Disclaimer

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

Introductory OpenFOAM® Course

From 8th to 12th July, 2013

University of Genoa, DICCA

Dipartimento di **I**ngegneria **C**ivile, **C**himica e **A**mbientale

UNIVERSITÀ DEGLI STUDI
DI GENOVA

Your Lecturer

Joel GUERRERO

joel.guerrero@unige.it

UNIVERSITÀ DEGLI STUDI DI GENOVA

guerrero@wolfdynamics.com

wolf dynamics

Acknowledgements

These slides and the tutorials presented are based upon personal experience, OpenFOAM® source code, OpenFOAM® user guide, OpenFOAM® programmer's guide, and presentations from previous OpenFOAM® training sessions and OpenFOAM® workshops.

We gratefully acknowledge the following OpenFOAM® users for their consent to use their material:

- Hrvoje Jasak. Wikki Ltd.
- Hakan Nilsson. Department of Applied Mechanics, Chalmers University of Technology.
- Eric Paterson. Applied Research Laboratory Professor of Mechanical Engineering, Pennsylvania State University.

Before we begin

Who does not have a working installation of OpenFOAM®?

- Afternoon session will be dedicated to OpenFOAM® installation issues (version 2.2.0 or 2.2.x and version 1.6-ext).
- The easiest way to install OpenFOAM® is by downloading a precompiled binary (Ubuntu Deb Pack and Suse RPM pack are available).
- The ideal way is to compile the latest source code release from the Git repository.
- To install OpenFOAM®, just follow the instructions given in http://www.openfoam.org/ and http://www.extend-project.de/.

Before we begin

Who does not have a working installation of OpenFOAM®?

- If for any reason the instructions given in http://www.openfoam.org/ and http://www.extend-project.de/ do not work for you, I am going to give you mine instructions, they work fine in OpenSUSE 11.4 or newer version.
- It is possible to do a native installation of OpenFOAM® in Windows and Mac OS X, but we will not discuss it.
- For windows and Mac users, I highly recommend to use GeekoCFD and install it using a virtual machine with VirtualBox (https://www.virtualbox.org/).
- GeekoCFD is a linux live distribution based on OpenSUSE – 64 bits, whose purpose is to provide easy and immediate access to open-source Computational Fluid Dynamics tools (including OpenFOAM®).

Before we begin

For those who do not have a working installation of OpenFOAM®, you can remotely run it in dagobah:

- ssh –X –l xxxxxx xxxxxx
- Password: xxxxxx
- cd OpenFOAM
- There are 6 cfd user directories, namely:
  cfd1, cfd2, cfd3, cfd4, cfd5, cfd6.
  Choose one, this will be your personal directory.

Note: the account will be working until Friday.

Note: you can only access dagobah within the department domain (xxxxxx.xxxxxx.xxxxxx).

Before we begin

You can download the course's handouts and tutorials from the following link

www.dicat.unige.it/guerrero/OpenFOAM_course2013.html

www.dicat.unige.it/guerrero/OpenFOAM_course2013.html

You can extract the files wherever you want. However, I highly recommend you to extract them in your OpenFOAM® user directory. From now on, this directory will become **$path_to_openfoamcourse.**

Provisional timetable

	Monday 8th	Tuesday 9th	Wednesday 10th	Thursday 11th	Friday 12th
Morning Session (9:30 am – 12:00 am)	• Before we start - Housekeeping issues. • Introduction - Overview of OpenFOAM®. • Physical models, solvers and utilities. • Library organization and code structure. • Setting cases in OpenFOAM®. Boundary conditions, initial conditions, physical model parameters and solver parameters. • Running my first OpenFOAM® case.	• Basic meshing and mesh conversion in OpenFOAM®. • Geometry generation, mesh generation and post-processing using Open Source tools. • Data analysis, sampling, graphing and post-processing in OpenFOAM®. • Mesh quality assessment. • Hands-on tutorials.	• Basics of C++ programming. • OpenFOAM® library organization, code structure and compilation. • Programming and modifying OpenFOAM® solvers and boundary conditions. • Implementing boundary and initial conditions using external libraries (pyFoam). • Hands-on tutorials.	• Turbulence modeling (RANS and LES). • Advanced Physical Modeling capabilities. • Running in parallel. • Extending OpenFOAM® capabilities. • Getting convergence from scratch. • Hands-on tutorials	• Advanced Physical Modeling capabilities. • Tips & Tricks. • OpenFOAM® extend project. • Wrap-up session. • Open Forum: Questions, doubts and attendees own cases.
Afternoon Session (2:00 pm – 5:30 pm)	**NO LECTURES** • OpenFOAM® installation issues. • Installing additional applications. • Shaking hands and final housekeeping issues. • Open Forum: Questions, doubts and attendees own cases.	• More on geometry generation, mesh generation and post-processing using Open Source tools. • Mesh manipulation and conversion. • Setting boundary and initials conditions. • Setting physical model parameters and solver parameters. • Solution monitoring and control. • Hands-on tutorials.	• Finite Volume Discretization: theoretical background. • Selecting solver parameters. • Controlling solution behavior. • More on post-processing and sampling. • More on mesh conversion and mesh quality related issues. • Hands-on tutorials.	**NO LECTURES** • Open Forum: Questions, doubts and attendees own cases. • Hands-on tutorials.	**NO LECTURES** • Open Forum: Questions, doubts and attendees own cases. • Follow up of first assignment and discussion of final assignment.

Additional Tools

As we will be working with the CFD simulation lifecycle (from geometry generation, to mesh generation, to governing equations solution, to post-processing), by using open source tools, the following additional tools might come in handy:

Before we begin

For geometry generation, mesh generation, and visualization, the following open source applications might come in handy:

CAD - 3D modeling - Surface modeling

- **SALOME** (http://www.salome-platform.org/).
- **Google SketchUp** (http://sketchup.google.com/).
- **Free-CAD** (http://sourceforge.net/apps/mediawiki/free-cad/).
- **Blender** (http://www.blender.org/).

Note: precompiled binaries are available for all these applications.

Note: it is possible to install Google SketchUp in linux by using wine (http://www.winehq.org/).

Before we begin

For geometry generation, mesh generation, and visualization, the following open source applications might come in handy:

Mesh Generation

- **SALOME** (http://www.salome-platform.org/). *
- **ENGRID** (http://engits.eu/en/engrid). *
- **GMSH** (http://www.geuz.org/gmsh/). *
- **Triangle** (http://www.cs.cmu.edu/~quake/triangle.html).
- **Tetgen** (http://tetgen.berlios.de/).
- **Overture** (http://www.overtureframework.org/).

* Note: precompiled binaries are available for these applications.

Before we begin

For geometry generation, mesh generation, and visualization, the following open source applications might come in handy:

Visualization - STL files manipulation

- **SALOME** (http://www.salome-platform.org/).
- **VISIT** (https://wci.llnl.gov/codes/visit/).
- **Paraview** (http://www.paraview.org/).
- **Meshlab** (http://meshlab.sourceforge.net/).
- **Netfabb** (http://www.netfabb.com/).

Note: precompiled binaries are available for all these applications.

Before we begin

For geometry generation, mesh generation, and visualization, the following open source applications might come in handy:

OpenFOAM® GUI

- **Discretizer** (http://www.discretizer.org/).
- **Helyx-os** (http://engys.com/products/helyx-os).

Note: precompiled binaries are available for all these applications.

Before we begin

And of course we will need,

OpenFOAM® solver

- **OpenFOAM®** (http://www.openfoam.org/).

Course objectives

- Introduce the CFD simulation lifecycle by using open source tools.

"*From geometry generation, to mesh generation, to governing equations solution, to post-processing*".

Course objectives

- Introduce OpenFOAM®.
- By the end of the week, help you to become an out-of-the-box user of OpenFOAM®.
- But also, introduce you the building blocks to help you to become an OpenFOAM® user at a developer level (introductory level).
- Empower you to learn more about OpenFOAM®.
- To increase the use of OpenFOAM® in our community.

Prerequisites

- No prior knowledge of OpenFOAM®, C++ or Linux is required, but a basic knowledge of Linux is beneficial.
- A basic knowledge in CFD is also desirably.
- For hands-on examples, you are required to bring your own laptop with a working installation of Linux and OpenFOAM® (version 2.2.0 or 2.2.x preferable).
- For those not able to install OpenFOAM® on their laptops, a session will be dedicated to OpenFOAM® installation issues.
- By the way, I use OpenSUSE Linux distribution.

What I will try to keep to a minimum

- This is not a C++ course, so I will try to keep C++ programming to a minimum.
- This is not a course on Finite Volume Methods - Computational Fluid Dynamics, so I will keep the theory to a minimum.
- This is not a Linux system administration course, so I will try to keep Linux system administration issues to a minimum.
- I am doing this for your own convenience and keeping things easy. Remember, after all this is an introductory course.

What I need from you

- Ask questions (feel free to interrupt me at anytime).
- Tell me if you do not understand.
- Tell me if an example does not work.
- Let me know if you have any specific requirement.
- If you have a case of your own, let me know and I will try to do my best to help you to setup your case. But remember,

<u>the physics is yours</u>.

What I need from you

- Based on this course, I am trying to write some lectures notes on CFD and related topics, a help is needed and much appreciated. To help me, take a look at the lectures notes and let me know if you find errors. Suggestions for better wording, figures or new material are also welcome.
- The lectures notes are available with the course's material.
- Follow-up problems, questions and suggestions at joel.guerrero@unige.it.

How to learn more after this course

- Learn by doing.
- User manual, programmer manual and source code.
- The Doxygen manual (http://www.openfoam.org/docs/cpp/).
- CFD-Online OpenFOAM® user discussion group (http://www.cfd-online.com/Forums/openfoam/).
- OpenFOAM® wiki (http://www.openfoamwiki.net).
- OpenFOAM® website (http://www.openfoam.com/).
- OpenFOAM® extend project (http://www.extend-project.de).

Where can I get Help?

The lack of maintained documentation makes it difficult for new users.

The user and programmer's guide do not provide much details, making the learning curve steep.

But before complaining about the lack of documentation,

Where can I get Help?

… read carefully and digest all the information contained in the user guide and programmer's guide. It might not be much, but it is enough to get you started.

Try to do all the tutorials available in the OpenFOAM® installation (or at least those that interest you), and dig into each one to learn more about all the applications and utilities available (that was how I managed to learn OpenFOAM®).

Where can I get Help?

Remember,

You have the source code so take some time and explore it.

Where can I get Help?

OpenFOAM® Internet Resources

- OpenFOAM® web pages (http://www.openfoam.org).
- OpenFOAM® user discussion group (http://www.cfd-online.com/Forums/openfoam/).
- OpenFOAM® community pages (Wiki) (http://www.openfoamwiki.net).
- OpenFOAM® research resources (news, presentation slides, papers, running projects, user contributions) (http://www.foamcfd.org).

CFD Resources on the Internet

- There exists a number of CFD discussion sites, depending on your interest feel free to explore or join them.
- General CFD discussion, commercial software discussion forum, popular jobs database (http://www.cfd-online.com).

CFD/FVM/C++ Bibliographical references

- **An Introduction to Computational Fluid Dynamics.**
  H. K. Versteeg, W. Malalasekera. 2007, Prentice Hall.

- **Computational Methods for Fluid Dynamics.**
  J. H. Ferziger, M. Peric. 2001, Springer.

- **Computational Fluid Dynamics: Principles and Applications.**
  J. Blazek. 2006, Elsevier Science.

- **Numerical Heat Transfer and Fluid Flow.**
  S. Patankar. 1980, Taylor & Francis

- **Error analysis and estimation in the Finite Volume method with applications to fluid flows.**
  H. Jasak. PhD Thesis, 1996. Imperial College, London.

- **A Finite Volume Method for the Prediction of Three-Dimensional Fluid Flow in Complex Ducts.**
  M. Peric. PhD Thesis, 1985. Imperial College, London.

CFD/FVM/C++ Bibliographical references

- **The C++ Programming Language.**
  B. Stroustrup. 2013, Addison-Wesley.

- **The C++ Standard Library**.
  N. Josuttis. 2012, Addison-Wesley.

- **C++ for Engineers and Scientists.**
  G. J. Bronson. 2012, Cengage Learning.

- **Sams Teach Yourself C++ in One Hour a Day.**
  S. Rao. 2012, Sams Publishing.

- **Sams Teach Yourself C++ in One Hour a Day.**
  J. Liberty, B. Jones. 2004, Sams Publishing.

- **C++ Primer.**
  S. Lippman, J. Lajoie, B. Moo. 2012, Addison-Wesley.

- http://www.cplusplus.com/

Thank you for your attention