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.

```
1051    ** Section: Solid_Epoxy
1052    *Solid Section, elset=_PickedSet3, material=Epoxy
1053    1.,
1054    ** Section: Solid_Carbon
1055    *Solid Section, elset=_PickedSet4, material=Epoxy
1056    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.

```
148    ** STEP: Step-1
149    **
150    *Step, name=Step-1, nlgeom=NO
151    *Static
152    0.1, 1., 1e-05, 0.1
153    **
154    ** BOUNDARY CONDITIONS
155    **
156    ** Name: BC-1 Type: Displacement/Rotation
157    *Boundary
158    _PickedSet4, 1, 1
159     PickedSet4, 2, 2
160    ** Name: BC-2 Type: Displacement/Rotation
161    *Boundary
162    _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.

```
118    ** Section: Solid_Placeholder
119    *Solid Section, elset=_PickedSet3, material=Epoxy
120    1.,

                            ⋮

141    ** MATERIALS
142    **
143    *Material, name=Epoxy
144    *Elastic
145    3500., 0.34
146    *Plastic
147     30.,      0.
148     35., 0.0005
149     40., 0.0015
150     45.,  0.003

                    ⋮
```

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

Results
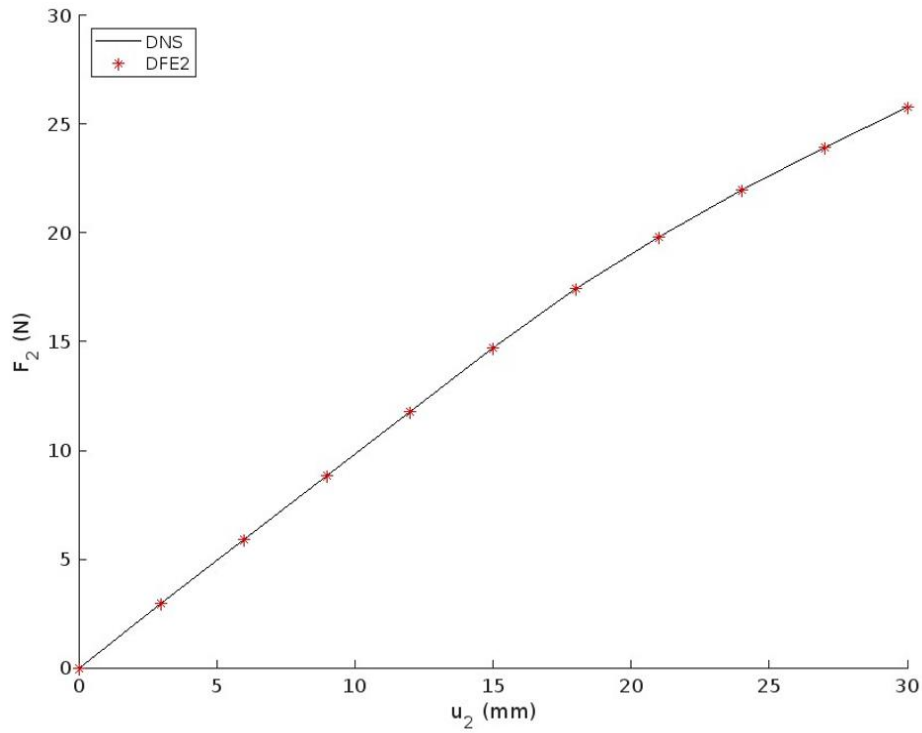
The results from both models are then compared.

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

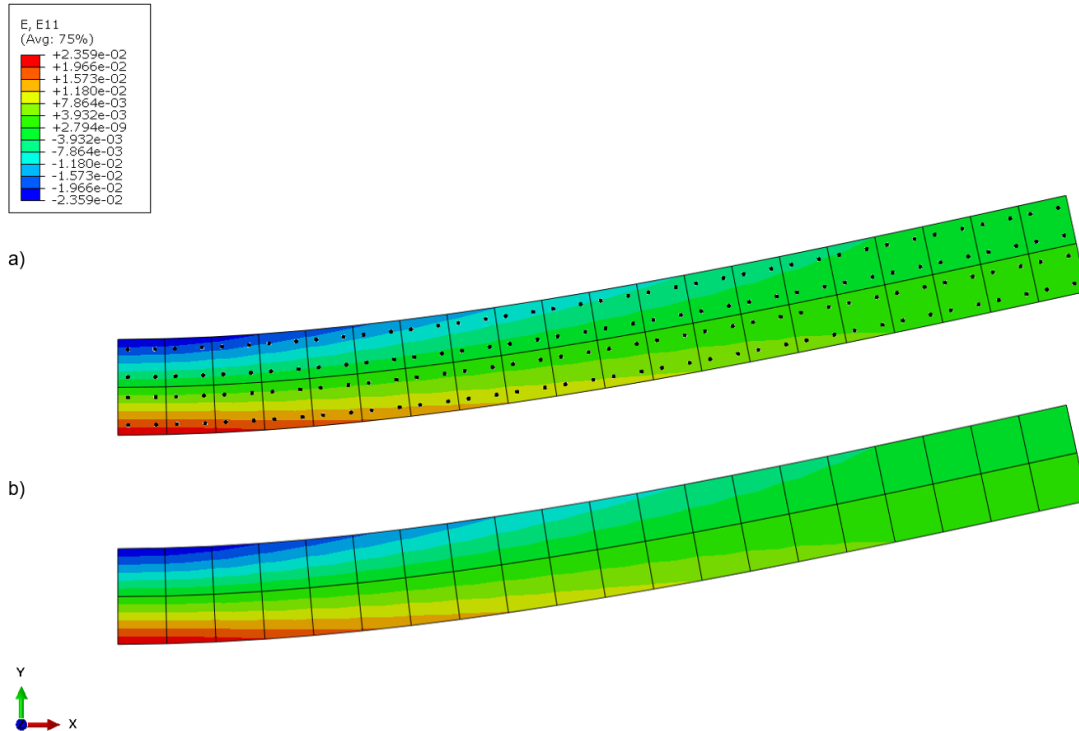contours from both models match, as seen in Figure 5 and Figure 6.

Figure 5 Axial strain contours for the homogeneous cantilever beam a) Direct FE^2 model; b) DNS model
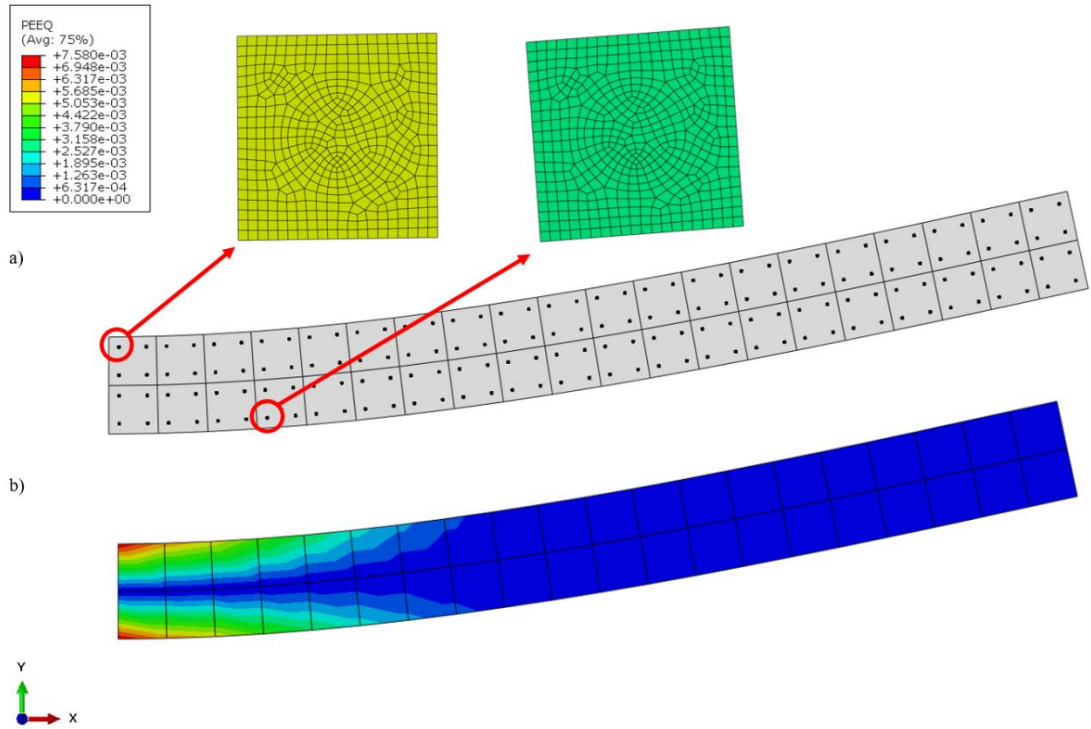
Figure 6 Equivalent plastic strain contours for the homogeneous cantilever beam a) Direct FE^2 model; b) DNS model

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.