

A 1D/Quasi-3D Coupled Model for the Simulation of I.C. Engines: Development and Application of an Automatic Cell-Network Generator

Augusto Della Torre, Gianluca Montenegro, Tarcisio Cerri, and Angelo Onorati
 Politecnico di Milano

ABSTRACT

Nowadays quasi-3D approaches are included in many commercial and research 1D numerical codes, in order to increase their simulation accuracy in presence of complex shape 3D volumes, e.g. plenums and silencers. In particular, these are regarded as valuable approaches for application during the design phase of an engine, for their capability of predicting non-planar waves motion and, on the other hand, for their low requirements in terms of computational runtime. However, the generation of a high-quality quasi-3D computational grid is not always straightforward, especially in case of complex elements, and can be a time-consuming operation, making the quasi-3D tool a less attractive option. In this work, a quasi-3D module has been implemented on the basis of the open-source CFD code OpenFOAM and coupled with the 1D code GASEDYN. The flexible OpenFOAM platform has been exploited for the development of both the quasi-3D cell-network generator and the implementation of the non-linear explicit quasi-3D solver. The grid generator is based on a fully automatic strategy, which takes as input the definition of the geometry in STL format. It works in two steps, firstly creating a refined background mesh, including all the small details of the geometry, and then agglomerating the cells, reducing the number of elements, in order to reach the desired cell density. The implemented quasi-3D solver, relies on a high accuracy numerical method specifically developed for working on such a coarse grid and it is adopted for the solution of the 3D domain. The coupled 1D/quasi-3D simulation tool has been firstly validated considering different silencer configurations, comparing the computed Transmission Loss with experimental values. Then, as further step of the validation, an entire engine configuration, including complex intake and exhaust systems, has been simulated. Furthermore, the performances of the quasi-3D numerical method are compared with those of a standard CFD solver, with particular focus on the numerical accuracy and computational burden of the two approaches.

CITATION: Della Torre, A., Montenegro, G., Cerri, T., and Onorati, A., "A 1D/Quasi-3D Coupled Model for the Simulation of I.C. Engines: Development and Application of an Automatic Cell-Network Generator," *SAE Int. J. Engines* 10(2):2017, doi:10.4271/2017-01-0514.

INTRODUCTION

The performances of an internal combustion engine are usually evaluated on the basis of global parameters such as power, torque, fuel consumption, pollutant and acoustic emissions. These depend to some extent on the performances of the single stand-alone components of the engine, but above all are influenced by how the single components are assembled to form the system. For this reason, when engine optimization is performed in a CAE environment, the numerical simulation needs to be extended to the entire engine system, comprising cylinders, pipe system, plenums, silencers, after-treatment devices. The main constraint for an entire engine simulation is the computational burden, therefore suitable modeling approaches have to be adopted in order to balance accuracy and simulation runtime, i.e. introducing suitable simplification in relation with the level of detail required by the analysis [1, 2]. In order to simplify the problem, making it possible the simulation of the entire system, the 1D approximation has been traditionally introduced. In this framework, the modeling of the complex shape devices (e.g. air-boxes, plenums, silencers, etc) is addressed resorting to simplified approaches, namely 0D approximation and equivalent 1D pipe schemes [3, 4]. These approaches, although simplified, generally

allow, after a process of refinement, calibration and improvement, to build a reliable model of the engine, which can be applied for the investigation of the performance of the engine under different operating conditions and for the optimization of the pipe system.

Nevertheless, when the analysis is focused on the investigation of the effect of the geometry of the complex shape 3D components on the engine performances, geometry-based three-dimensional approaches are needed. For this reason, in recent years, quasi-3D or fully-3D approaches have been largely investigated [5, 6, 7, 8] and progressively coupled with the main commercial and research 1D numerical codes [9, 10, 11, 12]. In particular, for application during the design phase of the engine, simplified quasi-3D methods have been demonstrated to be valuable approaches for the prediction of the non-planar waves motion, which is marginally influenced by the geometric details but it is responsible for most of the acoustic behavior and the engine performances. These approaches can be suitably exploited for the acoustic optimization of the intake and exhaust silencers, in order to find the best internal layout of the component which maximizes the noise abatement over a specific frequency range under real operating conditions, as described in [2].

On the other hand, for a deeper optimization of the geometrical details of the component, mainly finalized to the refinement of the solution from a fluid-dynamic point of view, the integration with fully-3D CFD tools becomes mandatory [13, 5]. The fact remains that, in case of 1D/3D analysis, the requirement in terms of computational runtime becomes significant, around 10-50 times that of an equivalent coupled 1D/quasi-3D investigation.

However, in case of the adoption of a quasi-3D model, the generation of the computational grid can be a time-consuming operation, making the quasi-3D tool a less attractive option since the gain in terms of lower computational burden is often counterbalanced by the longer time required for the setup of the model. This aspect is especially crucial when the method is applied in an industrial framework, where the reduction of the time-to-solution is a key factor. For this reason, in this work a fully automatic grid generator for quasi-3D model has been implemented, in order to accomplish the requirements of high quality of the mesh, reduction of the mesh generation time and automation of the process. Moreover, in order to evaluate the numerical performances of the quasi-3D method, in terms of accuracy, computational time and stability, the numerical method has been compared with a standard CFD solver. In order to directly compare their performances, the quasi-3D model was implemented on the same numerical platform of the CFD solver and the methods were applied to the solution of a set of experimental test cases, employing the same computational grid for the discretization of the domain.

In the following sections, at first the simulation methodology will be presented, highlighting the details of the implementation of the quasi-3D numerical method. Then, the implementation of the quasi-3D grid generator will be described. Finally, different test cases will be proposed, in order to assess the validation of the methodology on both purely acoustic cases and a more complex engine configuration.

SIMULATION METHODOLOGY

The simulation of the acoustic and fluid-dynamic phenomena occurring in the intake and exhaust system is a complex task, since requires an accurate description of the pulsating flow through the ducts and of the unsteady propagation of pressure waves. For this reason, the formulation of the computational models commonly adopted for the simulation of the intake and exhaust systems (being 1D, quasi-3D or fully-3D) relies on a non-linear time-dependent set of governing equations, which can be properly simplified depending on the level of approximation which is required for the particular study. Moreover, when a numerical simulation is performed, a key factor to be considered is the balance between accuracy and computational runtime, which basically depends on: a) the approximations introduced at the level of the physical modelling (e.g. simplification of the governing equations), b) the approximations introduced at the level of the geometrical modelling (e.g. simplification of the small details of the geometry) and c) the accuracy of the numerical methods applied. In the following sections, these aspects will be considered in order to describe the hypothesis at the basis of the implementation of fully-3D and quasi-3D models and to discuss the fields of applicability of the two simulation approaches when coupled to a 1D model of the engine.

Governing Equations

In order to describe the finite-amplitude pressure wave motion in the fluid domain, the compressibility of the medium must be taken into account. The general set of governing equations describing the motion of a compressible fluid is represented by the following [14]:

- Continuity equation, which states the conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \quad (1)$$

- Navier-Stokes equation, derived from the conservation of momentum:

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) = \rho \mathbf{g} - \nabla \cdot (\boldsymbol{\sigma}) \quad (2)$$

- Energy equation:

$$\begin{aligned} \frac{\partial \rho e}{\partial t} + \nabla \cdot (\rho e \mathbf{U}) &= \rho \mathbf{g} \mathbf{U} - \nabla \cdot (\boldsymbol{\sigma} \mathbf{U}) \\ &+ \nabla \cdot (\lambda \nabla T) + \rho Q \end{aligned} \quad (3)$$

The constitutive equation of the medium can be expressed, under the assumption of Newtonian fluid, as:

$$\boldsymbol{\sigma} = \left(p + \frac{2}{3} \mu \nabla \cdot \mathbf{U} \right) \mathbf{I} - \mu (\nabla \mathbf{U} + \nabla \mathbf{U}^T) \quad (4)$$

This set of equations is closed introducing the equation of state of the gas; for example, in case of the assumption of perfect gas:

$$\frac{p}{\rho} = RT. \quad (5)$$

Moreover, when turbulence phenomena become significant, the momentum equation Eqn. 2 can be reformulated in terms of Reynolds Average Navier Stokes equation (RANS) and proper turbulence models must be introduced for the calculation of the turbulent viscosity [14].

Basic Finite Volume Collocated CFD Solver

In this work, the open-source CFD code OpenFOAM [15, 16] has been applied for the solution of the full set of the unsteady Reynolds-Averaged Navier-Stokes equations, in order to address fully-3D simulation of the fluid flow in the system. OpenFOAM solvers are based on a finite-volume method, operating on a collocated computational grid, in which all the fluid-dynamic and thermo-physical properties are calculated in the cells while the fluxes are evaluated on the faces via interpolation from the cell quantities. The set of equations 1–3 are solved subsequently by means of a segregated solution procedure, employing a standard PISO algorithm to ensure pressure-velocity coupling [17].

The main purposes for the application of a fully-3D approach for the simulation of a 3D component included in a 1D engine schematic can be summarized in two points: a) to predict the complex three-dimensional flow and wave motion, which strongly affects the tailpipe noise and may also influence the volumetric efficiency, especially in cases of significant wave interaction between the cylinders and b) to provide an accurate description of the viscous and turbulent effects, which determines, along with inertial effects, the pressure loss through the component, which ultimately affects the engine performances.

From these objectives, particular requirements descend for the simulation setup. In particular, in order to obtain an accurate description of the wave motion inside the system, a second order discretization in both space and time should be preferred. Hence, in this work, linear face cell interpolation was used to determine the face center value of the conserved variables for the calculation of the fluxes while an implicit Crank-Nicholson marching procedure was adopted for the time discretization. On the other hand, to allow an accurate prediction of the pressure losses introduced by the system, it is necessary to generate an high quality mesh, with a good resolution in all the domain and especially at the wall boundary. However, in this work, the latter requirement will be disregarded, since the CFD solver will be applied on the same coarse mesh generated for the quasi-3D numerical approach, in order to directly compare the performances of the numerical approaches with regards to a quasi-3D analysis.

Implementation of the Staggered Quasi-3D Solver

The quasi-3D approach applied in this work, named 3Dcell, has been implemented as additional solver inside the Open-FOAM code. In this case, the assumption of inviscid flow has been considered as reasonable and the formulation of the non-linear set of Euler equations is adopted, taking into account that, at the high Reynolds numbers, typical of silencers along the intake and exhaust systems of I.C. engines, the problem is dominated by convection, making viscous phenomena negligible [18].

The approach has been extensively described in previous papers by the authors [11, 1]. In brief, the 3Dcell approach is conceived in order to reconstruct the complex 3D shapes by means of a low number of cells, in such a way to reproduce the main characteristic lengths / volumes of the 3D geometry and to neglect, on the other hand, the small geometrical details. The purpose of the method is: a) to accurately predict the three-dimensional wave motion inside the component, which is determined by the main lengths of the geometry and it is marginally influenced by the small geometrical details and b) to keep as low as possible the computational burden of the simulation, by means of the reduction of the number of cells obtain adopting a very coarse spatial discretization.

In order to fulfill these requirements, the numerical method operating on such a grid needs to be characterized by high accuracy, low numerical diffusivity and high stability. These objectives have been reached with the adoption of a staggered computational arrangement for the spatial distribution on the grid of the control volumes used for the solution of the equations (Figure 1). In this framework, the intensive properties such as pressure, temperature, density, internal

energy and other derived quantities are evaluated in the cell control volumes while velocity and momentum are defined in the face ones. This computational arrangement is applied in combination with an explicit time marching leapfrog method, providing a second order accuracy in both space and time.

The need to adopt two different set of control volumes for the solution of the conservation equations, according to the staggered arrangement (Figure 1), has required the implementation of a particular treatment for the solution of the momentum equation. In particular, according to the 3Dcell definition, the solution of the momentum equation is addressed on the face control volume, which is defined by the face surface S and the distance ΔL between the two neighboring cell centers. On this control volume, the mass flow $\rho U_n S$ through the face is determined solving a 1D momentum equation along the direction x_n normal to the face, leading to:

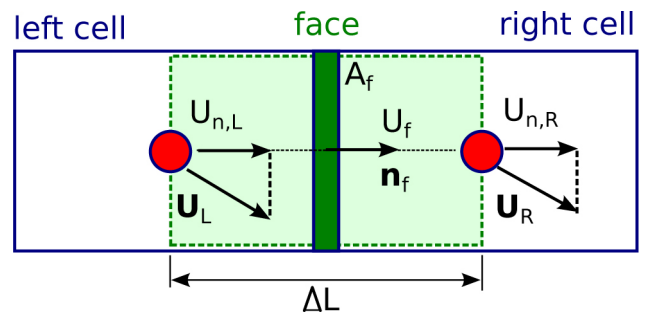


Figure 1. Schematic of the control volumes adopted for the solution of the equations according to the staggered arrangement: circles (red) represents two neighbor cells, rectangle (green) represents the face. Momentum equation is solved on the face control volume (green, dotted line) while mass and energy on the cell control volume (blue, continuous line).

$$(\rho U_n S)_f^{n+\frac{1}{2}} = (\rho U_n S)_f^{n-\frac{1}{2}} + S \frac{\Delta t}{\Delta L} \left[(\rho U_n^2 + p)_L^n - (\rho U_n^2 + p)_R^n \right], \quad (6)$$

where the subscripts L and R indicate the quantities in the left and right cells respectively. The momentum in the cell is then determined by the momentums of all the ports linked to the cell, weighted by the their distance from the cell center:

$$(\rho \mathbf{U} S)_c = \sum_{f=1}^{N_f} (\rho \mathbf{U} S)_f \Delta L_{L/R} \cdot \mathbf{n}_f. \quad (7)$$

The other conservation equations, namely mass and energy, are solved on the standard cell control volumes and, therefore, they do not need any particular treatment with respect to the standard collocated procedure implemented in Open-FOAM. According to the staggered leapfrog solution procedure, the mass flux needed for the solution of the mass conservation equation is directly provided by the solution of the momentum equation on the faces (Eqn. 6). Figure 2 reports the time evolution of the solution on a space-time diagram, illustrating how the staggered leapfrog method is implemented: the face control volumes and the cell control volumes are centered at

different locations both in the scale of space and time. In this way the fluxes calculated in the faces at a generic time step $n + 1/2$ can be used to update the solution in the cells from time step n to time step $n+1$. With regards to the energy equation, the total enthalpy flux is computed starting from the mass flow and the enthalpy value interpolated at the face location.

The base numerical method, being second order accurate, does not satisfy the stability requirements and therefore, in case of the simulation of pulsating flows or high-amplitude pressure perturbations, can give rise to unphysical spurious oscillations affecting the numerical solution. In order to stabilize the method, a diffusion source term is added on the momentum equation (DTM) according to the formulation reported in [11]. The DTM method operates on the momentum equation applying a diffusive flux in order to counterbalance the anti-diffusive flux introduced by the truncation error, responsible for the instability of the scheme. From an implementation point of view, this has required the addition of a source term on the momentum equation Eqn. 6, which value is determined on the basis of the gradients of the momentum solution evaluated on the cells. On the other hand, the approach does not require any modification of the collocated mass and energy conservation equations with respect to the standard formulation.

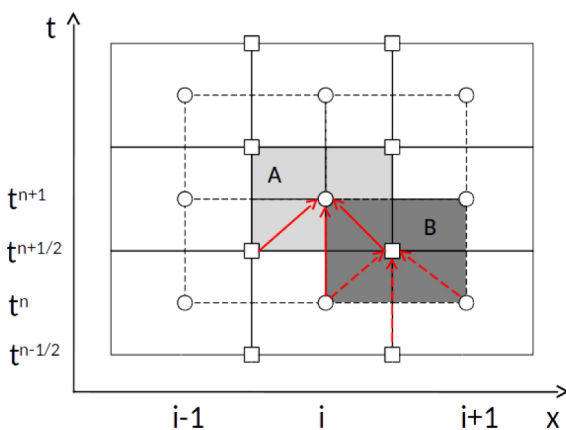


Figure 2. Time marching procedure adopted for the solution of the governing equations according to the staggered leapfrog method: the circles (A) represents the cells (control volumes for the mass and energy solution) and the square (B) represents the faces (control volumes for the momentum solution).

Coupling between Fully-3D / Quasi-3D and the 1D Model

The fully-3D and quasi-3D OpenFOAM models are coupled to the 1D code GASDYN [19], in order to address the simulation of the entire engine system. The strategy adopted for the coupling of the two codes is described in [20, 1]. It is based on the solution of multiple local Riemann problems, each one centered on a face constituting the boundary patch which interfaces the 1D domain with the 3D one. This approach allows to locate the 1D-3D interface even in regions close to highly 3D shapes, therefore allowing the treatment of non-uniform fields on the 3D side without compromising the stability of the solution.

GENERATION OF THE COMPUTATIONAL GRID

Nowadays, the generation of the computational grid for a fully-3D CFD calculation has become a straightforward operation. A wide range of tools, both commercial and open-source, are available, providing different strategies for the generation of the grid, with different levels of automation of the process, ranging from completely user-dependent to fully-automatic. On the other hand, with regards to the generation of quasi-3D computational grid, the range of tools applied in the literature [9] is more restricted and, therefore, it seems that there is space for further improvements of the available approaches. In particular, the most crucial aspects to address when a quasi-3D network is generated are: a) the correct description of the main acoustic lengths, which can be compromised by an excessive geometrical simplification; b) the correct description of the volumes and c) the fact that the procedure cannot be easily automatized and is usually strongly relying on the user expertise. In the following section, an approach based on a fully automatic methodology is proposed in order to address these issues of the generation of the quasi-3D computational grid.

Basic Automatic Mesh Generator for Fully-3D Calculation

The developed procedure is based on the standard mesh generator tool included in the OpenFOAM code. This mesh generator, named snappyHexMesh, implements an automatic procedure to create a grid which is predominantly hexaedral, with a small percentage of polyhedral elements mainly located near the boundaries of the computational domain. The basic procedure takes as input the definition of the surface geometry in STL format and consists of three steps, as shown in Figure 3: a) generation of a cartesian background mesh; b) refinement of the grid near the boundaries, on the basis of an hex-splitting algorithm, and removal of the external domain; c) conformation of the external boundary of the mesh to the STL geometry surface, by means of a iterative boundary adaptation procedure. These steps can be eventually followed by the final addition of boundary layers near the walls, in order to provide a better discretization of the velocity profile at the wall.

Implementation of an Automatic Quasi-3D Cell Network Generator

The quasi-3D cell network is generated starting from a base snappyHexMesh grid by means of a cell agglomeration procedure. The aim of the procedure is to operate over a refined computational grid (e.g. a grid generated for fully-3D calculation), which accurately reproduces the geometry of the domain and, therefore, provides a high-resolution description of the characteristic lengths and volume of the domain. On the basis of this detailed grid, the quasi-3D cell network is generated, by means of a progressive agglomeration of the cells, in such a way as to reduce the number of cells without, on the other hand, simplifying the description of the geometry at the boundaries.

The agglomeration procedure consists of different steps. At first, a first-level cell agglomeration is performed on the basis of a background agglomeration grid (Figure 3-d). The agglomeration grid