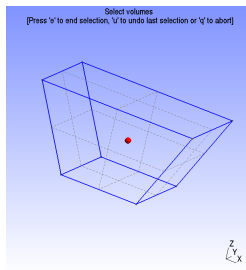


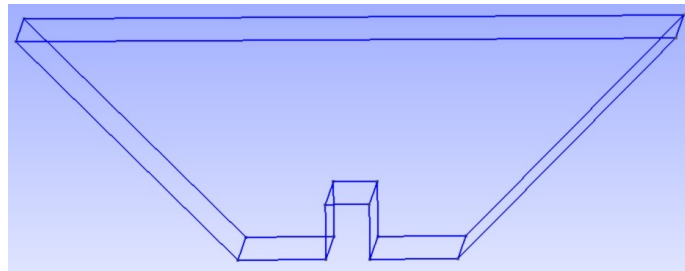
# CEE 618 – Scientific Parallel Computing: Homework #10

Name: \_\_\_\_\_

March 22, 2013



(a)



(b)

1. In Figure (a), set the top flow 1.0 m/s in  $+x$  direction and the bottom flow 2.0 m/s in  $-x$  direction, i.e.,  $v_x = 1.0$  m/s and  $v_x = -2.0$  m/s at the top and bottom surfaces, respectively.
  - (a) Use gmsh, OpenFOAM, and Paraview for mesh generation, icoFoam (for laminar, incompressible flow), and visualization for pressure, velocity, and streamlines.
  - (b) Make the mesh finer and repeat the simulation using serial and parallel modes. Compare speed up.
  - (c) [Upload your movies on YouTube and share links with me.](#)
2. (Optional) Generate a mesh for Figure (b), use OpenFOAM to simulate pressure and velocity, and visualize them using Paraview. You can choose appropriate sizes and lengths for the geometrical object of Figure (b) as you want.

Keep working on your final project proposal and polish it. To decide a thesis topic, three months of reading is necessary. You are spending about three (or more) weeks to finally setup your final project and write a proposal. A full proposal (5 pages) will be due April 5, 2013 (after spring break). Proposal format will be announce later, which will be similar to that of US National Science Foundation<sup>1</sup>

---

<sup>1</sup>[http://www.nsf.gov/publications/pub\\_summ.jsp?ods\\_key=gpg](http://www.nsf.gov/publications/pub_summ.jsp?ods_key=gpg)