

Linear Elasticity Tutorial 2D bar problem clamped at one end while being pulled at the other end (Dirichlet–Neumann case)

Mohd Afeef Badri

Abstract

This document details a single tutorial of ‘linear elasticity’ module of PSD in a more verbose manner.

Similar simulation, as in the previous tutorial is presented in this section. We showcase the 2D bar problem simulation with one end clamped while being pulled at the other end. Just like the previous simulation the body force is neglected, However now the non clamped ends pull is approximated with Neumann force $\int_{\partial\Omega_N} (\mathbf{t} \cdot \mathbf{v}^h)$. To simulate the pull we assume traction vector $\mathbf{t} = [t_x, t_y] = [10^9, 0]$ acting on the non clamped right end of the bar, i.e., force in x direction is 10 units. Here is how PSD simulation of this case can be performed.

Step 1: Preprocessing

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

```
1 PSD_PreProcess -problem linear-elasticity -dimension 2 -dirichletconditions 1 -tractionconditions 1 -postprocess u
```

the commandline flag `-dirichletconditions 1`, notifies to PSD that there is one Dirichlet border —the clamped end of the bar— in this simulation. And the flag `-tractionconditions 1` notifies to PSD that there is one traction border —the right end of the bar— in this simulation. To provide the clamped boundary condition ($u_1 = 0, u_2 = 0$) set the variables `Dbc0On 2`, `Dbc0Ux 0.`, and `Dbc0Uy 0.` in `ControlParameters.edp`. In the same file traction boundary conditions are provided via the variables `Tbc0On 4` and `Tbc0Tx 1.e9`, which mean apply traction force $\mathbf{t} = [t_x, t_y] = [10^9, 0]$ on label number 4 (right) of the mesh. If user wishes to add traction force, for instance $t_y = 100.$, simply add the missing macro `macro Tbc0Tx 1.e9 //`.

Step 2: Solving

Let us now use 5 cores to solve this problem. To do so enter the following command:

```
1 PSD_Solve -np 5 Main.edp
```

Notice, that this is the exact same command used in solving the previous bar problems from other sections, with only difference that we now use `-np 5`.

Step 3: Postprocessing

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



Figure 1: 2D bar results. Partitioned mesh (left) and 100X warped displacement field (right).

Note now in~1 there are five subdomains in the partitioned mesh since five cores were used. Contrary to previous tutorial, as expected, we see that the right end of the bar which is being pulled now contract in y direction. This is due to the fact that there is no Dirichlet condition at this end now.