MicroSim Installation: GP OpenFOAM solver

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.

OpenFOAM-in-Box-20 is recommended for new users due to ease of installation.

Download:

- 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

Install:

1. **OpenFOAM**

   The following commands can be used in the terminal to begin the installation:

   ```
   mkdir -p ~/OpenFOAM/OpenFOAM-in-Box
   # go to the folder where OpenFOAM-in-Box-20.09v2-linux64.sh is stored, then:
   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 'alias ofbox20="source ~/OpenFOAM/OpenFOAM-in-Box/OpenFOAM-in-Box-20.09v2-22-g178c07ee/OpenFOAM-dev/etc/bashrc"' >> ~/.bashrc
   source ~/.bashrc
   ```

2. **GSL**

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

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

Using OpenFOAM modules

Load environment variables of OpenFOAM using (To use OpenFOAM or paraview in a new terminal, you need to run the following command)

```
ofbox20
```

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
```

Visualisation

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 documention to know more about OpenFOAM. The below link can be useful:

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