Fracture mechanics Tutorials - Tensile cracking of a pre-cracked cube a 3D example of PSD parallel solver

Mohd Afeef Badri

Abstract

This document details some tutorials of 'fracture mechanics' module of PSD. These tutorials are not verbose, but does instead give a kick start to users/developers for using PSD's 'fracture mechanics' module.

A three-dimensional test synonymous to its two-dimensional counterpart introduced above is used here as an tutorial example. The problem of interest is now a unit extrusion (along z-axis) of the 2D case above. Cracking is initiated and propagated under tensile loading. The unit cube with its pre existing crack is clamped at the bottom $u_1 = u_2 = u_3 = 0$ (first boundary condition) and is loaded quasi-statically $u_2 = u_2 + \Delta u_2$ on its top surface till the crack propagates through its walls. So there are two Dirichlet conditions one on the top border and one on the bottom one.

Just like in the 2D case, to model this test PSD's' hybrid phase-field modelling technique is used. We will again use ParaView post-processing of displacement u and phase-field d to visualise the cracking process. A PSD simulation is a two step process, with step one being the PSD_PreProcess :

```
PSD_PreProcess -dimension 3 -problem damage -model hybrid_phase_field \
-dirichletconditions 2 -postprocess ud
```

Notice that the flags used here are almost similar except for the -dimension 3 flag, which indeed specifies three-dimensional problem.

Once the step above has been performed, we solve the problem using four MPI processes, with the given mesh file tensile-crack.msh. This is step two of the PSD simulation PSD_Solve.

```
PSD_Solve -np 3 Main.edp -mesh ./../Meshes/3D/tensile-crack.msh -v 0
```

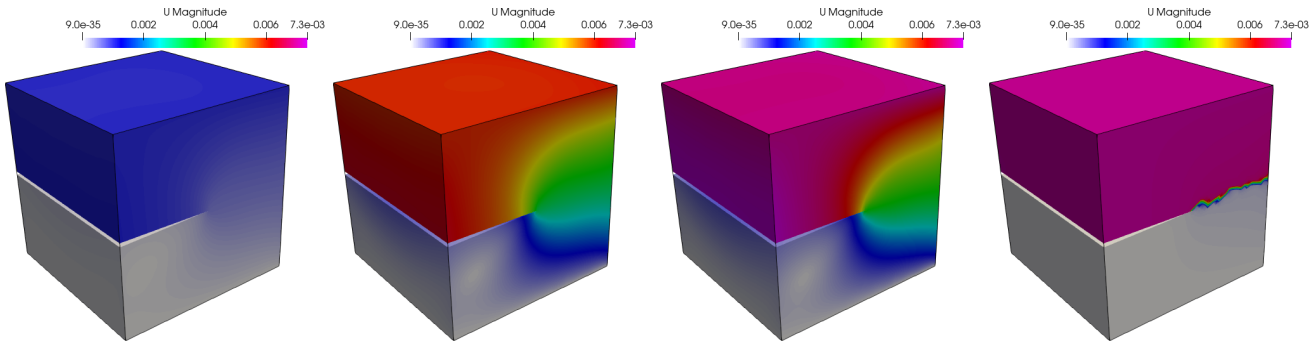

Figure 1: Finite element displacement visualised for the 3D problem with ParaView at different timesteps (quasi-statics). Time progresses from left to right in a row and top to bottom when comparing rows.

Figures 1 and 2 present the finite element displacement and damage field of the 3D problem, which enable us to visualise the cracking of the cubic specimen.

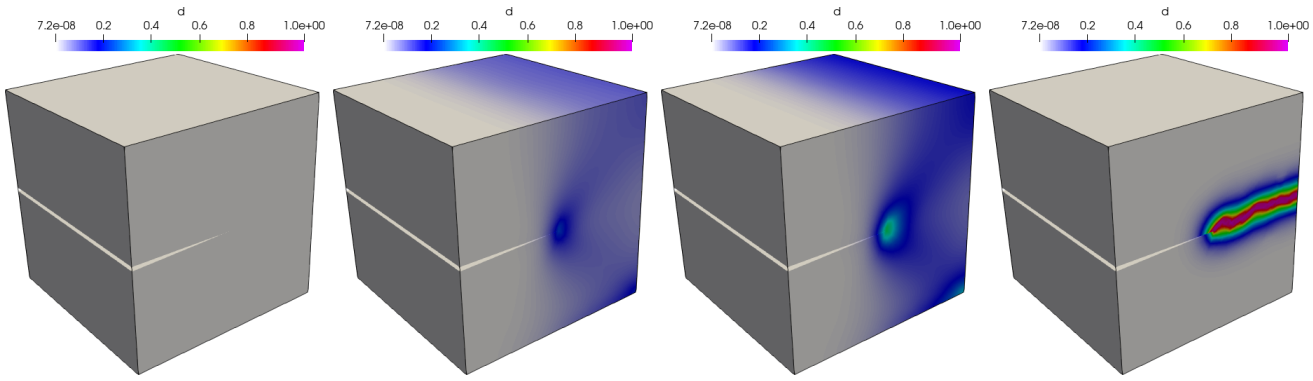

Figure 2: Finite element damage visualised for the 3D problem with ParaView at different timesteps (quasi-statics). Time progresses from left to right in a row and top to bottom when comparing rows.