Linear Elasticity Tutorial 2D bar problem clamped at one end wile being pulled at the other end (Dirichlet-Neumann-Point boundary conditions case)

Mohd Afeef Badri

Abstract

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

Similar simulations, as in the prvious tutorial is presented in this section. We showcase the 2D bar problem simulation with one end clamped wile being pulled at the other end. Contrary to simulation in the previous tutorial, the clamped end just restricts x movement, i.e, $u_x = 0$. Just like simulation from the prvious tutorial the body force is neglected. Just like simulation in the prvious tutorial , the non clamped ends pull is approximated with Neumann force $\int_{\partial\Omega_{\mathrm{N}}^h}(\mathbf{t}\cdot\mathbf{v}^h)$. 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 10^9 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.

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

Additional flag -dirichletpointconditions 1 now appears, this notifies to PSD that there is one Dirichlet point boundary condition. Edit the ControlParameters.edp to communicate the desired point boundary conditions, set the variables Pbc0Ux 0. and Pbc0Uy 0. to specify $u_x = 0, u_y = 0$, and variable PbcCord = [[0. , 0.]]; to specify the point coordinates $(x, y) = (0, 0)$. Via the flags we specified that -dirichletconditions 1, i.e., there is one Dirichlet border. To provide the Dirichlet condition $(u_x = 0)$ set the variables Dbc0On 2 and Dbc0Ux 0. in ControlParameters.edp. PSD understands that 4 is the mesh border label on which Dirichlet is applied and $(u_x = 0)$ is the condition to be applied.

Step 2: Solving

Let us now use 6 cores to solve this problem. To do so enter the following command:

```
PSD_Solve -np 6 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 6.

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~fig. 1 there are six subdomais in the partitioned mesh. As expected, we see that the right and the left end of the bar which is being pulled now contract in y direction, and the bar elongates in x direction.