International Multidimensional Engine Modeling User's Group Meeting at the SAE Congress April 12, 2010, Detroit MI, USA

Flexible Meshing Process and Multi-cycle Methodology for Simulating Reacting Flow in SI High Performance Engine with ANSYS CFX

Stefano Toninel¹, Martin Kuntz, Thomas Frank

ANSYS Germany GmbH

Gian Marco Bianchi

University of Bologna, DIEM

Simone Di Piazza, Massimo Rosso

Ducati Motor Holding

Copyright © 2010 ANSYS Inc.

Introduction

The investigation and optimization of the combustion process by means of Computational Fluid Dynamics (CFD) may play a critical role in the design of Internal Combustion Engines (ICEs), providing the manufacturer with a better insight of the complex phenomena involving turbulent reacting flows and allowing an efficient virtual prototyping covering all phases of a full engine cycle.

The success of applying a given combustion model in an engine simulation depends on two main factors. The first one is related to the distinctive features of the model itself. The second one deals with the input data fed into the model, which eventually works for the user as a black-box system. Even a complex model, plenty of details concerning the description of physical phenomena, will return an inaccurate prediction if the wrong turbulent flow field in the combustion chamber is used as initial guess. The quality of the available initial conditions is varying on a wide range. Research engines with optical access sometimes offer a detailed description of the velocity field and turbulence quantities at spark ignition, but this option is of course not always available for production engines or expensive prototypes, like in racing applications.

High performance engines present such challenging working conditions, in terms of gas-dynamics and turbulence regimes, that the flow field initialization at Spark Ignition (SI) cannot rely on basic assumptions but rather has to be defined by means of accurate full cycle simulations. The setup complexity (engine model size, boundary conditions, etc.) may vary arbitrarily according to a trade-off between accuracy and computational costs. One popular approach deals with the use of intake and exhaust ducts whose length is limited within few diameters away from the cylinder. According to this choice, boundary conditions derived from one-dimensional simulations are applied. This method offers the clear advantage of limiting the number of mesh elements for a given spatial discretization. However, the enforcement of one-dimensional boundary conditions, especially at the domain inlet, might affect the in-cylinder flow initialization thus biasing the results. A more comprehensive approach deals with the multi-cycle engine simulation where the one-dimensional boundary conditions at the intake runner are eliminated. The flow naturally develops and a three-dimensional wave reflection is achieved at the runner inlet sections. As a result, the accuracy of the in-cylinder flow field initialization for a combustion analysis is greatly increased by reducing the risk of an ill-conditioning at boundary inlet section.

In this work an innovative multi-cycle simulation methodology is presented which allows initializing the in-cylinder flow field for a combustion analysis without applying one-dimensional boundary conditions at the intake inlet, thus minimizing the risk of an ill-conditioning. The simulations were carried out with the ANSYS CFX software, according to a new workflow specifically tailored for ICE application and having distinguishing features in terms of physical models and mesh generation, able to cope with the complexity of the task.

¹ Corresponding Author. E-mail: <u>stefano.toninel@ansys.com</u> Tel: +49 8024 9054 88

Test Case Description

The new simulation approach was applied to the Port Fuel Injection (PFI) engine equipping the Ducati 848 motorbike whose main geometrical features and maximum performance are summarized in Table 1.

Layout	L-Twin
Total Displaced Volume	848 cm ³
Bore	94 mm
Stroke	61.2 mm
Compression Ratio	12.9
Max Power	98.5 kW (134 CV) @ 10,000 rpm
Max Torque	96 Nm (9.8 kgm) @ 8,250 rpm

Table 1 - Ducati 848 engine, geometry and performance

In particular, in this work the maximum power operating point was investigated. Some remarkable data reported in Table 1 like the compression ratio (12.9), the maximum power rotating speed (10,000 rpm) and the maximum specific power (116 kW / 1, 158 CV/l) are indices of very challenging working conditions to be investigated by means of CFD.

Simulation Methodology

Figure 1 shows a reduced CAD model of the ports-valves-cylinder assembly for the Ducati 848 engine, where the intake (on the left) and exhaust (on the right) ducts have been cut above the valve-stem insertions. Because of symmetry reasons, only one half of the model was considered. The charge motion during the full engine cycle could be simulated on this geometrical configuration by applying on the inlet and outlet sections boundary conditions from one-dimensional system-simulations. However, in order to avoid a possible ill-conditioning on the intake side, as explained in the introductory section, the computational domain was extended as depicted in Figure 2, by including the whole intake port up to the trumpet, the throttle body and a plenum mimicking the air-box volume.





Figure 1- Reduced CAD Model

Figure 2 – Extended CAD Model

The overall methodology is split in three steps, each corresponding to a stand-alone run: exhaust, multi-cycle intake and combustion. Each one consists of one or more *configurations*. The latter is used in order to identify a specific engine mesh topology and the related simulation setup, i.e. a proper set of physical models, boundary and initial conditions.

While the combustion phase is covered by one single configuration, from the Intake Valve Closing (IVC) to the Exhaust Valve Opening (EVO), the exhaust simulation is structured in two configurations, illustrated at the bottom of Figure 3. In particular, the C1BC configuration is started a few crank angle degrees before the EVO, just in order to move the charge in

the combustion chamber according to the piston displacement. Then, it is followed by the C2EO configuration where the transonic flow through the exhaust valve is computed, up to the IVC.



Figure 3 – Intake (top) and exhaust (bottom) configurations of the multi-cycle initialization

The exhaust simulation is part of the simulation process required for providing a proper initialization of the flow field in the cylinder and exhaust port at the Intake Valve Opening (IVO), to be used at the beginning of the second configuration for the intake stroke simulation (Figure 3, on the top). During this second methodology step the whole intake system is always activated. A constant pressure of 1 atm is applied at the top of the plenum, consistently with the actual conditions at the inlet of the motorbike air-box. Pressure waves are thus generated and propagated through the duct by the interaction with the engine moving walls and the subdomains activated and deactivated during the C2BO and C3IO configurations. Several consecutive cycles have to be computed in order to let the flow self-develop and adjust, reaching a cycle-to-cycle convergence, depending on the engine under investigation. The success of such an approach requires a flexible setup flexible setup, allowing the combination of arbitrarily complex configurations, and for a reliable meshing process. The former requisite is fulfilled by some ICE-specific extensions of ANSYS CFX-Pre which simplify and speed-up the pre-processing of the engine simulation. The latter requirement is satisfied by a flexible, robust and automatic meshing process based on ANSYS ICEM-CFD.

Mixture Composition and Reacting Flow Modeling

All simulations were carried out by selecting ANSYS CFX framework for fully/partially premixed combustion with exhaust gas recirculation [1]. The reacting flow is modeled as a three-stream mixture, made of fuel, oxidizer and residual gases. The EGR is assumed to be stoichiometric. Four variables are required for describing this system: the mixture fraction mean value \tilde{Z} and the mass fractions (or tracers) of fuel, oxidizer and residual gases, respectively \tilde{Tr}_F , \tilde{Tr}_O , \tilde{Tr}_{EGR} . The problem is closed by solving two transport equations, for \tilde{Z} and \tilde{Tr}_F plus two additional algebraic constraints:

$$\widetilde{Tr_F} + \widetilde{Tr_O} + \widetilde{Tr_{EGR}} = 1$$
$$\widetilde{Z} = \widetilde{Tr_F} + Z_{st}\widetilde{Tr_{EGR}}$$

The Favre-averaged description of the mixture fraction is completed by solving a transport equation for its variance $\widetilde{Z''^2}$. Figure 4 shows how of the composition and the thermo-dynamical status of the reacting mixture is computed. The mean value of the reaction progress \tilde{c} , representing the probability of finding burnt gases, is evaluated according to the combustion model chosen for tracking the premixed the flame front. In particular, the Burning Velocity Model (BVM) and the Extended Coherent Flame Model (ECFM) solve directly a transport equation for \tilde{c} , while the G-equation model computes it according to an algebraic expression, by using different primitive variables, namely the mean value of the G-scalar and its variance [2]. During the exhaust and intake simulation the solution of the progress variable (\tilde{c} or \tilde{G}) is deactivated and \tilde{c} is frozen to zero, since no flame propagation is occurring and the residual gases act as ballast material.



Figure 4 – ANSYS CFX solving framework for premixed combustion

The reacted mixture composition is evaluated by look-up in flamelet libraries generated by the ANSYS CFX-RIF. This tool solves the steady diffusive flamelet equations [3] for different fuel and reaction mechanisms and returns the Favre-averaged mass fractions in the burnt mixture $\tilde{Y}_{i,b}$ as a function of $\tilde{Z}, \tilde{Z''^2}$ and the scalar dissipation rate χ .

Consistently, the species mass fractions in the fresh mixture with residual material are computed from the mixture fraction and the fuel tracer as follows:

$$\widetilde{Y}_{i,fr} = \left(1 - \widetilde{Tr}_{EGR}\right) \widetilde{Y}_{i,mix} + \widetilde{Tr}_{EGR} \widetilde{Y}_{i,b}$$

where the mass fractions for the inert mixing, $\tilde{Y}_{i,mix}$, and the flamelet solution, $\tilde{Y}_{i,b}$, are provided by the flamelet library. The reaction progress \tilde{c} is then used in order to blend $\tilde{Y}_{i,b}$ and $\tilde{Y}_{i,fr}$, returning \tilde{Y}_i .

Eventually, the species mass fractions are used in combination with the solution \tilde{H} of the energy equation in order to extract by means of Newton-iterations the absolute temperature *T* of the mixture.

Mesh Generation Process

Each step of the methodology requires a set of key-grids. During each simulation the mesh is deformed and smoothed according to a Laplacian algorithm, without introducing topological changes. When a specified mesh quality index reaches a threshold, the CFD computation is stopped and a remeshing process is triggered and performed in batch mode by ANSYS ICEM-CFD. The last solution is then interpolated to the newly generated grid and the solver is restarted.

The whole engine model represented in Figure 2 was split in two domains. The first one, including the plenum and the throttle body, was meshed with hexahedral elements according to a complex blocking strategy aimed to get selective refinements near critical geometric details with a limited amount of cells and is represented in Figure 5. The second domain, including all moving boundaries, was meshed according to different grid-generation techniques combining tetrahedral, prism and hexahedral elements. For instance, Figure 6 shows the computational grid at IVC, obtained by combining tetrahedra for the intake port, the dome and the piston bowl with extruded prisms in the cylinder region.

One critical region to mesh is the gap between the valve wall and its seat. For the exhaust phase different meshing solutions were tested, reflecting different compromises between computational costs (i.e. number and size of the elements) and accuracy. Figure 7 and Figure 8 depict the valve-gap region at EVO meshed respectively with tetrahedra and prisms or, additionally, with an axis-symmetric hexahedral block, automatically inserted during the key-grid generation process, according to a predefined template.



Figure 5 - Intake plenum, hexahedral mesh



Figure 7 – Valve gap. tet/prism mesh



Figure 6 – C3IO configuration tet/extrusion mesh



Figure 8 - Valve gap, tet/prism/hexa mesh

Results

The main goal of the intake simulation was to assess the capability of ANSYS CFX to handle complex multiconfiguration simulations, like the one illustrated in Figure 3, by means of a flexible setup and a robust meshing process. While a simple single-cycle solution on the reduced model represented in Figure 1 can be easily obtained, a scalable multicycle simulation is also possible, where the user can define in a modular way the number of cycles to be performed without any additional overhead in the pre-processing phase.

Figure 9 shows the average pressure on a cross-section of the intake port, located at 202 mm from the trumpet inlet, at the interface between the static hexahedral plenum and the dynamic tetrahedral domain, monitored during three consecutive

cycles. One may observe that, while the i^{th} cycle shows some differences in terms of peak amplitudes and phases, the next two iterations, starting from 180° CA, are superposed, and a cycle-to-cycle convergence is achieved. Eventually the $(i^{th}+2)$ solution could be used in order to initialize a combustion analysis at IVC.



Figure 9 - Average absolute pressure, cross-section at 202 mm from the trumpet inlet

The modeling framework described previously and summarized in Figure 4 provides the user with a detailed description of the mixture composition, allowing ultimately the evaluation of critical engine parameters like the volumetric efficiency. For instance, Figure 10 shows the local distribution of the residual product mass fraction respectively at EVC, highlighting the mixing between the fresh charge from the intake (blue) with the internal EGR trapped in the cylinder from the previous cycle and leaving through the exhaust (red).



Figure 10 – EGR at EVC

References

- 1. ANSYS CFX Solver Theory Guide, Release 12.1, ANSYS Inc., 2009
- Toninel, S., Forkel, H., Frank, T., Durst, B., Hasse, C. and Linse, D. "Implementation and Validation of the G-Equation Model Coupled with Flamelet Libraries for Simulating Premixed Combustion in I.C. Engines", SAE Int. J. Engines 2(1): 674-690, 2009
- 3. Peters, N. Turbulent Combustion, Cambridge University Press, Cambridge, 2000.