OpenFoam on Odyssey and Linux Desktop

- Introduction
- Openfoam via singularity container
  - On Odyssey
    - 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 FASRC Cluster (currently Odyssey). 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 odyssey (in both the interactive and batch modes) or on a local linux machine with singularity installed.

On Odyssey

Running interactively

Connect to odyssey 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/openfoam6/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/openfoam6/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:
Modeling on your own desktop/laptop

The singularity image on Odyssey (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.