

MicroSim GP OpenFOAM solver installation

Requirements:

To build and run the solver modules, the users need OpenFOAM and GSL. The modules are tested using OpenFOAM-in-Box-20 in Ubuntu 18.04 and Ubuntu 20.04, and OpenFOAM-6 in Ubuntu 18.04. Installation image of Ubuntu 20.04 can be downloaded from the link below:

<https://releases.ubuntu.com/20.04.4/>

OpenFOAM-in-Box-20 is recommended for new users due to ease of installation, and can be downloaded from the links below:

- Official link with installation guide: <https://www.cfdsupport.com/openfoam-in-box.html>
- Unofficial link: <https://drive.google.com/file/d/17gkbpQTK54Hq1A7GNk-s6rktQDfrtcnn/view?usp=sharing>

Installation:

The following commands can be used in the terminal to begin the installation (check official link above if needed):

```
mkdir -p ~/OpenFOAM/OpenFOAM-in-Box
cp OpenFOAM-in-Box-20.09v2-linux64.sh ~/OpenFOAM/OpenFOAM-in-Box/
cd ~/OpenFOAM/OpenFOAM-in-Box/
bash OpenFOAM-in-Box-20.09v2-linux64.sh -install
echo "source ~/OpenFOAM/OpenFOAM-in-Box/OpenFOAM-in-Box-20.09v2-22-g178c07ee/OpenFOAM-dev/etc/bashrc" >> ~/.bashrc
source ~/.bashrc
```

Cases can be run after creating the directory below:

```
mkdir -p $FOAM_RUN
```

GSL can be installed using the following command in the terminal:

```
sudo apt install libgsl-dev
```

For visualization and post-processing, ParaView 5.6 and 5.8 are tested. ParaView 5.8 comes along with OpenFOAM in Box 20 and can be launched using the command:

```
paraview
```

To view the plots, gnuplot can be used, which can be launched using the command:

```
gnuplot
```

It is advised to refer to the OpenFOAM documentations to understand the methods involved while using. The below link can be useful:

- <https://cfd.direct/openfoam/documentation/>

Using OpenFOAM modules

Solver has to be compiled from the solver directory. For instance:

```
cd solver
wclean
wmake
```

Finally, switch to the cases directory, e.g. coolingAlZn:

```
cd cases/coolingAlZn
```

Note: Allclean and Allrun must be set as executables so that the above Allrun and Allclean command can work. You might have to run the following command just once for all the cases after you download the cases from the repository:

```
chmod +x Allrun
chmod +x Allclean
```

To run the case with the default parameters, execute the following:

```
./Allclean
./Allrun
```

To check the results in ParaView:

```
paraview phaseField.foam
```