# Coupling methodology of 1D finite difference and 3D finite volume CFD codes based on the Method of Characteristics

J. Galindo, A. Tiseira, P. Fajardo *, R. Navarro

*CMT - Motores Térmicos, Universidad Politécnica de Valencia, Camino de Vera S/N, 46022 Valencia, Spain*

## ARTICLE INFO

## ABSTRACT

This paper describes the methodology followed to perform a co-simulation between 1D (OpenWAM) and 3D (FLUENT) CFD codes. The Method of Characteristics (MoC) has been chosen to transfer the information between the two domains by properly updating the boundary condition at the shared interface. A short explanation of the MoC is provided, including the modifications needed by the Riemann invariants when dealing with non-homentropic flow. The implementation of the coupling is explained, focusing on the particular approach required by FLUENT in order to obtain the Riemann invariants. Two validation tests have been performed. The Sod's problem has been used to test the numerical accuracy of the coupling methodology. On the other hand, an impulse test rig configuration has been simulated to show the potential capability of a co-simulation in terms of reducing the computational cost. In both cases a good agreement in the solution is found.

## 1. Introduction

In the last decades, the legislation on internal combustion engines (ICEs) has severely reduced the limits for pollutant and noise emissions. These requirements have established the research activity at design phase as a key stage in the engine production process. Therefore, an intensive investigation on ICEs has been carried out, focusing on the optimization of performances and fuel consumption. In particular, an important effort has been done seeking the improvement of the combustion and gas exchange processes, using tools such as Computational Fluid Dynamics (CFD).

CFD simulations allow researchers to understand flow behavior and quantify important flow parameters such as mass flow rates or pressure drops, provided that the CFD tools had been properly validated against experimental results. For reasons such as the aforementioned, CFD simulations have become a valuable tool in helping both the analysis and design of the intake and exhaust systems of an ICE.

It is important to highlight that the accuracy of a CFD simulation relays heavily on the assumptions made. Generally, the less restrictive are the hypothesis made, the more accurate the results will be. However, assumptions can imply a significant reduction in the time consumption without a loss in the accuracy, as long as the physical problem that is being modeled meets the requirements made.

Simulating an intake or exhaust system is just a great exponent of this sort of problems. These systems are mainly composed of ducts, which can be accurately simulated by means of one-dimensional, non-viscous codes. However, there are several components that manifest a complex three-dimensional flow behavior, such as turbomachinery or manifolds, therefore being unable to be simulated properly by 1D codes, and thus requiring viscous, 3D codes.

---

* Corresponding author. Tel.: +34 963877650; fax: +34 963877659.
*E-mail addresses:* galindo@mot.upv.es (J. Galindo), anti1@mot.upv.es (A. Tiseira), pabfape@mot.upv.es (P. Fajardo), ronagar1@mot.upv.es (R. Navarro).