OpenFoam on Cannon and Desktop/Laptop

- Introduction
- Openfoam via singularity container
  - On the RC-Cluster
    - Running interactively
    - Running in batch
  - Modeling on your own desktop/laptop

Introduction

This page has information on running OpenFoam interactively as well as in batch mode on the RC-Cluster (currently Cannon). To learn basics of modeling with OpenFoam, please visit:

OpenFOAM - Modeling Basics

Openfoam via singularity container

We have built a singularity container with freecad, gmsh, openfoam and paraview. The container can be used on the RC-Cluster (in both the interactive and batch modes) or on a local linux machine with singularity installed.

On the RC-Cluster

Running interactively

Connect to the RC-Cluster (currently Cannon) via OnDemand (https://www.rc.fas.harvard.edu/resources/documentation/virtual-desktop/). Open a terminal and create a directory for OpenFoam:

```bash
mkdir openfoam-test
cd openfoam-test
```

Run the openFoam container with shell option to obtain a command shell.

```bash
singularity shell /n/seas_computing/scientific_software/freecad-OpenFoam.simg
```

Once inside shell, you should be able to run a simple test job as follows:

```bash
source /opt/openfoam8/etc/bashrc
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/pitzDaily .
cd pitzDaily
blockMesh
simpleFoam
```

Running in batch

To run openfoam in batch, you can create a bash script with openfoam setup, meshing and run commands in a script (say scr_pitzDaily) and invoke the script via singularity in a slurm submit script (say openfoam_pitzDaily.sh). The script "scr_pitzDaily" might look like:

```bash
#!/usr/bin/env bash
source /opt/openfoam8/etc/bashrc
rm -rf pitzDaily
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/pitzDaily .
cd pitzDaily
#Create mesh
blockMesh
#Run openfoam via the solver simpleFoam
simpleFoam
```

The submit script "openfoam_pitzDaily.sh" might look like:
#!/bin/bash
#SBATCH -n 1  #Number of cores
#SBATCH -N 1  #Number of nodes
#SBATCH -t 60  #Runtime in minutes
#SBATCH -p general  #Partition to submit to
#SBATCH --mem-per-cpu=500 #Memory per cpu in MB (see also --mem)

module load intel/17.0.4-fasrc01 impi/2017.2.174-fasrc01

gesture exec /n/seas_computing/scientific_software/freecad-OpenFoam.simg ./scr_pitzDaily

# Report some useful info
date
exit

Modeling on your own desktop/laptop

Installing OpenFOAM (with FreeCAD, GMSH, and ParaView) on Desktop/Laptop.

The singularity image on The RC-Cluster (freecad-OpenFoam.simg) can be downloaded to your desktop/laptop and run in a flavor of Linux (eg: ubuntu) with singularity installed. For this you need:

- Either a linux desktop/laptop or linux installed in a virtualized environment (such as virtualbox). For help with this, just use a modern search engine (google, bing, etc.) and look for "ubuntu installation in virtualbox" etc. There is plenty of documentation available on this.
- Install singularity. Visit the following and look under documentation for users:
  https://www.sylabs.io/docs/
- Once you have singularity installed, the commands are identical to those under "OpenFoam via singularity – running interactively" described above.