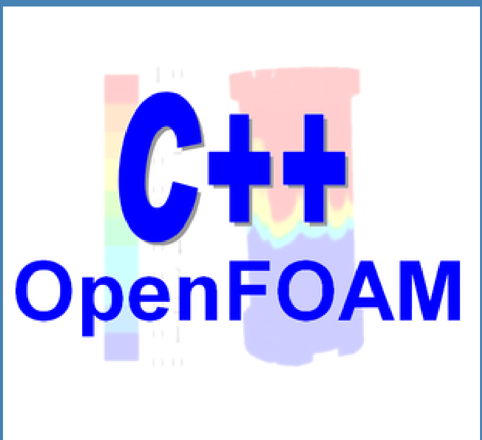
Online course on C++ applied to OpenFOAM

Technical Courses
Cursos Técnicos

www.technicalcourses.net
info@technicalcourses.net
+34 604859663

OpenFOAM (Open Field Operation and Manipulation) is a free, open-source CFD (Computational Fluid Dynamics) software. It has a large number of users in most areas of engineering and science, in both commercial and academic organisations.



This course has been developed entirely by Technical Courses and is not offered by OpenCFD Limited, the producer of the OpenFOAM software and owner of the trademarks OPENFOAM® and OpenCFD®

Aimed at those interested in C++ development for OpenFOAM management. Students with no prior experience in C++ will receive additional material to acquire a basic level of knowledge and thus be able to follow the course.

A basic understanding of the OpenFOAM solver is required to take this course. For those with no prior knowledge of OpenFOAM, we recommend first taking the CFD with OpenFOAM course, also offered by Technical Courses.

The aim of the course is to acquire the knowledge and practical skills necessary to understand OpenFOAM solvers and modify them to suit your needs.

It includes PDF manuals and exercises. It is estimated that students must dedicate at least 25 hours to complete the course. The maximum time to complete the course is two months. Students who pass the course will receive a diploma.

Our online platform offers various resources such as chat, forums, messaging, videoconferencing, etc. The teachers (M.I. Lamas and C.G. Rodríguez) have professional and accredited teaching experience in CFD and OpenFOAM.

The course material is entirely in English. However, consultations with teachers can be conducted in English or Spanish.

Online course on C++ applied to OpenFOAM

Chapter 1: Introduction to C++. Application to OpenFOAM

1.1 Introduction 1.2 C++
1.3 Basic structure of a C++ program.
1.4 Preprocessor directives.
1.5 Variables.
1.6 Operators.
1.7 Inputs and outputs.
1.8 Control structures.

Chapter 2: C++. Application to OpenFOAM

2.1 Introduction.
2.2 Typedefs.
2.3 Functions.
2.4 Pointers.
2.5 Data structures.
2.6 Classes.
2.7 Constructors.
2.8 Destructors.
2.9 Friends.
2.10 Inheritance.
2.11 Virtual member functions.
2.12 Abstract classes.
2.13 Templates.
2.14 Namespaces.
2.15 Solving partial differential equations in OpenFOAM.
2.16 Programation in OpenFOAM.

Chapter 3: Development of own code in OpenFOAM

3.1 Development of a new solver.
3.2 Compilation of applications and libraries.
3.3 Development of a new boundary condition.
3.4 Development of a new turbulence model.
3.5 Development of a new transport model.
3.6 Development of a new thermophysical model.
3.7 Development of a new postprocessing utility.

EXERCISES

1 C++ program 1 - 2 C++ program 2 - 3 C++ program 3 - 4 C++ program 4 - 5 C++ program 5