Additive-Manufacturing Code User Guide

by Swapnil Shadanan Bhure

📌 This code has been developed by Ravi Kumar Singh as a part of his Master's Thesis at Materials Modelling Group , Indian Institute of Science, Bangalore, India. Digital copy of his thesis is available on our GitHub repository.

⚠️ This code has been tested on and works with OpenFOAM-4.x and requires third-party tool swak4Foam.
The code is currently not compatible with newer versions of OpenFOAM.

Installation of OpenFOAM-4.x and swak4Foam works for Linux OS versions up to Ubuntu 18.04. I have tied installing them on Ubuntu > 20.04 but had a lot of trouble due to core dependencies and didn't work.

The code has also been installed and tested on cluster system with CentOS 7.

Table of contents:

Pre-requisites

Install OpenFOAM 4.x and Paraview:

This is the specific version in which the code was written and simulations were run.

Follow instructions on this page: (choose appropriate Ubuntu version)

https://openfoamwiki.net/index.php/Installation/Linux/OpenFOAM-4.x/Ubuntu#Ubuntu_18.04

This will take hours to install depending on number of cores being used to install.

There is no option of choosing the number of cores to run the code for: Section_2.5-9-3 (for Ubuntu 18.04)

```
./makeParaView -python -mpi -python-lib /usr/lib/x86_64-linux-gnu/libpython2.7.so.1.0 > log.makePV 2>&1
```

For subsequent major installation steps (third party products like Paraview), we can specify how many number of cpu cores to use (I suppose they mean physical cores and not hyper-threads)

💡 **Note**: Every time we open a new terminal to use OpenFOAM we need to source the OpenFOAM environment by running the command: `of4x`

This is done because we might have different versions of OpenFOAM on our system and each might have its own set of home directory and commands. This functionality of sourcing every time helps us choose the appropriate OpenFOAM version on the go.

> 💡 **Note**: I haven't used or tested point 13 which says: 'When you want to update your build, follow the instructions on section Steps for updating on the parent page.

Installing swak4Foam:

> 💡 The installation instructions have been given on the website only till Ubuntu 15.10. But the instructions work till Ubuntu 18.04.

1. Follow instruction in section 2.2 on this page.

   Follow step 1. Install necessary packages.

2. Follow the instructions in **point 5** on this page to download swak4Foam dev version.

   > 💡 **Note**: swak4foam's stable version 0.40 does not work with OpenFOAM 4.0 to 4.x. **We need to use swak4Foam development version** for it to work.

3. Continue ahead again on this page from point 3.
4. Check if the installation was successful by following this.

   a. Errors that I got in log.make file:

   ```
   No file 'swakConfiguration'. Python etc won't work. See README for details
   Try 'ln -s swakConfiguration.automatic swakConfiguration' for automatic configuration. BEWARE: this does not work on some sys
   Checking swak4Foam-version and generating file
   Swak version is 20xx.yy.0
   hg info: b8e73355892c (develop) tip
   No 'bear' installed
   ```

   Solution:

   ```
   ln -s swakConfiguration.automatic swakConfiguration
   sudo apt-get update -y
   sudo apt-get install -y bear
   ```

   b. Again run

   ```
   ./Allmake > log.make 2>&1
   ```

   I got some errors say - `not found`. Lets ignore that for now because the code did not abort anywhere.

   At the end of the file, I am getting this ever since I ran `Allmake` for the first time:

   > 💡 If you want to use swakCoded-function object or compile software based on swak set the environment variable SWAK4FOAM_SRC to /home/swapnil/OpenFOAM/swapnil-4.x/swak4Foam/Libraries (most people will be fine without setting that variable)

   Let's ignore this as well for now.

   c. Run:

   ```
   funkySetFields
   ```

   Error I am getting:

```
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
swakVersion: 20xx.yy (Release date: Next release)
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

-> FOAM FATAL ERROR:
funkySetFields: time/latestTime option is required

From function main()
in file funkySetFields.C at line 713.

FOAM exiting
```

This error seems to be as expected because we are running funkySetFields command out of the blue without setting up any OpenFOAM Case. So it will show some errors. **The error means that the system recognizes `funkySetField` command**. That's all we wanted to check here. This error confirms that the installation was successful.

If the error would have been something like `command funkySetField not found`. Then that would have meant that there is some issue with our installation. In that case, we would have to again refer to Understanding errors messages page.

Using Additive-Manufacturing code

> **For the following guide on additive codes, it is assumed that you are well conversant will the basic operation of OpenFOAM. You should be able to edit OpenFOAM cases, compile solvers, run cases and view results in Paraview.**

Compiling Solvers:

1. Open a new terminal window and run:

```
of4x
```

> Run this command every time you open a new terminal window or tab. This makes sure that the OpenFOAM v4.x commands and environment variables are sourced. This is crucial in case if we have multiple versions of OpenFOAM.

If we have just one version of OpenFOAM installed on our system, then this command can be automated by adding the relevant bash script lines to Ubuntu's `.bashrc` file because `.bashrc` file is automatically sourced every time we open a new terminal.

This is highly discouraged as it might create unforeseen issues that would be difficult to deal for non-expert user so not discussing here how to avoid `of4x` command.

2. **Compiling the solver:**

a. Go to `marangoniBoundary_Condition` directory and run

```
wmake
```

b. Go to `Additive_manufacturing_solver` directory and run:

```
wmake
```

Both of these should compile without any error for the first time or else it would mean that the installation is not done correctly or the code is not compatible with the OpenFOAM version.

The code has been successfully tested on Ubuntu 18.04, OpenFOAM-4.x and swak4Foam-dev.

> **Note**: This is the first time you have compiled additive solver. The solver needs to be compiled again only if you make any changes in it.

Preparing additiveTestCase:

1. Go to additiveTestCase directory via terminal. Clean the old/previous simulation data (if any) by running the following command:

   ```
   ./Allclean
   ```

   If this command does not work, make sure you ran `of4x` in the terminal and that the `Allclean` file is executable. To make the file executable run the following command:

   ```
   chmod +x Allclean
   ```

   You need to make a file executable just once. It will remain that way until you move around to an external drive or share via email, etc. In such a case, just run the above command.

2. `additiveTestCase` directory is similar to any `OpenFOAM case` directory which contains directories like `0, constant, system` and few other files. Edit them as per your problem.

   > 💡 **Note:** Remember that computation uses physical cores and not hyper-threads. So specify number of physical cpu cores you want to use in `numberOfSubdomains` in `decomposeParDict` file. Beware, if you have a quad-core processor on a laptop then for testing prefer using 2 cores. Using all the 4 cores on a laptop might slow down other apps or heat-up the laptop. You may use all cpu cores if you have a gaming or workstation laptop. Make a decision based on what your laptop can handle. For lab desktops and clusters you may use all the physical cores.

Running additiveTestCase:

> 💡 Remember to run `of4x` if using a new terminal tab or window as explained here.

To run the case, use:

```
./Allrun
```

Important points to remember:

1. There should not be any spaces in the file path in OpenFOAM.
2. Whenever you open a new terminal, source the OpenFOAM environment variables from the version that we want to use. For OpenFOAM-4.x use:

   ```
   of4x
   ```

3. To unload the environment variable, use:

   ```
   wmUNSET
   ```

   Thus, we don't have to close the window to start another version of OpenFOAM.