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](https://www.rc.fas.harvard.edu/resources/documentation/virtual-desktop/)

**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/](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.