Linear Elasticity Tutorial 3D mechanical piece (Dirichlet-Neumann case) with complex mesh

Mohd Afeef Badri

Abstract

This document details a single tutorials of 'linear elasticity' module of PSD in a more verbos manner.

So far in the previous cases we only concentrated on bar simulations, which were more or less trivial cases. Moreover, the bar meshes are provided with the PSD solver. In this section we now turn towards 3D simulation of a mechanical piece, the geometry of which is shown in fig. 1. The left (small) hole is fixed: $u_1 = u_2 = u_3 = 0$, while as traction force $t_x = 10^9$ is applied on the large hole.

You can grab a copy of CAD geometry for the mechanical piece (the Gmsh .geo) your local Gmsh installation folder gmsh/share/doc/gmsh/demos/simple_geo/piece.geo. To generate the mesh piece.msh simply do

```
gmsh -3 piece.geo -format msh2
```

Now the PSD simulation can be performed.

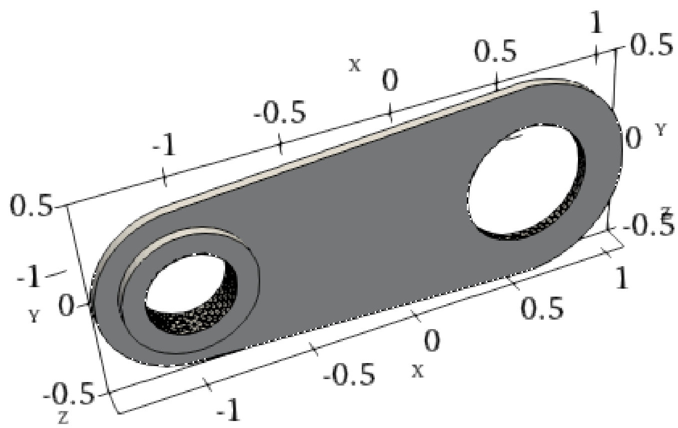

Figure 1: 3D mechanical piece.

Step 1: Preprocessing

As a prerequiste, place the generated mesh piece.msh in any folder of your choice (assuming you are working in a folder psd-complex-simulation). For "PSD Preprocessing" phase, in the folder psd-complex-simulation launch the terminal there and run the following command.

```
PSD_PreProcess -problem linear-elasticity -dimension 3 -dirichletconditions 1 -tractionconditions 1 -postprocess u
```

Here, by using these parameters we have generated one Dirichlet condition and one traction condition, respectively to be applied to the small and the large holes in the mesh. Further, by using -dimension 3 we have let PSD know that the problem is 3D .In the /PSD/Meshes/3D/piece.msh generated, the label 4 (resp. 3) corresponds to the Dirichlet (resp. traction) border. To provide Dirichlet conditions on label number 4 ($u_x = 0, u_y = 0, u_z = 0$) in ControlParameters.edp use set Dbc0On 4, Dbc0Ux 0., Dbc0Uy 0., and Dbc0Uz 0.. To add the values and label numbers of the traction borders edit the ControlParameters.edp, set Tbc0On 3 and Tbc0Ty -1.e9. For this end $\mathbf{t} = [t_x, t_y, t_z] = [0., 10^9, 0.]$. Finally we use steel properties for the material, so in ControlParameters.edp the parameters real E = 200.e9; and real nu = 0.3; should be used. These represent E and ν, respectively.

Step 2: Solving

Let us now use 2 cores to solve this problem. To do so enter the following command:

```
PSD_Solve -np 2 Main.edp -mesh ./piece.msh
```

Step 3: Postprocessing

Launch ParaView and have a look at the .pvd file in the PSD/Solver/VTUs_DATE_TIME folder.

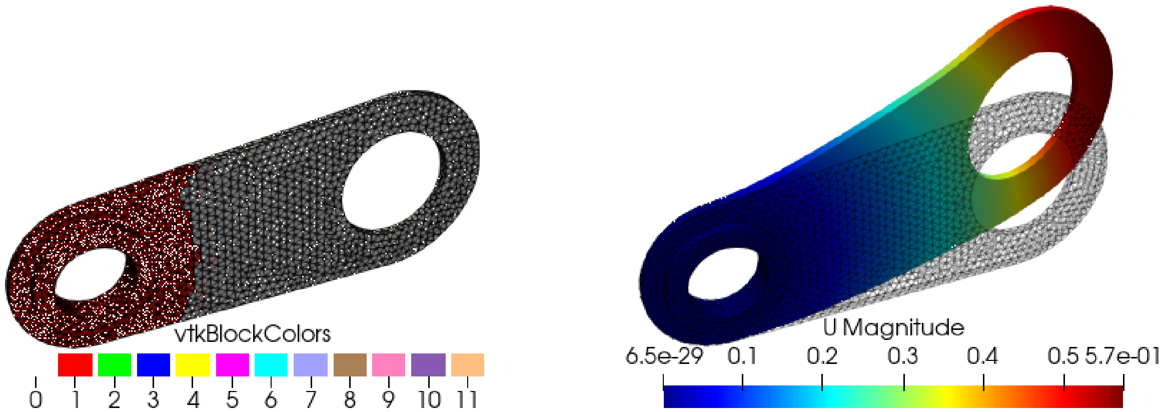

Figure 2: Mechanical piece test results. Partitioned mesh (left) and warped displacement field (right).

Redoing the test with different conditions

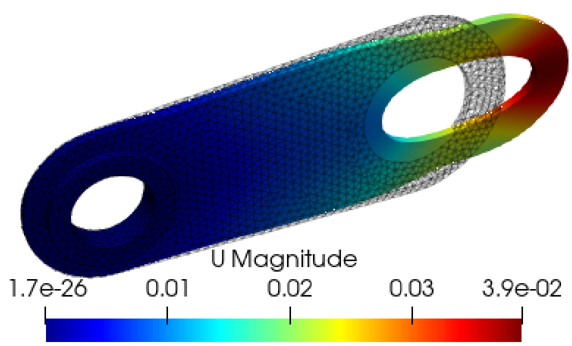

Figure 3: Mechanical piece test results: real tx0=1.e9, ty0=0, tz0=0.,;

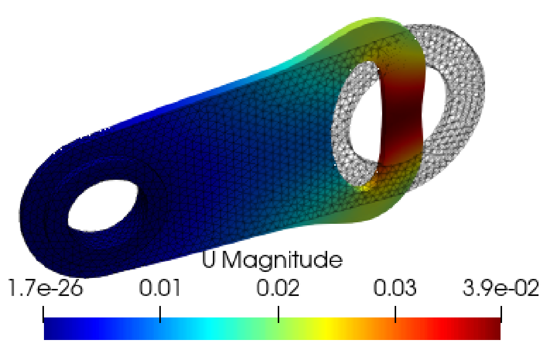

Figure 4: Mechanical piece test results:real tx0=1.e9, ty0=0, tz0=0.,;