

A universal OpenFOAM build script

Using Ansible to make your machine "OpenFOAM ready"


Bernhard F.W. Gschaider

HFD Research GesmbH

Graz

3. July 2019

Outline

- 
- 1 Introduction
 - This presentation
 - Who is this?
 - What is your problem?
 - 2 Ansible
 - What it is
 - How it works
 - 3 The "build" script
 - Advanced Ansible
 - Preparation
 - Building
 - Supported
 - 4 The End
 - Controversial topics
 - Goodbye

Outline

1 Introduction

- This presentation
- Who is this?
- What is your problem?

2 Ansible

- What it is
- How it works

- Advanced Ansible

3 The "build" script

- Preparation
- Building
- Supported

4 The End

- Controversial topics
- Goodbye

HFD

What it's about

A programmer is a person that does a lot of work because he is lazy

- I often compile OpenFOAM
 - And sometimes I have to set up machines to do so
- This is repetitive work
- So I wrote something to do it for me
- In the time I could have set up a lot of machines
 - To make it worth the while other people should use it

Outline

1 Introduction

- This presentation
- Who is this?
- What is your problem?

2 Ansible

- What it is
- How it works

- Advanced Ansible

3 The "build" script

- Preparation
- Building
- Supported

4 The End

- Controversial topics
- Goodbye

HFD

Bernhard Gschaider

- Working with OPENFOAM™ since it was released
 - Still have to look up things in Doxygen
- I am **not** a core developer
 - But I don't consider myself to be an *Enthusiast*
- My involvement in the OPENFOAM™-community
 - Janitor of the openfoamwiki.net
 - Author of two additions for OPENFOAM™
 - [swak4foam](#) Toolbox to avoid the need for C++-programming
 - [PyFoam](#) Python-library to manipulate OPENFOAM™ cases and assist in executing them
 - Organizing committee for the OPENFOAM™ *Workshop*
- The community-activities are not my main work but *collateral damage* from my real work at ...

Heinemann Fluid Dynamics Research GmbH

The company



- Subsidiary company of *Heinemann Oil*
 - Reservoir Engineering
 - Reservoir management

Description

- Located in Leoben, Austria
 - ... and Vienna
- Works on
 - Fluid simulations
 - OPENFOAM™ and Closed Source
 - Software development for CFD
 - mainly OPENFOAM™
- Industries we worked for
 - Automotive
 - Processing
 - ...

Outline

1 Introduction

- This presentation
- Who is this?
- What is your problem?

2 Ansible

- What it is
- How it works

- Advanced Ansible

3 The "build" script

- Preparation
- Building
- Supported

4 The End

- Controversial topics
- Goodbye

HFD

Binary distributions

There are OpenFOAM-binaries

- The Foundation supplies Ubuntu-Packages
- OpenCFD supplies RPM-packages
- Both supply Docker-images that run on Linux, Windows and MacOS
- There is a number of binaries rolled by other people

Why I still compile OpenFOAM

- Docker makes me uneasy
 - Especially if people don't publish their Dockerfile
- There is no binary for **this** OpenFOAM version on **this** operating system
- I often need a Debug-version
 - Because my code has bugs
- Sometimes I need an ancient version of OpenFOAM
 - Recently compiled OpenFOAM 2.3. Don't ask

Steps to compile

Compilation always requires similar steps

- 1 Make sure the required software is installed
- 2 Get the sources
- 3 Put the sources in the right place
- 4 Download the necessary third-party sources
- 5 Extract in the correct places
- 6 Edit configuration to use the correct versions
- 7 Start the compilation
- 8 Curse because I forgot one of steps 1-6

It is not rocket science but repetitive error-prone work

- which is the worst kind of work

Bonus: compile the compiler

- Sometimes one needs to compile a compiler from the sources.
Because
 - the compiler on the system is too old for OpenFOAM
 - a different compiler is needed. For development. Bug reproduction.
Because.
- Then additional steps are needed
 - 1 Download the compiler sources
 - 2 Download additional library sources
 - 3 Extract everything in the right places
 - 4 Edit the compiler setting
 - 5 Run the compiler script in ThirdParty
- Boring. Repetitive

Outline

- 1 Introduction
 - This presentation
 - Who is this?
 - What is your problem?
- 2 Ansible
 - What it is
 - How it works
- 3 The "build" script
 - Preparation
 - Building
 - Supported
- 4 The End
 - Controversial topics
 - Goodbye
- Advanced Ansible



Outline

1 Introduction

- This presentation
- Who is this?
- What is your problem?

2 Ansible

- What it is
- How it works

■ Advanced Ansible

3 The "build" script

- Preparation
- Building
- Supported

4 The End

- Controversial topics
- Goodbye

HFD

Wikipedia says

Ansible is an open-source software provisioning, configuration management, and application-deployment tool

In other words:

- A tool to set up computers automatically
 - Especially remote computers
 - Which computers are set up is listed in an *inventory*
- Setup is written in *playbooks*
 - Stored on the management machine
- Similar packages are Puppet, CfEngine and Chef
 - But they need special *Agents* on the target machines

Basically: you write how the environment should look like and Ansible tries to make it look that way

- Around since 2012
 - The developing company is part of RedHat since 2015
- Written in Python

What it can do

There is a lot of modules but the most important ones do

- install software
- create files, directories and links (and set their attributes)
- create users and groups
- edit files (add lines, remove lines)
- manipulate source control repositories
- download files (and check their integrity)
- copy files to the remote machine

Implemented in such a way that subtle difference between distros don't matter

- ... and if all else fails it executes regular commands

What it requires

On the master

- Ansible
 - Which needs Python

- An ssh client

On the target machines

- Python
- A ssh server
- **No** Ansible

Except for Ansible this is pretty standard on Linux machines

Outline

1 Introduction

- This presentation
- Who is this?
- What is your problem?

2 Ansible

- What it is
- How it works

■ Advanced Ansible

3 The "build" script

- Preparation
- Building
- Supported

4 The End

- Controversial topics
- Goodbye



YAML: Python of file formats

- OpenFOAM was written in C++
 - The file format looks like C++
 - With ; { }
- YAML looks like Python
 - Indentation is important
 - Less "special" characters
 - More lines

YAML

```
---  
number: 2  
name: foo  
text: "Long␣text"  
list:  
  - one  
  - two  
dict:  
  a: 1  
  b: 2
```

OpenFOAM

```
number 2;  
name foo;  
text "long␣text";  
list ( one two );  
dict {  
  a 1;  
  b 2;  
}
```


Specifying how the world should be

- Ansible does this in **playbooks**
 - These are YAML-files with Jinja-expressions
- Parts of playbooks
 - hosts** On which machine should this be executed
 - vars** pre-defined variables
 - tasks** main part. A list of tasks. A task has
 - name** A readable name that is reported
 - copy / file / lineinfile / debug** Name of the module which is a dictionary with its parameters.
 - register** Save the result of the task to a variable
 - when** condition to switch the task on and off
- Ansible supplies variables with information about the system

Creating and editing a file

The playbook

```

---
- hosts: localhost
  vars:
    fname: foo
  tasks:
    - name: "Create_{{fname}}_in_/tmp"
      copy:
        src: "/etc/passwd"
        dest: "/tmp/{{fname}}"
        force: no
      register: target
    - name: "Mark_my_entry"
      lineinfile:
        path: "{{target.dest}}"
        line: "*****_we_are_here_*****"
        insertbefore: "^{{ansible_user_id}<brk>
          <cont>}}:"
        register: insert
    - name: "Report_success"
      debug:
        msg: "Modified_{{target.dest}}_on<brk>
          <cont>_{{ansible_nodename}}"
      when: insert.changed
  
```

What it does

- Copies the contents of /etc/passwd to /tmp/foo
- Searches for the entry of the current user
 - Inserts a "marker" line before that if there is none
- If a line was inserted it prints a message

Run it once

The command `ansible-playbook` executes the playbook

```
> ansible-playbook setLine.yml
[WARNING]: provided hosts list is empty, only localhost is available. Note that the <brk>
<cont>implicit localhost does not match 'all'

PLAY [localhost] *****

TASK [Gathering Facts] *****
ok: [localhost]

TASK [Create foo in /tmp] *****
changed: [localhost]

TASK [Mark my entry] *****
changed: [localhost]

TASK [Report success] *****
ok: [localhost] => {
  "msg": "Modified /tmp/foo on bg-postmac-budgie"
}

PLAY RECAP *****
localhost                : ok=4    changed=2    unreachable=0    failed=0
```

Now everthing is good

- Running it a second time doesn't change anything
 - This is called **idempotence**

```
> ansible-playbook setLine.yml
[WARNING]: provided hosts list is empty, only localhost is available. Note that the <brk>
<cont>implicit localhost does not match 'all'

PLAY [localhost] *****

TASK [Gathering Facts] *****
ok: [localhost]

TASK [Create foo in /tmp] *****
ok: [localhost]

TASK [Mark my entry] *****
ok: [localhost]

TASK [Report success] *****
skipping: [localhost]

PLAY RECAP *****
localhost                : ok=3    changed=0    unreachable=0    failed=0
```


Outline

1 Introduction

- This presentation
- Who is this?
- What is your problem?

2 Ansible

- What it is
- How it works

■ Advanced Ansible

3 The "build" script

- Preparation
- Building
- Supported

4 The End


- Controversial topics
- Goodbye

HFD

Roles

- Ansible-roles are usable like regular modules
 - Can have parameters
- Organized in a special directory structure with
 - Tasks
 - Variable files
 - Data files (including templates)
 - Information about dependencies
 - roles that are automatically run before running the tasks in this role
- All the tasks of the *build script* are organized in roles

Outline

- 
- 1 Introduction
 - This presentation
 - Who is this?
 - What is your problem?
 - 2 Ansible
 - What it is
 - How it works
 - 3 The "build" script
 - Preparation
 - Building
 - Supported
 - 4 The End
 - Controversial topics
 - Goodbye

Organization

- The "build script" is organized as a couple of *roles* that depend on each other
 - Before the role `openfoambuild` can build OpenFOAM it needs `openfoamdir`
 - Before `openfoamdir` can create a proper directory structure in needs `openfoamrequirements`
 - `openfoamrequirements` makes sure everything needed for the build is there
- The user writes a little playbook that specifies
 - Which OpenFOAM-version he wants
 - How they should be built
- Executing the playbook does everything else

The following slides explain **what** is done not **how** it is done

Outline

- 1 Introduction
 - This presentation
 - Who is this?
 - What is your problem?
- 2 Ansible
 - What it is
 - How it works
- 3 The "build" script
 - Preparation
 - Building
 - Supported
 - Advanced Ansible
- 4 The End
 - Controversial topics
 - Goodbye



Creating a openfoam-user

- This is the user that "owns" the OpenFOAM-installation
 - Not a "normal" (human) user
 - But it should not be root
- It is made sure that such a user-exists
 - A different name can be selected
 - Name of the current user if no "special" user is desired
 - But you can name him `fluent` as well

Installing requirements

- This part makes sure that required software is installed like
 - Developer tools (compilers etc)
 - System tools (sudo)
 - Source control tools (git and mercurial)
 - Development libraries (zlib etc)
 - MPI-implementations
- It assumes that this is a vanilla installation
 - Minimal installation of the Linux distro
 - **Nothing** is there
 - Adds software repositories
- This is the part that strongly depends on the Linux-distro
 - Switches to the correct playbook depending on the distro

Creating a OpenFOAM-directory

- OpenFOAM usually assumes that it is in a directory OpenFOAM in \$HOME
 - Directory foam for foam-extend
- Script creates that directory
 - Make openfoam the owner
 - Adds a site directory for config files
 - A symbolic link foam pointing to it
- openfoambuild will add the proper sub-directories to it

Outline

- 1 Introduction
 - This presentation
 - Who is this?
 - What is your problem?
- 2 Ansible
 - What it is
 - How it works
- 3 The "build" script
 - Advanced Ansible
 - Preparation
 - Building
 - Supported
- 4 The End
 - Controversial topics
 - Goodbye



What it does

- Downloads the correct sources (ThirdParty as well)
 - Either a tarball
 - a git archive
- Extracts the sources to the correct directories
- Puts the sources under version control (optional)
- Applies patches
- Runs the compilation script
 - If this is successful the script will **not** be run again
- All this is controlled by settings for each distro in a dictionary `distros`
 - Settings can be overwritten per Linux-distro (because of different requirements) in `distroarch`

Example for a distro-specification

From defaults/main.yml

```

p1906:
  name: OpenFOAM-v1906
  # where to load the sources from. Only used if 'gitrepo' is undefined
  tarball: https://sourceforge.net/projects/openfoamplus/files/v1906/OpenFOAM-  
<cont>v1906.tgz
  # tarball is checked against this checksum before extracting
  tarballcheck: md5:ab7017e262c0c0fceec55c31e2153180
  # tarball with the ThirdParty-Sources. If no repo is defined
  tarball3rd: https://sourceforge.net/projects/openfoamplus/files/v1906/  
<cont>ThirdParty-v1906.tgz
  tarball3rdcheck: md5:81af204ef7aa804bc6b633f20f28749e
  name3rd: ThirdParty-v1906
  thirdParty:
    - metis51
  version: v1906
  # The API version that was introduced in v1812
  api: "1906"
  hasBuildDir: True

```

This is all that had to be added to support OpenFOAM v1906

Building ThirdParty

- Regular ThirdParty-sources are downloaded automatically
- Additional third party software is downloaded
 - Specified in the `thirdParty`-setting
 - May be different depending on the distribution
 - If there is no appropriate MPI-implementation for instance `openmpi211` can be specified to get `OpenMPI 2.11`
- Appropriate files in `OpenFOAM/site` are modified to use it
- This software is automatically compiled
- Name, locations etc specified in configuration dictionary `thirdParty`

Building compilers

- Compilers are a special case of third party software
 - Depend on multiple tarballs
 - OpenFOAM-scripts expect a special directory structure
- This is configured in `compilerSpec`
- If a compiler is compiled then it is made the default for this distro
 - People who use more than one compiler consider this a bug

Example for a playbook

Build a OpenFOAM-distro for development

exampleConfigs/normalAndDebug.yml

```
---
- hosts: localhost
  roles:
    - role: openfoambuild
      vars:
        buildopts: WM_COMPILE_OPTION=Opt
    - role: openfoambuild
      vars:
        buildopts: WM_COMPILE_OPTION=Debug
  vars:
    patchdevelopmode: 1
    foamgroup: openfoam
    compiler: Clang80
```

Single command entertains the computer for a couple of hours

```
> sudo ansible-playbook exampleConfigs/normalAndDebug.yml --extra-vars="distro=p1906"
```

Building additional software

- There is a role `openfoamdocu` that builds the Doxygen-documentation
 - Makes sure that the distro is compiled
 - Installs the necessary software (`doxygen`, `graphviz`)
- Roles for selected software packages
 - `cfmesh` for distros that don't bring this
 - `swak4foam` The *Swiss Army Knife For OpenFOAM*. No machine should be without this
 - `pyfoam` Python library and utilities for OpenFOAM
- These roles
 - Make sure that the distro is compiled
 - Install into a special `AdditionalSoftware`-directory for every distro separately

Outline

- 1 Introduction
 - This presentation
 - Who is this?
 - What is your problem?
- 2 Ansible
 - What it is
 - How it works
- 3 The "build" script
 - Advanced Ansible
 - Preparation
 - Building
 - Supported
- 4 The End
 - Controversial topics
 - Goodbye



How to get it

- The "Homepage" of the script is `https://openfoamwiki.net/index.php/Installation/Ansible`
- Easiest way to get it: Mercurial

Download

```
> hg clone http://hg.code.sf.net/p/openfoam-extend/ansibleFoamInstallation
> cd ansibleFoamInstallation
```

- Further documentation in the README

Supported OpenFOAM forks

The three forks are supported in their latest versions

[OpenFOAM Foundation](#) Version 6. Expecting 7 in the next month

[OpenFOAM ESI / Plus](#) Version 1906

[Foam Extend](#) the nextRelease (Version 4.1) branch

Older versions as well (check the documentation)

Supported Linux distributions

Tested distros are

CentOS / RedHat Version 6 and 7

Ubuntu The LTS-versions 18.04 and 16.04

ArchLinux / Manjaro Because this is a rolling distro it is not 100% stable

Implemented but not tested in a long time

Fedora last I checked was 25

Darwin Not a Linux. And software installation has to be done by hand

AlpineLinux Not yet completed

Outline

1 Introduction

- This presentation
- Who is this?
- What is your problem?

2 Ansible

- What it is
- How it works

■ Advanced Ansible

3 The "build" script


- Preparation
- Building
- Supported

4 The End

- Controversial topics
- Goodbye



Outline

- 
- 1 Introduction
 - This presentation
 - Who is this?
 - What is your problem?
 - 2 Ansible
 - What it is
 - How it works
 - 3 The "build" script
 - Advanced Ansible
 - Preparation
 - Building
 - Supported
 - 4 The End
 - Controversial topics
 - Goodbye

The user

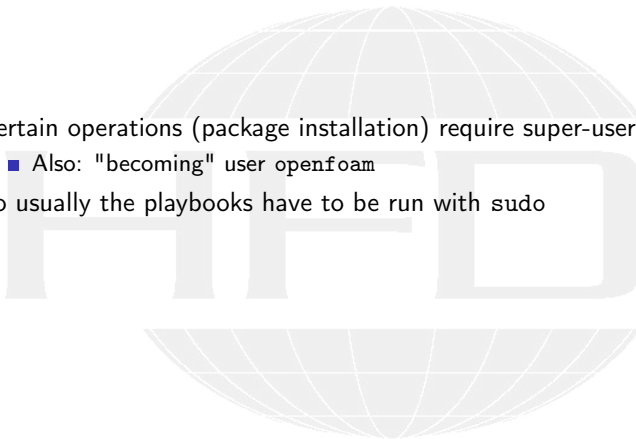
- Some sysadmins think that creating a user is a bit intrusive. So
 - If you're in an environment with a directory service (LDAP etc): create the user there by hand because this Ansible-script will create only local users
 - There is an additional variable for a `openfoam-group`. But that is optional
 - Can be nice if you want all OpenFOAM-users in that group

The location

- With the standard-settings a directory `/home/openfoam/OpenFOAM` is created
 - **Everything** will go into that directory
 - Make sure it is on a drive with sufficient space
- It is recommended to move this to an appropriate location
 - and set a symlink
 - But this has not been "ansibled" because it depends on the specific environment

Superuser rights

- Certain operations (package installation) require super-user rights
 - Also: "becoming" user openfoam
- So usually the playbooks have to be run with sudo



Outline

1 Introduction

- This presentation
- Who is this?
- What is your problem?

2 Ansible

- What it is
- How it works

3 The "build" script

- Preparation
- Building
- Supported

4 The End

- Controversial topics
- Goodbye



Contributions

- Contributions are most welcome
- Especially support of new Distros
 - Also Windows if you want to
- Testing of the unsupported ones
- If somebody wants to write a role to compile ParaView: great

OpenFOAM Workshop

See you in Duisburg later this month (23rd to 26th)



<http://openfoamworkshop.org/>

License of this presentation

This document is licensed under the *Creative Commons Attribution-ShareAlike 3.0 Unported* License (for the full text of the license see

<http://creativecommons.org/licenses/by-sa/3.0/legalcode>).

As long as the terms of the license are met any use of this document is fine (commercial use is explicitly encouraged).

Authors of this document are:

Bernhard F.W. Gschaider original author and responsible for the strange English grammar. Contact him for a copy of the sources if you want to extend/improve/use this presentation