Mesh generation and coding in FEniCS

Yuxiang Gao

FEniCS

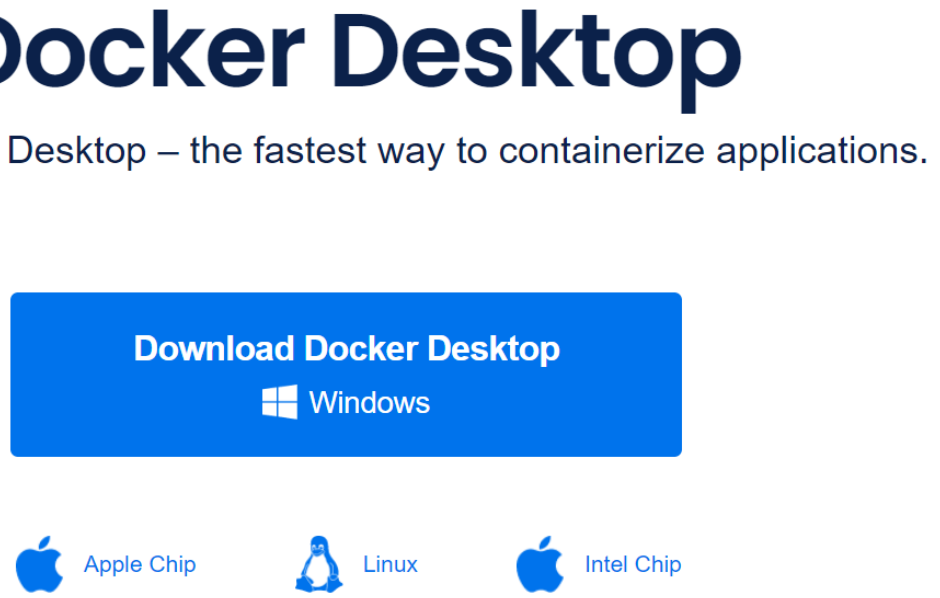
- An open-source computing platform for solving partial differential equations (PDEs).
- Python and C++ interface.
- Can run on a multitude of platforms but mainly for Linux.
- FEniCSx is the latest project but there are less tutorial and code resources than FEniCS.

FEniCS installation

- Official Download tutorial https://fenicsproject.org/download/archive/
- Run FEniCS code in **Docker** so it works for all OS (Windows, Mac and Linux).
- Docker can provide a virtual OS and enable you to download and run the prebuilt FEniCS image (based on Linux).

Install Docker

- Install Docker for desktop and laptop
  https://www.docker.com/products/docker-desktop/



Start FEniCS environment

1. Start Docker
2. Change the path to the working directory (folder) in the terminal window
3. Enter the following command in the terminal window to start FEniCS

Start FEniCS environment

Windows:

docker run -ti -p 127.0.0.1:8000:8000 -v %cd%:/home/fenics/shared -w /home/fenics/shared quay.io/fenicsproject/stable:2016.2.0

MacOS:

docker run -ti -p 127.0.0.1:8000:8000 -v $(pwd):/home/fenics/shared -w /home/fenics/shared quay.io/fenicsproject/stable:2016.2.0

Reference:

127.0.0.1:8000:8000 : The port of the container (your FEniCS environment). Need to be different for different containers.

%cd% or **$(pwd) :** Current path. Only need to note that it's different for Windows and MacOS/Linux.

2016.2.0 : The version of the FEniCS. Can be replaced with **current**.

More reference of docker command: https://docs.docker.com/engine/reference/run/

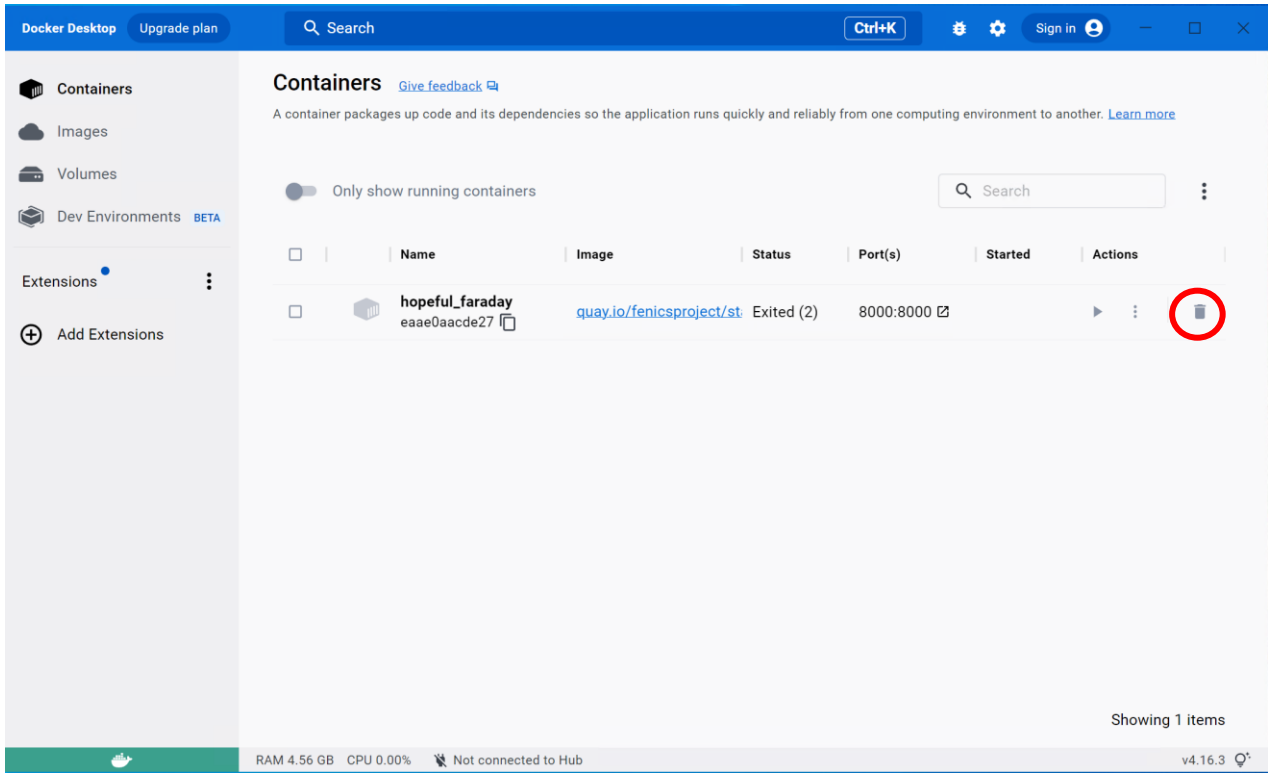
Run FEniCS python code

The python based FEniCS program can be run by

python filename.py

Delete the container

By GUI (Windows example):



By command in Linux (In a new terminal):

First, stop all containers by

sudo docker stop $(sudo docker ps -q);

Then, remove all stopped containers by

sudo docker rm $(sudo docker ps --filter status=exited -q)

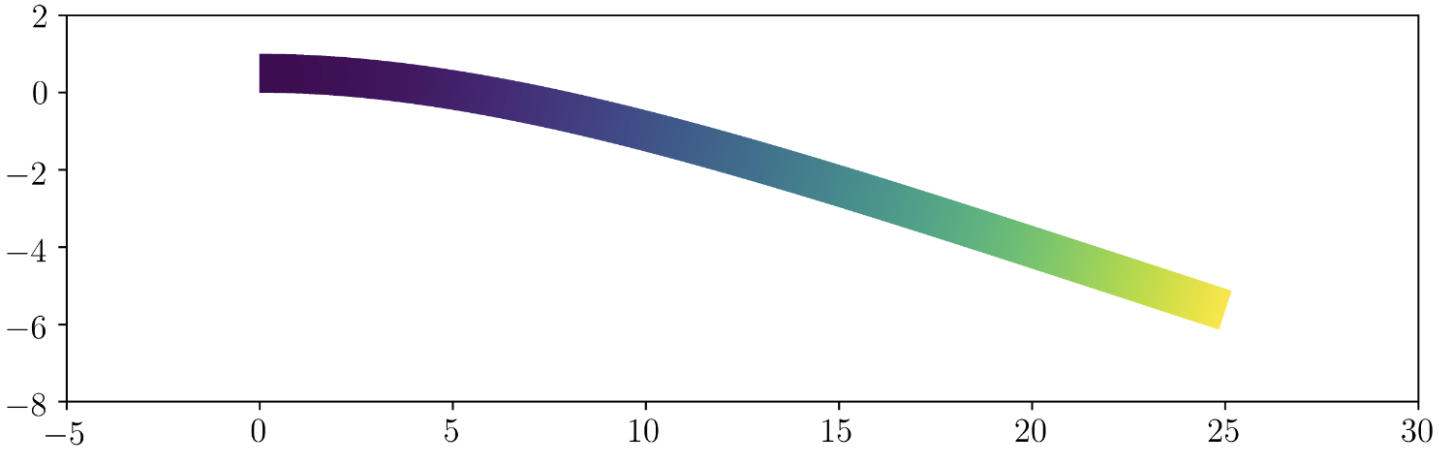
2D elasticity problem

An example from

Bleyer, J. (n.d.). Numerical tours of continuum mechanics using FEniCS. 125.

https://comet-fenics.readthedocs.io/en/latest/demo/elasticity/2D_elasticity.py.html

Cantilever beam under gravity (body force).



Mesh generation by Gmsh

- Free: Open-source 3D finite element mesh generator
- Parametric modeling: The Gmsh application programming interface (API) allows to integrate the Gmsh library in external applications written in C++, C, Python, Julia or Fortran.

Document: https://gmsh.info/doc/texinfo/gmsh.html

For using Python script, you can install Gmsh by

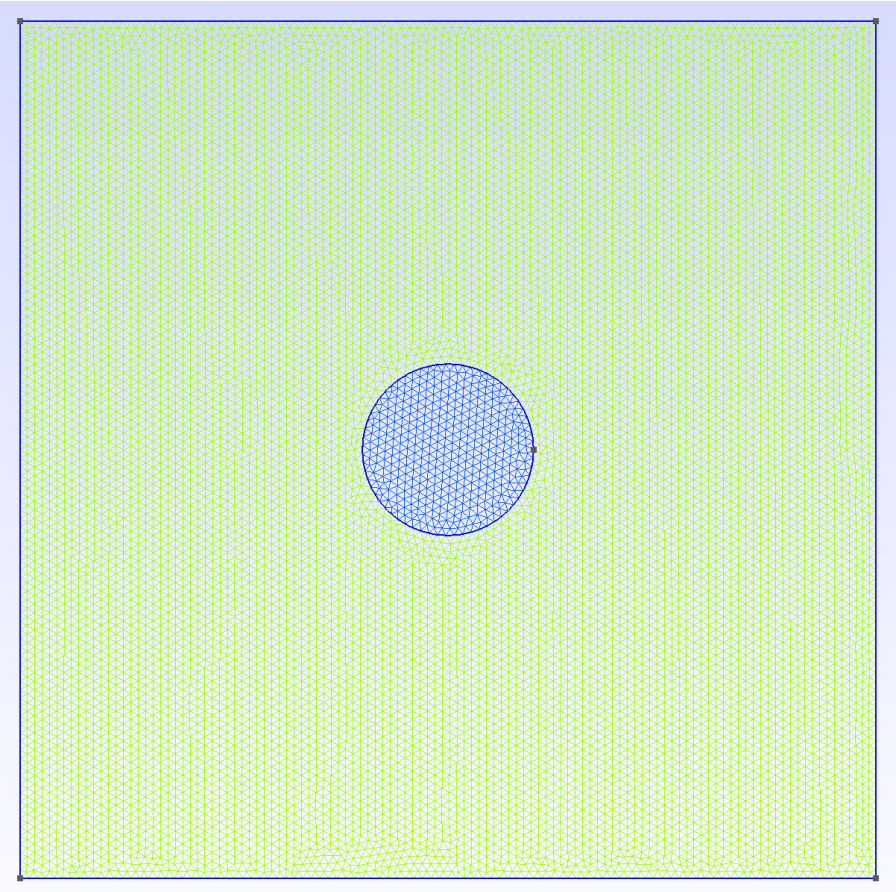
pip install --upgrade gmsh

Mesh generation by Gmsh Python script

1. Draw geometry by Gmsh inbuild CAD functions
2. Pass the geometry to Gmsh meshing
3. Define the meshing parameters
4. Generate the mesh and save them

Mesh generation by Gmsh Python script

An example: Generate FEM mesh for a square plate with fiber by Python.



Import 2d triangle mesh from Gmsh

- Generate mesh in Gmsh and save the mesh as .dat file
- Convert the .dat file to .xml file with the format loadable by FEniCS by Gmsh2xml.py (posted on GitHub)
  - You need to state the name of the mesh file in the code.
- In FEniCS code, import the xml mesh by
- *mesh = Mesh("mesh.xml")*

Materials

- Codes are on GitHub
  - https://github.com/YuxiangGao0321/FEniCS_tutorial_2023
- Books for FEniCS
  - Bleyer, J. (n.d.). Numerical tours of continuum mechanics using FEniCS. 125.
  - Langtangen, H. P., & Logg, A. (n.d.). Solving PDEs in Python – The FEniCS Tutorial Volume I. 153.
  - Logg, A., Mardal, K.-A., & Wells, G. (Eds.). (2012). Automated Solution of Differential Equations by the Finite Element Method (Vol. 84). Springer Berlin Heidelberg.