OpenFOAM directory organization

We will use the Linux command `tree` to examine the directory structure:

```
tree -L 1 -d $WM_PROJECT_DIR
```

yielding:

```
$WM_PROJECT_DIR
|-- applications
|-- bin
|-- doc
|-- etc
|-- lib
|-- src
|-- tutorials
`-- wmake
```

In `WM_PROJECT_DIR` you can also find `ReleaseNotes` etc., but most importantly:

```
Allwmake
```

which compiles all of OpenFOAM, as discussed earlier.

The applications directory

```
tree -L 1 -d $WM_PROJECT_DIR/applications
```

yields:

```
$WM_PROJECT_DIR/applications
|-- bin
|-- solvers
|-- test
`-- utilities
```

Here is a short description of the applications directory contents:

- `bin` contains the binaries generated when compiling the applications
- `solvers` contains source code for the distributed solvers
- `test` contains source code that test and show example of the usage of some of the OpenFOAM classes
- `utilities` contains source code for the distributed utilities

There is also an `Allwmake` script, which will compile all the contents of `solvers` and `utilities`

The src directory

This directory contains the source code for all the libraries

It is divided in different subdirectories each of them can contain several libraries

The most relevant are:

- `finiteVolume`. This library provides all the classes needed for the finiteVolume discretization, such as fvMesh, divergence, laplacian, gradient discretization operators, matrix solvers, and boundary conditions.
- `OpenFOAM`. This library includes the definitions of the containers used for the operations, the field definitions, the declaration of the mesh and of all the mesh features such as zones and sets
- `turbulenceModels` which contains several turbulence models
- `engine` declaration of classes for engine simulation
- `dynamicMesh` for moving meshes algorithms

The bin, doc, etc, lib, and tutorials directories

The `bin` directory contains *shell scripts*, such as `paraFoam`, `foamNew`, `foamLog` ...

The `doc` directory contains the documentation of OpenFOAM:

- Programmers and User Guide
- Doxygen generated documentation in html format

Usage:

```
acroread $WM_PROJECT_DIR/doc/Guides-a4/UserGuide.pdf
acroread $WM_PROJECT_DIR/doc/Guides-a4/ProgrammersGuide.pdf
firefox file://$WM_PROJECT_DIR/doc/Doxygen/html/index.html
```

(The Doxygen documentation will not work now since it is not compiled. Instead, have a look at `www.openfoam.org`)

The `etc` directory contains environment set-up files, global OpenFOAM instructions, and default `thermoData`.

The `lib` directory contains the *binaries* of the dynamic libaries.

The `tutorials` directory contains example cases for each solver.

The wmake directory

OpenFOAM uses a special make command: `wmake`.

`wmake` understands the file structure in OpenFOAM and has some default compiler directives that are set in the `wmake` directory. There is also a command, `wclean`, that cleans up (some of) the output from the `wmake` command.

If you added a new compiler name in the `bashrc` file, you should also tell `wmake` how to interpret that name. In `wmake/rules` you find the default settings for the available compilers.

You can also find some scripts that are useful when organizing your files for compilation, or for cleaning up.

User directory organization (1/2)

Some of the OpenFOAM environment is set up for a specific user directory organization, in `$WM_PROJECT_USER_DIR`.

In a clean installation of OpenFOAM you find there two directories.
`tree -L 1 -d $WM_PROJECT_USER_DIR` yields

```
$WM_PROJECT_USER_DIR
|-- applications
`-- lib
```

In `applications`, it is recommended to put user developed applications in the same structure as in `$WM_PROJECT_DIR/applications`

In `$WM_PROJECT_DIR/applications/bin`, the binaries of the user developed applications will be located

In `lib`, the binaries of the user developed libraries will be located

User directory organization (2/2)

It is recommended to create two more directories:

```
$WM_PROJECT_USER_DIR/run
$WM_PROJECT_USER_DIR/src
```

Place user developed library source code in `src` directory, with the same directory structure as in `$FOAM_SRC`, and case files in the `run` directory.