OpenFOAM® Basic Training

Appendix C
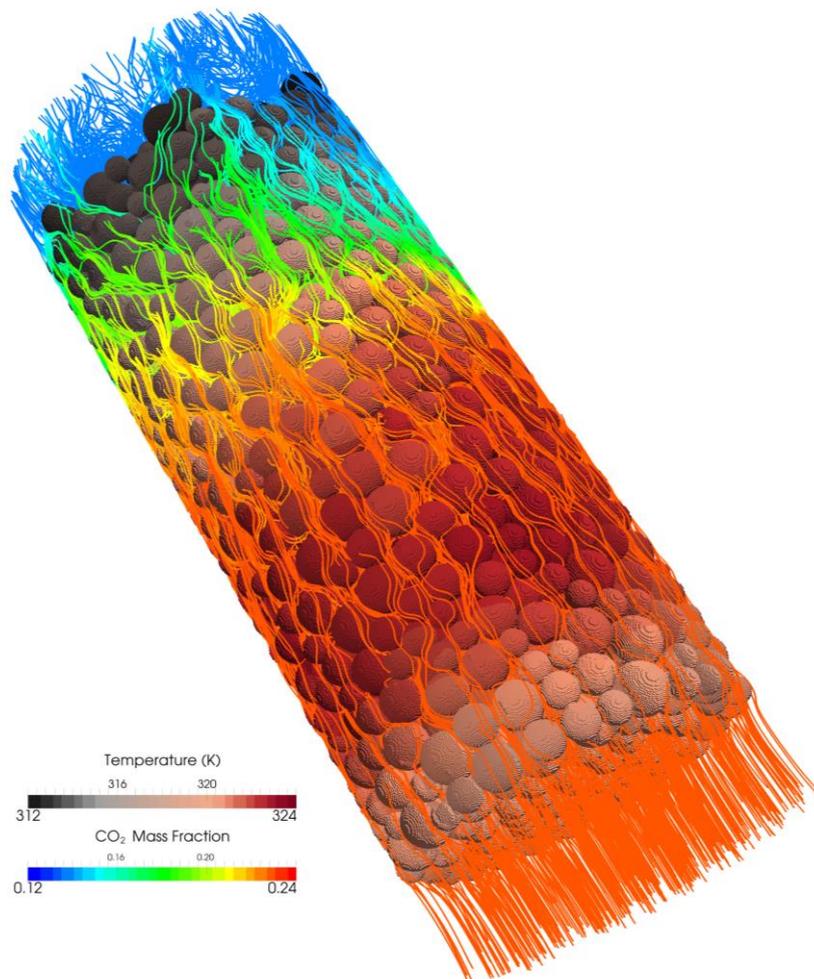
Temperature (K)

CO$_2$ Mass Fraction

3rd edition, Feb. 2015

Editors and Contributors:
- Bahram Haddadi (TU Wien)
- Christian Jordan (TU Wien)
- Jozsef Nagy (JKU Linz)
- Clemens Gößnitzer (TU Wien)
- Vikram Natarajan (TU Wien)
- Sylvia Zibuschka (TU Wien)
- Michael Harasek (TU Wien)

Cover picture from:
- Bahram Haddadi, The image presented on the cover page has been prepared using the Vienna Scientific Cluster (VSC).

Appendix C

Frequently Asked Questions (FAQ)

Q -   What should I do in case of a GAMBIT failure?

A -   e.g. Program stops responding:
- Type "ps" in the command window, search for the GAMBIT process number.
- "kill -9 PROCESS NUMBER" Enter

GAMBIT creates lock files, which must also be deleted, otherwise it is not possible to open of the affected files:
- "rm *. lok" Enter

Furthermore, "junk" (temporary files from GAMBIT) should be disposed of:
- "rm -fr GAMBIT.xxx" erases the complete directory, xxx again is the process number.
- If you have forgotten, to save before the crash, you should copy the file "jou" (it contains all the commands that have been executed and can be processed automatically in GAMBIT) from the directory, to resume its status before the crash.

Q -   How can I prevent typing long commands in the terminal for couple of times?

A -   Using curser keys to navigate line by line.

Type beginning of the command and use Tab (auto completion).

By using reverse search, use ctrl+r to search for previous commands typed in the terminal, e.g. typing a part of command show the suggestions and you can navigate through them.

Q -   My VNC is not responding from server side?

A -   First you should kill your VNC server:

```
>vncserver –kill :[YOUR DISPLAY NUMBER]
```

Restart your VNC server (according to SSH forward):

```
>vncserver:[YOUR DISPLAY NUMBER] -geometry 1600x800 -depth
24
```

Q -   I have deleted some of my files accidently. What should I do?

A -   Sorry, no recycling or undelete in Linux

Q -   Why can I not connect to the server?

A -   Check to see if you have an IP address for your network card.

Q -   How can I start VNC Viewer from my Linux computer terminal?

A -   Use command:

```
>vncviewer :[NUMBER OF LOCAL PORT, e.g. 1 or 2]
```

Q - Error "command not found"?

A - Make sure OpenFOAM® and ParaView are installed correctly. Check Appendix B for starting OpenFOAM®.

Q - Does foamToVTK command not work for chtMultiRegionFoam?

A - Use command:

```
>foamToVTK -region[REGION NAME]
```

Q - Is it possible to export animations from ParaView?

A - Yes, by choosing .ogv file format from "file/save animation" menu. The output will be a video file with .ogv format. In the new ParaView versions (4.3.0) the animation can also be saved using .avi format.

Q - Is there any tool in Linux to convert series of ParaView pictures to video?

A - Yes, command line tool ffmpeg:

```
>ffmpeg -r [FRAME  PER SECOND RATE] -f image2 -i [images
names, e.g. rho.%4d.jpg] [OUTPUT FILE NAME].[OUTPUT FILE
FORMAT, e.g avi]
```

Q - How can complex geometries be patched?

A - During creating the geometry in the preprocessing software, e.g. GAMBIT, create volume zones, which you will need to patch later (see software user manual for creating regions in each software). For converting the mesh to the OpenFOAM® mesh use the appropriate tool with "-writeZones" flag to import zones to OpenFOAM®, e.g.:

```
>fluentMeshToFoam  -writeZones  <your mesh>
```

then in the setFieldDict change it like this:

```
regions
(
    zoneToCell
    {
        name air; // region name which you assigned in gambit
        fieldValues
        (
            volScalarFieldValue alpha.water 0 // the value of property
                                              //which you want to patch
        );
    }
);
```

Then after running setFields tool, it will assign the values to that region.

Q - How can I create a bash scripting file for executing couple of command in series?

A - Instead of typing command sequences one by one after each other and executing them. It is possible to put all those commands in a file and execute that file to run them. This is known as "bash scripting".

Bash scripting is typically used in the cases when the same simulation should be run with identical settings a couple of times, but with a few changes. For bash scripting create an empty file (e.g. using nano editor creating text file "go"):

> nano go

Add the commands to this file (e.g. commands for running blockMesh, setFields, decomposePar, compressibleInterFoam in parallel mode and reconstructPar):

```
blockMesh
setFields
decomposePar
mpirun –np 4 compressibleInterFoam –parallel >log
reconstructPar
```

Exit the editor and save the file (ctrl+x , y, enter for nano editor).
For changing this file to an executable file, file permissions should be set. By using this command file permissions are displayed:

>ls –la go

-rw-r--r-- 1 e166**** E020D166 73 Aug 23 9:15 go

The first 'r' shows that this text file can be read by user, the 'w' shows that user has the permission to write this file, but the '–' sign shows that this file is not executable by the user. To change this permissions execute following command:

>chmod u+x go

Now this file is executable:

>ls –la go

-rwxr--r-- 1 e166**** E020D166 73 Aug 23 9:15 go

Now you can run the simulation by this executable text file:

>./go

After executing the file, the commands added to the file will be executed one by one. In most of the OpenFOAM® tutorials there are **Allrun** and **Allclean** files, which are bash scripts for running the case and cleaning a case, respectively.