

## 2016.08.11 - Next generation of blueCFD-Core 2016-1 has been released

blueCAPE is proud to officially present blueCFD®-Core 2016-1. For those not familiar, blueCFD-Core is an open source project provided by blueCAPE Lda, which started in 2009. On March 2016 we made our port of OpenFOAM® 2.3.x available for free on our website. And as of the 4th of August 2016, we have taken the next step, namely to release blueCFD-Core as a fully open source project.

This project provides source code and high quality builds of OpenFOAM® technology and related open source projects, as unofficial ports for up-to-date 64 bit Windows 7 to 10. Additional tweaks, executables and scripts are also provided, in order to maintain usability when working within Microsoft Windows Operating Systems.

Contributions and collaborations are very welcome and developments will be kept open and public. For a full Mission Statement, Vision and History of this project, please visit the About page we have placed there.

### Latest News:

-

On the 4th of August 2016, we published the project and the source code in git form for OpenFOAM 4.x.

More details provided here: [blueCFD-Core 2016-1 heralds new release policy](#)

-

On the 11th of August 2016, we have released the first installer for the new project: blueCFD-Core 2016-1, which provides MSys2, OpenFOAM 4.x and ParaView 5.1.2, all 64-bit, compatible with Windows 7 through 10.

More details provided here: [blueCFD-Core 2016-1 \(64-bit\) Installer now available](#)

### Disclaimers:

-

blueCAPE is responsible for the development and maintenance of the blueCFD®-Core project.

-

blueCFD® is a registered trade mark of blueCAPE Lda.

-

This offering is not approved or endorsed by OpenCFD Limited, producer and distributor of the OpenFOAM software and owner of the OPENFOAM® and OpenCFD® trade marks.

-

blueCAPE Lda is a contributor to the OpenFOAM Foundation, but the majority of the specific modifications needed in the source code for running OpenFOAM natively on Windows, will not be accepted in the official releases of OpenFOAM by the OpenFOAM Foundation, mostly due to the maintenance overhead associated to supporting closed-source operating systems.