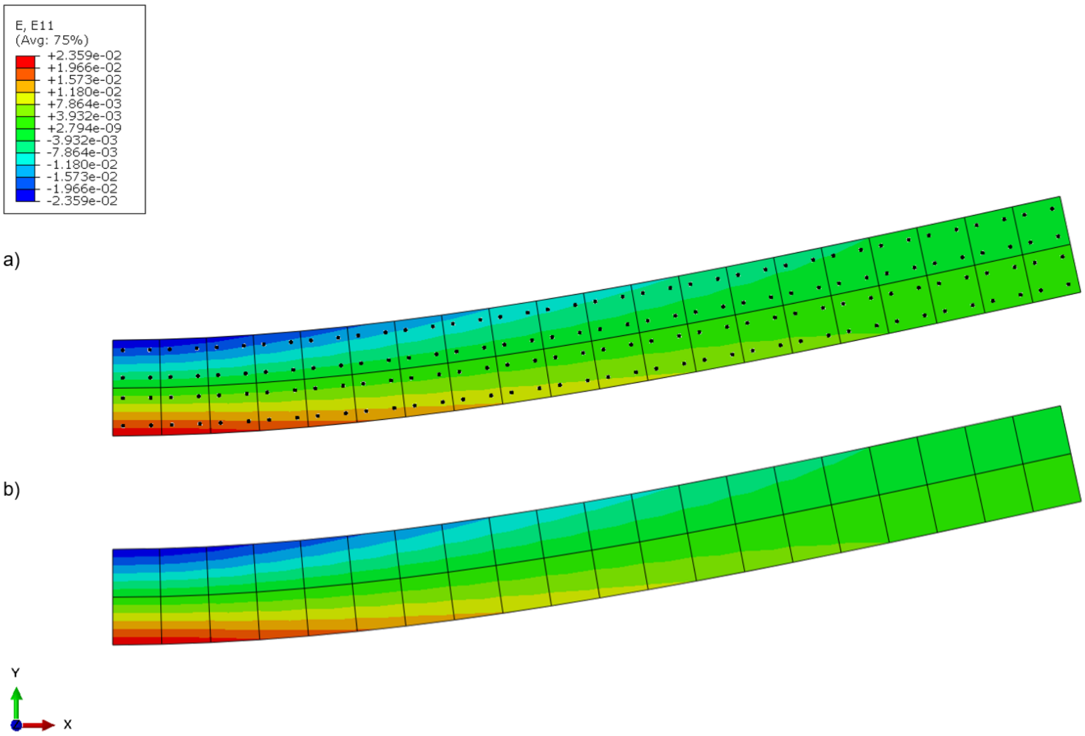

Figure 5 Axial strain contours for the homogeneous cantilever beam a) Direct FE^2 model; b) DNS model

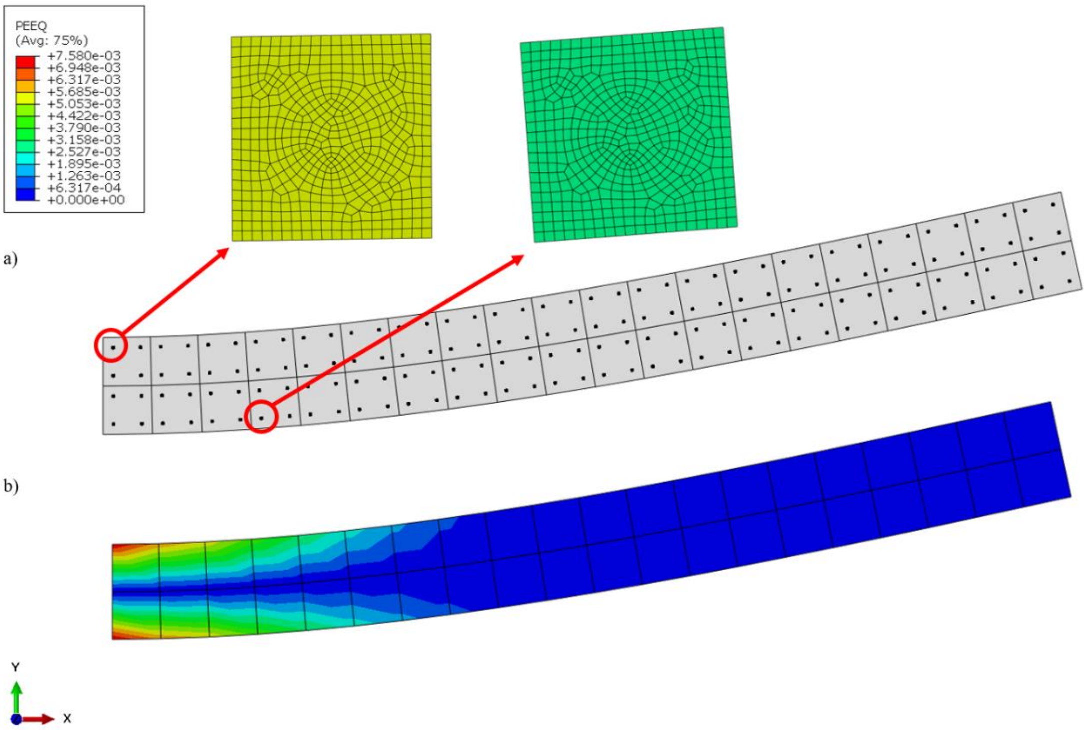

Figure 6 Equivalent plastic strain contours for the homogeneous cantilever beam a) Direct FE^2 model; b) DNS model

These results demonstrate that the Direct FE^2 model generated by the Python scripts is able to scale up the effective constitutive behaviour obtained from the RVE to the macroscale structure and obtain the same response as a typical FE model. As such, the script has been validated to correctly generate a Direct FE^2 multiscale model.