

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`. The listing of the file is also given in @. To generate the mesh `piece.msh` simply do

```
1 gmsh -3 piece.geo
```

Place the generated mesh `piece.msh` in `/PSD/Meshes/3D/piece.msh`. Now the PSD simulation can be performed.

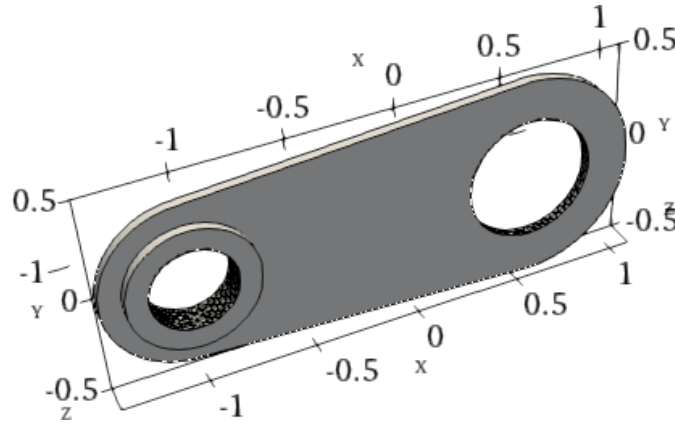


Figure 1: 3D mechanical piece.

Step 1: Preprocessing

For “PSD setup” go to any folder, launch the terminal there and run the following command.

```
1 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.

With all the properties and boundary conditions set we now use `string ThName = “../Meshes/3D/piece”;` in the `ControlParameters.edp` file, this notifies PSD about the name of the mesh used for this simulation.

Step 2: Solving

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

```
1 PSD_Solve -np 2 Main.edp
```

Step 3: Postprocessing

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

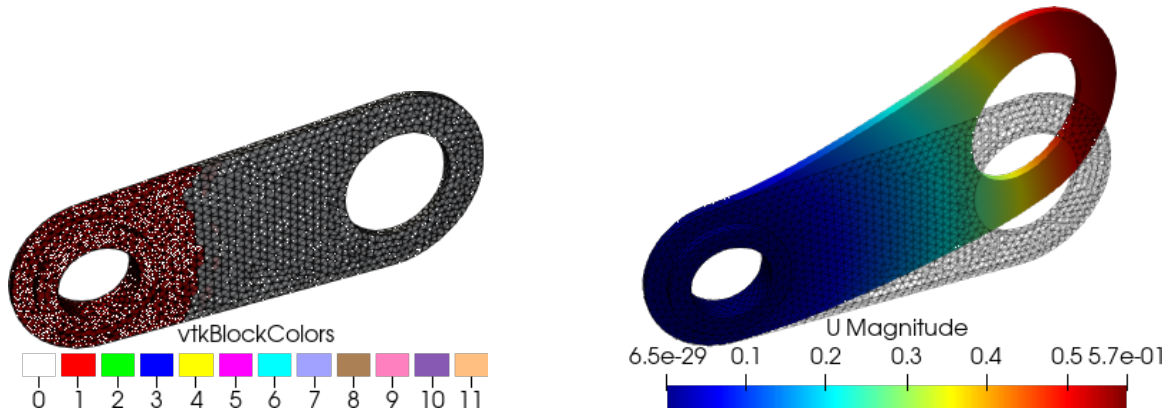


Figure 2: Mechanical piece test results. Partitioned mesh (left) and warped displacement field (right).

Redoing the test with different conditions

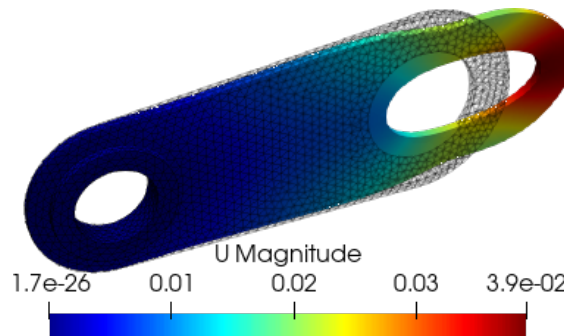


Figure 3: Mechanical piece test results: `real tx0=1.e9, ty0=0, tz0=0.;`

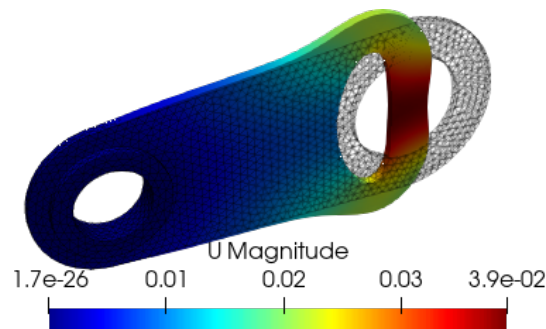


Figure 4: Mechanical piece test results: `real tx0=1.e9, ty0=0, tz0=0.;`