blueCFD-Core

{tab=Introduction}

blueCAPE is proud to present blueCFD®-Core, an open source project that 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.

The source code for the patched versions are tracked in the blueCFD project at Github.

The lastest release of blueCFD-Core 2017-1 provides:

-

OpenFOAM 5.x working on Windows 7 through 10, all 64-bit;

-

Functionality with the original scripts of OpenFOAM on Windows;

-

The original ParaView 5.4.1 for Windows 64-bit.

-

Both MS-MPI 7.1 (default) and MS-MPI 8.1 support are provided for a multi-core and multi-machine environment on Windows. And it can also be easily built from source code to support Open-MPI, MS-MPI 2008R2, MS-MPI 2012 and MS-MPI 2012R2.

-

All features in OpenFOAM 5.x that require compiling, will build as intended in blueCFD-Core 2017-1.

-

Customized solvers and libraries can also be compiled directly with OpenFOAM 5.x on Windows.

-

MSys2 is included, as it provides an open source software development infrastructure and is more up-to-date than the original MSys project. There are several pre-installed packages, including: GCC 7.2.0, Python 2.7.14, Python 3.6.2, GDB 8.0.1, Gnuplot and so on...

-

Additional third-Party software is also provided, namely Notepad2.

-

A Portable functionality, that allows copying the installed blueCFD-Core into an USB drive and ready to be used in other Windows machines.

-

A public issue tracker (for bug reports and feature requests), for any issues that you may find with our ports and software stack.

-

For full details about our project, please visit blueCFD-Core at GitHub. There you'll also find all of the details you need.

For additional Training and CFD Consultancy work, see our Services page.

The Frequently Asked Questions page should provide answers the more common questions about blueCFD-Core (here).

{tab=How to Get It}

blueCFD-Core is available via one of the following possible ways:

-

You can build yourself the software packages that make up the contents of blueCFD-Core, for which instructions will be referred to at the project's website;

-

You can also install blueCFD-Core 2017-1 for free, by downloading from the blueCFD-Core Download page on our Download Center.

Beyond the software that is provided for free and is completely open-source, we also provide:
   {location:-pt}

-

For 50€/h, we can provide hourly support on issues in installing and using blueCFD-Core, OpenFOAM and CFD.

   {/location} {location:pt}

-

For 50€/h + VAT, we can provide hourly support on issues in installing and using blueCFD-Core, OpenFOAM and CFD.

   {/location}

Further to the above, and as a small company, we will be glad to discuss your project if you see it can benefit from our expertise, as evidenced by the existence of blueCFD-Core:

-

If you want to provide a set of executables from blueCFD-Core for a book, classes, workshops or software distributions that you might be preparing.

-

If you need training in using blueCFD-Core.

-

If you need a customized GUI for using with blueCFD-Core.

For more questions, please refer to our statement of service below and our Services page as well.

   {tab=Statement of Service}

blueCFD®-Core is an open source project and it's provided as a service by blueCAPE to the public community, including complete builds of OpenFOAM® for Windows and documentation. We have decided to do this out of our own need to have a good and reliable compilation of OpenFOAM on Windows 32 and 64-bit versions.

You, the user, can get hold of blueCFD-Core as described in the section "How to Get It".

For those who purchase the hourly support packages, can get support on several topics, such as:

-

Get support on issues related to the blueCFD-Core package;

-

Request updated builds of OpenFOAM;

-

Request new ports or support in installing OpenFOAM and related technology for Windows, e.g. swak4Foam;

-

Request fixes for issues related to compiling - in other words, things that work in Linux, but fail in Windows. Note: some requests may currently be impossible to grant, since there are still some limitations to the porting process.

-

Get support on issues related to the Third-Party software installed with the blueCFD-Core installer or request additional open source software to be packaged with it;

Beyond these, please also refer to the following links for public information about CFD and OpenFOAM:

-

CFD Online, for information on all things CFD

-

CFD Online Forums

-

Official OpenFOAM Forum

-

OpenFOAM Installation on Windows, Mac and other Unsupported Platforms

-

Unofficial OpenFOAM Wiki

If you need more information, please consider the following:

-

contact us if you have questions about building OpenFOAM for Windows;

-

contact forums and/or read wikis for general questions about CFD and/or OpenFOAM;

-

contact us regarding training in CFD and in using blueCFD-Core, as shown in our Services page.

-

contact us regarding CFD consultancy work.

{tab=Disclaimer}

blueCFD® is a registered trademark of blueCAPE Lda. blueCAPE Lda is responsible for blueCFD®-Core and this post of OpenFOAM® for Windows. This offering is not approved or endorsed by OpenCFD Limited, producer and distributor of the OpenFOAM software via www.openfoam.com and owner of the OPENFOAM® and OpenCFD® trade marks. Also, this offering is not approved or endorsed by Kitware, the producer of the ParaView software and owner of ParaView trade marks. Nor does blueCFD-Core have endorsements from any other company.

The blueCFD-Core software package is provided under the GNU Public License (GPL version 3), as are most of the softwares included in it. This includes the NO WARRANTY issue as indicated in the GPL. blueCFD is provided for a small fee if you, the user, decide to receive it via post, since it also includes added value as a service and there are costs that we cannot cover at the moment.

OpenFOAM is free software; you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation; either version 3 of the License, or (at your option) any later version. See here, for a description of the GNU General Public License terms under which you can copy the files.

Trademarks:

-

OPENFOAM® is a registered trade mark of OpenCFD Limited, producer and distributor of the OpenFOAM software via www.openfoam.com.

-

ParaView is a registered trademark of Kitware.

-

Linux is a registered trademark of Linus Torvalds.

-

Cygwin is a registered trademark of Red Hat, Inc.

-

Windows operating systems are registered trademarks of Microsoft.

-

Other softwares included may have their own trademarks, but those references weren't found at the time of the creation of this document.

{/tabs}