

## 2014.07.14 - PyFoam for Windows

blueCAPE is proud to announce that we have now made available PyFoam for Windows available as a repository on our blueCFD project at Github.

For those unfamiliar with PyFoam, it is a Python library which provides several utilities and functions for managing OpenFOAM cases. Its official page: <http://openfoamwiki.net/index.php/Contrib/PyFoam>

In addition, PyFoam was developed to work on POSIX systems, such as Linux and Mac OS X; unfortunately, Windows is not a fully compliant POSIX system, which means that some of the POSIX-only features used in PyFoam through Python, will not work as intended on Windows.

We at blueCAPE provided the first porting efforts of PyFoam to Windows in blueCFD-Core 2.0-1 back in 2011. Then in January 2013, we provided our changes to the PyFoam project, which were mostly integrated into PyFoam 0.6.0.

The objective of our new repository at Github is to create a development environment for continuing the work that has been done so far:

-

Github's fork-pull request features give a good way of allowing others to contribute changes to the main repository where the port for Windows is being incubated: <https://github.com/blueCFD/PyFoam>

-

The issue tacker has been turned on, for keeping track of issues related to making PyFoam work on Windows: <https://github.com/blueCFD/PyFoam/issues>

-

A release infrastructure, which makes it a bit easier to keep track of the releases made so far: <https://github.com/blueCFD/PyFoam/releases>

The very first release "PyFoam-Windows-0.6.3.tar.gz" is already available on the Releases page. For installing PyFoam 0.6.3 for Windows in blueCFD-Core, simply follow the instructions provided in the file "README.Windows".

For more information about the changes made between the official PyFoam 0.6.3 and this release of PyFoam for Windows, see the respective bug report at PyFoam's bug tracker: <https://sourceforge.net/p/openfoam-extend/ticketspyfoam/166/>