Appendix B

Running OpenFOAM®

```
basic@openfoamTutorials:~$ simpleFoam
/*---------------------------------------------------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Website:  https://openfoam.org                  |
|   \\  /    A nd           | Version:  7                                     |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
Build  : 7
Exec   : simpleFoam
Date   : Sep 01 2019
Time   : 09:00:00
Host   : "openfoamTutorials"
PID    : 52826
I/O    : uncollated
Case   : $FOAM_TUTORIALS/incompressible/simpleFoam/pitzDaily
nProcs : 1
sigFpe : Enabling floating point exception trapping (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster
(fileModificationSkew 10)
allowSystemOperations : Allowing user-supplied system call operations

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
Create time

Create mesh for time = 0
```

Bahram Haddadi

**ENGINEERING
FLUID
DYNAMICS**

7th edition, March 2025

Contributors:

- Bahram Haddadi
- Christian Jordan
- Michael Harasek
- Sylvia Zibuschka
- Yitong Chen
- Jozsef Nagy

Technische Universität Wien

Institute of Chemical, Environmental

& Bioscience Engineering

This offering is not approved or endorsed by ESI® Group, ESI-OpenCFD® or the OpenFOAM® Foundation, the producer of the OpenFOAM® software and owner of the OpenFOAM® trademark.

Available from: www.fluiddynamics.at

1. Running OpenFOAM® on a Local Linux PC (or virtual machine):

- Open a terminal

- Go to the OpenFOAM® installation directory (e.g. /opt/openfoam10) in the opened terminal

- Change to the etc directory in the OpenFOAM® installation directory

- Run the following command:

> `. ./bashrc`

- If a new terminal is opened, the same procedure should be repeated in that in order to activate OpenFOAM® in here.

2. Running OpenFOAM® on Remote Computers via SSH (e.g. server):

2.1. Windows:

- Run PuTTY (search for: PuTTY windows).

- Set the following:

  `Category>Session`

  Host name: openhost.university.edu

  Connection type: SSH

  `Category>Connection>SSH>Tunnels`

  Source port: 5901

  Destination: localhost:59**[1]

- Do not forget to press *Add*!

  > Please make sure that display is not used by others.

  `Category>Connection>Data`

  Auto-login username: openFoamUser[2]

  `Category>Session`

  Saved Sessions: openFoamUser

- Press *Save.*

- Now choose from "saved sessions" your session (*openFoamUser*) and press *Open*. In the opened Command (Prompt) window, it prompts for

---

[1] Display number

[2] Session ID

your password. The password is not echoed to the screen and the passwords are case sensitive.

- Immediately after entering your password, your computer will attempt to establish a connection to your server. If it is your first time connecting to that server, you will see a message asking you to confirm the identity of the machine. Make sure you entered the address properly, and type *yes*, followed by the *return* key, to proceed.

- Change to etc directory in OpenFOAM® installation directory

- Execute the following command:

>. ./bashrc

- To log out use whatever command is used to logout from the server you are logged into (typically ctrl + d).

2.2. Mac OS X and Linux:

- Open your Terminal application. You will see a window with a $ or > symbol and a blinking cursor. From here, you may issue the following command to establish the SSH connection to your server (be careful about upper case 'L' in the -gL).

>ssh -gL 5901:localhost:59** openFoamUser@university.edu

- Immediately after issuing this command, your computer will attempt to establish a connection to your server. If it is your first time connecting to that server, you will see a message asking you to confirm the identity of the machine. Make sure you have entered the address properly, and type *yes*, followed by the return key, to proceed.

- You will then be prompted to enter your password. Type or copy/paste your SSH user password into the Terminal. You will not see the cursor moving while entering your password. This is normal. Once you are finished inputting your password, press return on your keyboard. At this point, you will be connected to your server remotely through SSH.

- Change to etc directory in OpenFOAM® installation directory

- Execute the following command:

>. ./bashrc

2.2.1. Running OpenFOAM® in Graphical Interface (VNC):

- Connect to remote machine via SSH connection using part B.

- Make sure <u>VNC Server</u> is installed on the remote machine and it is started (ask administrator for display number, port and other information, for starting VNC Server check FAQ)

- Install the appropriate <u>VNC Viewer</u> and run it (search for: vnc viewer):

  VNC Server: localhost:01

- Press Connect

- Press Continue

- Enter your password

- Press *Ok*

- On VNC desktop open a terminal

- Change to etc directory in OpenFOAM® installation directory

- Execute the following command:

```
>. ./bashrc
```

If a new terminal in the VNC desktop is opened, the last two steps should be done in that to activate OpenFOAM® in there.