

# Porting OpenFOAM to MAC OS X

Bernhard F.W. Gschaider<sup>1</sup> Hrvoje Jasak<sup>2</sup>

<sup>1</sup>ICE Strömungsforschung

<sup>2</sup>Wikki Ltd

June 9, 2007

# Outline

## ① Basics

It's only UNIX (but I like it)  
Vendor provided software

## ② Porting

Preparing the port  
Additional software  
Compiling  
Paraview

## ③ Unported and Future stuff

Problems  
The Future

# The architecture of MAC OS X

- Starting with MAC OS X10.0 the APPLE<sup>TM</sup>-OS is based on (BSD)UNIX
  - Has a shell and all our beloved CLI-utilities
  - Some differences to regular UNIX-machines
  - This layer is know as DARWIN. Therefor the \$WM\_ARCH is called darwin
- The APPLE<sup>TM</sup>-specific interface and services are built on top of the UNIX-layer
  - Origins in the NEXT<sup>TM</sup>-operating system
  - None of these things are needed by OPENFOAM
- So it *should* be easy to port OPENFOAM to MAC OS X

## BSD vs System V

- Historically there are two types of Unix:
  - **System V** Stems from AT&T
  - **BSD** From the university of Berkley
- Both conform to the POSIX-Standard. Sort of
- OPENFOAM was developed on LINUX<sup>TM</sup>
  - More on the SYSTEM V side
- MAC OS X is based on a BSD-kernel
  - With a *sugar coating* by APPLE<sup>TM</sup>

## Case-insensitive filesystems

- ext2 (the standard LINUX™-filesystem is **case-sensitive**:

```
1 touch theCase  
touch TheCase
```

Yields two files: theCase and TheCase


- APPLE HFS is case-insensitive: The above commands yield only theCase
  - Case of the file is known
  - ... but not used to distinguish files
- Variant APPLE HFSX is case-sensitive
  - but using it on your system-disk kills the system
    - Well, actually we never dared to try
- OPENFOAM has source file that only differ by case
  - Example: Scalar.H, scalar.H

## Development tools

- Every Mac comes with media that install a complete development system
  - g++**, **gcc** Compiler (currently in version 4.0.1)
  - make** A GNU MAKE
  - flex**, **bison** Compiler generators
  - JDK** JAVA™ (currently 1.5)
- These are the most important tools to port OPENFOAM

## X-Windows

- The Mac's GUI is called AQUA - programmed with the API COCAO
- The standard API in the UNIX-world is X11 (X-Windows)
- APPLE™ provides a X-Server (and the X11-API)



```
xterm
Executing: /Users/bgschaid/OpenFOAM/OpenFOAM-1.4/.bashrc
Executing: /Users/bgschaid/OpenFOAM/OpenFOAM-1.4/.OpenFOAM-1.4/apps/ensightFoam/
bashrc
Executing: /Users/bgschaid/OpenFOAM/OpenFOAM-1.4/.OpenFOAM-1.4/apps/paraview/bas
hrc
BGS-iBook:~ bgschaid$ cd $FOAM_SRC
BGS-iBook:~/OpenFOAM/OpenFOAM-1.4/src bgschaid$ ls
Allmake                foamUser
Allmake"               foamUtil
Gstream                fvMotionSolver
LESmodels              lagrangian
MGridGenGangAgglomeration  malloc
ODE                    meshTools
OpenFOAM               randomProcesses
Pstream                sampling
dynamicFvMesh          thermophysicalModels
dynamicMesh            topoChangerFvMesh
edgeMesh               transportModels
engine                 triSurface
errorEstimation        turbulenceModels
finiteVolume
BGS-iBook:~/OpenFOAM/OpenFOAM-1.4/src bgschaid$
```

# Getting a case-sensitive filesystem

(Sorry. I only have a german system. Therefor the names of the GUI-elements are “back-translations”)

- 1 Open the program Applications-Utilities - Disk Utility
- 2 Click on New Image
  - Select a name and a location for the image
  - Select a size (1 GB should be sufficient. Can be grown later)
- 3 Click in the pane on the left on the created image
- 4 Go to the Erase-tab
  - For Volume-Format select one of the case-sensitive Mac OS Extend-variations
  - Click the Erase-button
- 5 Try to mount the image by double-clicking it
  - Try the touch-test described above

## Permanently adding the disk image

If you want your OPENFOAM-installation to be permanent:

- Mount the Image at login

- 1 Go to your User-settings in System Preferences
- 2 Select the Start Objects-tab
- 3 Click on the Plus-button and add your image

- Symlink the image to the “usual” place

- In a shell say

```
ln -s /Volumes/Foam ~/OpenFOAM
```

(assuming your image mounts as Foam)

- Modify the `.bashrc`

- 1 Add a call to the OF-initialization file (described in the Readme-file)
- 2 Add the line

```
1 ulimit -S -n 2048
```

(this prevents an exhaustion of file-handles with `wmkdep`)

# Preparing the sources

## 1 Go to the disk image

```
1 cd ~/OpenFOAM
```

## 2 Untar the source-distribution there (nothing else is needed)

```
1 tar xvzf ~/Downloads/OpenFOAM-1.4-General.tgz
```

## 3 Download from [http://openfoamwiki.net/index.php/Howto\\_compile\\_OpenFOAM\\_Mac](http://openfoamwiki.net/index.php/Howto_compile_OpenFOAM_Mac)

- A tar with the wmake-rules for the darwin-architecture. To be untarred
- Two patches. To be applied in order. To apply a patch do

```
1 cd ~/OpenFOAM/OpenFOAM-1.4  
  patch -p0 -b <~/Download/thePatch
```

- A new version of error.C (allows backtraces)

## What the patches do

Issues the patches take care of:

- Add `DARWIN` to the known architectures
- Deficits in the `APPLETM`-headers (missing math-stuff)
- `malloc.h` is in a different place
- Libraries that are missing in the options-files of some utilities
- Reset the `MICO` to the `DARWINPORTS` one (thus enabling the compilation of `FoamX` and `patchTool`)
- Help to compile the `paraview-filter` without human intervention
- Remove warnings about unsupported options for the `head-command` from some scripts

These patches will be inserted into the `SUBVERSION`-repository

# MacPorts (FKA DarwinPorts)

To get the missing software:

- 1 Go to <http://www.macports.org/> and install it
  - A similar project is <http://www.finkproject.org/> (provides binaries as well)
- 2 Install the missing packages:

**Mico** If you don't want to use the version that comes with  
**OPENFOAM**

```
sudo port install mico
```

The sources (and everything they depend on) are  
automagically downloaded and compiled

**binutils** If you want to use the backtrace-facility

```
1 sudo port install binutils
```

**MPI** The MPI-implementation of your choice. For instance

```
1 sudo port install openmpi
```

The files in **OpenFOAM-1.4/.OpenFOAM-1.4** have to be  
adapted accordingly

## Do it like the README says

This is the “hardest” part

- 1 Check whether everything is in place
- 2 Do

```
1 cd ~/OpenFOAM/OpenFOAM-1.4  
  ./Allwmake
```

- 3 Get some coffee
- 4 Wait
- 5 Get more coffee
- 6 Watch PVRReader-compilation fail (should be the only failure at that stage)
- 7 Celebrate the end of the compilation with a cup of coffee
- 8 Try some utilities (blockMesh, interFoam) to make sure it worked

## Why compile PARAVIEW™

- paraFoam is a script that adds a plugin to PARAVIEW™ and calls the original paraview
  - This plugin is the PVReader that failed during our first compilation
- To compile a plugin for PARAVIEW™ you have to have the source-code
  - ... or know the settings with which your binary was compiled
- The surest way to know the settings is to compile PARAVIEW™ yourself

Q.E.D.

# Compiling it

- 1 Download the source code (2.4.x is “supported” but 2.6.x works as well. **No** 3.x-versions)
- 2 Untar and compile it
  - Doesn't have to be on a case-sensitive filesystem
  - For details look at the Wiki-page
- 3 Adapt the initialization files so that `$ParaView_Dir` is correct
- 4 Open a new shell and in that retry `Allwmake`
  - Now `PVReader` should compile
- 5 Try `paraFoam`

## An alternative: PARAVIEW3™

- PARAVIEW3™ already has reader for OPENFOAM included
  - But it is not represented in the Open File-Dialog of the GUI
- Reader can be added to the Dialog with a .... plugin
  - [http://openfoamwiki.net/index.php/Tip\\_Build\\_A\\_Paraview3\\_Plugin](http://openfoamwiki.net/index.php/Tip_Build_A_Paraview3_Plugin)
  - PARAVIEW3™ has to be compiled from source

### Advantages

- Cool new features
- X11 no longer needed (native AQUA-GUI)

### Disadvantages

- Reader seems to be a bit unstable
- Works **only** if there are **no** .gz-files in the case

## Problem: FoamX won't kill its sub-processes

- After running FoamX some processes remain
  - FoamXHostBrowser
  - nsd
- Should be killed by the FoamX-script
  - Are not (because of the different output of the BSD-ps)
- Workaround: manually kill the processes
  - Or rewrite the scripts
- The same is true for patchTool

## Problem: No right-click in FoamX

- Two contradicting concepts:
  - MACs only have one mouse-button
  - UNIX-software usually assumes three mouse-buttons
- Right mouse-button is emulated by Ctrl-Click
  - This doesn't work in FOAMX
- Theory: The JAVA virtual machine “swallows” the Ctrl-key
- Workaround: Use an external USB-mouse (with three buttons)
- Same is true for patchTool

## A Tale of two Architectures

- Currently MAC OS X exists on two processor architectures:
  - **ppc** The POWERPC-architecture (no longer used by APPLE™)
  - **i386** The INTEL™ processors
- Currently only ppc-compilations are known (though i386 shouldn't be a problem)
- Using both compile options `-arch ppc -arch i386` produces *Universal binaries*
  - Currently untested (lack of time, disk space and an INTEL™-Mac)
  - Disadvantage: Binaries are twice the size they should be

# How will it be distributed?

Two basic ways of distribution

## 1 In source form

- On the forthcoming `openfoam-extend-SUBVERSION-Repository`
- Disadvantage: the preparation of the environment has to be done by the user
- Propably will not be incorporated in the official distribution

## 2 As disk images

- Have been provided by Hrv in the past
- Disadvantage: Updates will only be available when someone has the time to produce a new image
- Problem: for which architecture (`ppc`, `i386` or *Universal*)
- Proposal: prepare disk image for `SVN`-access (in other words: give the image the possibility to update itself)

## Acknowledgments

People who contributed to this port by providing contributions/feedback:

**Michael Wild** Pointed out that the Disk Utility **can** produce case-sensitive file-systems

**Doug Hunsaker** Asked a lot of questions that told me what is “not so trivial”

**Simon Stapleton** Who pointed out that the FOAMX-orphans are a BSD-problem and promised a fix

**Bouke** Got foamInstallationTest to run

**James Turner** Found a problem with the wmkdep-tool

**Petr Vita** For having nothing to do with this port

... these are the ones I remember. Sorry for everyone I forgot

# The End

**Thanks for listening!**

Questions?