Efficient Gasoline Engine Combustion – Today's Role of CFD

Authors (please underline main author): Andreas Ennemoser, Karl Wieser, Marina Thelliez, Alexander Machold

Company/ University, Department: AVL List GmbH, PTE/DAC

Keywords: CFD, Combustion, CO2

Abstract:

In the past few years, the automotive industry heavily adopted turbo-chargers for gasoline engines. The drivers have been fuel economy ("down-sizing") and "low-end torque" requirements. This resulted in a large variety of new development programs at many OEMs. It happened to be that meanwhile CFD has been widely adopted for combustion system development at the same time. This resulted in a significant progress and strength of the applied methods and the contribution of CFD to the product development boosted up.

An integral part of CAE methods is the availability to predict numerous physical and chemical states in space and in time. Apart from accurate numerical models and well-applied meshing tools, proper criteria for the assessment of the combustion quality and efficiency have to be derived out of that. AVL's CAE procedures implement "performance attributes" to support the virtual design release process. These are technical release criteria which capture the essential properties of the investigated parts and systems with respect to their function. A set of attributes related to CFD supported combustion system development will be presented.

This presentation reports on the role of CFD, specifically the application of AVL FIRE, for the development of efficient gasoline engines as substantial part in fuel-efficient powertrains.

Pollutant emission optimization for Diesel engines

Authors (please underline main author): Marc Chauvy, Olivier Davodet

Company/ University, Department: PSA Peugeot Citroën, Combustion & Emission modeling

Keywords: optimization, combustion, Diesel, pollutants, simulation

Abstract:

(please use Times New Roman, 12 pt., max. 400 words)

The evolution of norms requires a constant and complex work to reduce pollutants emissions on automobile engines. In addition to the engine testing, numerical simulation being used ever more. On one hand, for early stages, more technological solutions (and also breakthrough solutions) are studied. On other hand, it is necessary to understanding combustion phenomena with analysis of simulation results.

In Diesel, there are several levers to achieve the objectives imposed by norms: for example, play on the injection parameters or changing the shape of the bowl. So far changes of these parameters were performed manually but for some years optimization tools help designers to "play" with these parameters automatically. The aim of this work is to present a tool that use an experimental design to obtain a response surface that will achieve this optimization in terms of emissions of soot and NOx. The goal is therefore minimizing soot and NOx and maximization of the IMP-HP although have a similar operating point for the different bowl shape. To achieve this goal, we will change the parameters describing the shape of the bowl and the injection (in total fifteen parameters), we will ensure, at the same time, to keep a chamber volume, maximum pressure and temperature at the exhaust opening constant (in order to permit comparison of the solutions together). We will present this approach, its implementation and the results of the optimization on a real sector Diesel engine.

Thermodynamic analysis with various Compression Ratios in direct injection diesel engines

Authors (please underline main author): Laetitia Passilly, Oliver Kastner

Company/ University, Department: Continental Automotive GmbH

Keywords: Compression ratio, Heat Exchange, Swirl, AVL-FIRE

Abstract:

Diesel engines have to fulfill a number of conflicting requirements. Apart from the ever tightening emissions legislations throughout the world the reduction of fuel consumptions became a target of highest priority.

The modification of compression ratio is considered as a possible lever to find the best compromise between engine efficiency and pollutant emission production. In order to understand and quantify the consequence of a compression ratio modification, an experimental and simulation study is carried out.

First, various possible piston geometry are considered. The selection of the geometry is performed based on air flow characterization using 3D-CFD simulation – Air flow only.

Once the samples are manufactured, tests are carried out on single cylinder. The results of these tests are used to validate a 3D-DFD simulation, this time including the injection, mixture formation and combustion process, using a sector approach.

A detailed analysis of the simulation results provides hints about the resulting effect of the modification of the compression ratio on heat transfer, spray development, mixture formation and finally combustion process. The relative importance of these modifications is tracked over time during the different phases of the engine cycle.

This Methodology delivers information that will be used for the further evaluation of the required performances of the injection system depending on the targeted compression ratio.

3D CFD Simulation of Oil and Gas Flow Across a 2-piece Piston Oil Control Ring

Authors (M. Carlsson, D. Konstanzer, H. Herbst):

Company/ University, Department: Scania CV AB

Keywords:

Abstract:

The oil consumption of a heavy duty diesel engine is important for a number of reasons, it affects the operating cost by the refill interval and it has an environmental impact directly and also indirectly as it affects the performance of the exhaust after treatment system. The oil loss can occur at a number of places, gaskets, ccv, turbo shaft seals, but the majority escapes over the pistons.

Minimizing the oil flow over the piston is the work of the piston oil control ring. Due to the difficult environment the development of this important component has mostly been done in engine tests where oil consumption and component wear are analyzed during and after the test.

This presentation will demonstrate one method to simulate the oil and gas flow across the oil control ring. Using boundary conditions from AVL Excite Piston and Rings a 2-phase model is set up with multiple in- and outlets, ring package motion, wall velocity and applied acceleration.

The results are in agreement with AVL Excite Piston and Rings as well as engine tests. The simulations have been used to increase the understanding of the flow in the ring package and we have been able to propose measures to decrease the oil overflow.

Engine Configurator based on BOOST and CAMEO

Hans – Michael Koegeler and Josef Wolkerstorfer

AVL Graz, ITS, PTE:

Keywords:

Gasoline Engine, Combustion Concept, Gas Exchange Simulation, Behavior Modelling, Optimization

Abstract:

The Engine Configurator introduced in this contribution uses advanced Thermodynamic knowledge combined with rich experience gained in many Simulation loops and test bed verifications. Applying this configurator enables the engineer to decide for the proper combustion concept regarding Engine Compression Ratio, Intake Cam length (Miller) and EGR range in the early engine development phase.
A quite complete Engine Model predicting relevant engine output parameters like

- BSFC values,
- Knocking intensity or
- Full load Torque
- MFB50% needed

based on combustion relevant parameter combinations such as

- Speed and
- BMEP target, but also
- Compression Ratio,
- Miller Cam Length,
- EGR and
- Charge Motion.

is realized on the base of AVL-BOOST in the crank angle domain.

The huge design space of different possible combinations of the design parameters needs a methodological approach: Similar as on engine test bed AVL-CAMEO uses AVL-BOOST to deliver the engine data results in an advanced DoE Approach.

CAMEO uses the identified multidimensional Behavior-Models for trade off Optimization. The user understands the reachable target ranges using the full variability, but also the constraints given by existing hardware platforms.

Consequently, the engineer can define the combustion system (Engine Compression Ratio, Intake Cam length (Miller) and EGR range) to be further developed in hardware for his engine-vehicle use case.

Modeling of the air fuel mixing and flame lift-off of a Diesel spray

Authors (please underline main author): Ender Celik

Company/ University, Department: Continental Automotive GmbH

Keywords: Flame lift-off, air/fuel mixing, AVL-FIRE

Abstract:

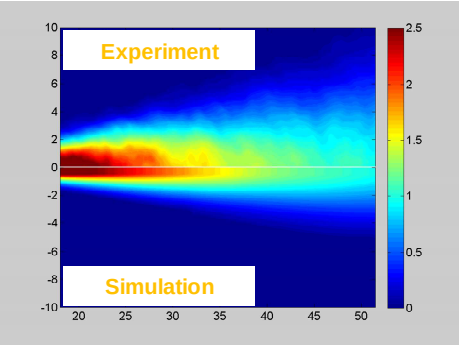
In the present study, first non-reacting, vaporizing spray simulations were performed focusing particularly on the air/fuel mixing. Spray A test case of ECN (Engine Combustion Network) has been used. A single-hole nozzle is employed in this test case to inject n-Dodecane into the constant volume vessel (with N2 at 60bar pressure and 900K temperature) with an injection pressure of 1500 bar and no cavitation is observed at the nozzle exit.

CFD simulations were carried out with RANS based turbulence models, namely standard k-ϵ and k-ζ-f models and liquid spray phenomena are modelled using the Lagrangian approach with the well established spray submodels.

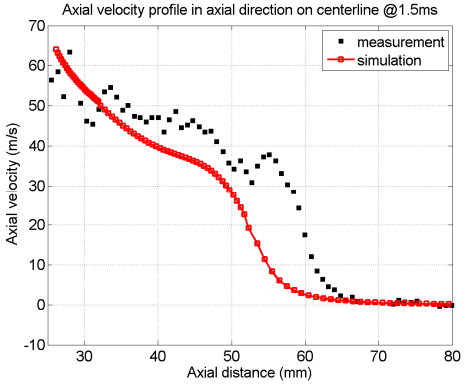
For the validation of the numerical methodology, maximum liquid and vapour penetration measurements, Schlieren measurements of fuel vapour boundary evolution, PIV measurements of gas phase axial velocity and Rayleigh scattering measurements of equivalence ratio field at the quasi-steady state were compared with the simulated data.

Secondly, for reacting cases, a formula has been written to quantify the flame lift-off length based on the predicted OH concentration field. The definition used for flame lift-off length determination is “the first axial location of Favre-averaged OH mass fraction reaching 2% of its maximum in the domain at that time level.”

Consequently, it has been observed that grid resolution and breakup parameters have an influence mainly on the maximum liquid and vapour penetrations whereas it has an insignificant effect on the fuel vapour distribution. However, the turbulence modelling has a major effect on the air/fuel mixing.



Equivalence ratio field at quasi-steady state,(top - Rayleigh scattering measurements,bottom - simulation)



Modeling of Diesel Spray from Multi-hole Nozzle under Off-Axis Needle Displacement

Authors Chiatti,G., Chiavola, O., Palmieri, F.

Company/ University, Department: Roma Tre University

Keywords:

Abstract:

(please use Times New Roman, 12 pt., max. 400 words)

Relatively recent investigations, basing on experiments as well as on modeling, have highlighted that the needle displacement in common-rail diesel injectors is affected by radial components. The effects of such "off-axis" needle displacement on fuel flow features have been so far investigated within the nozzle, only.

The objective of this work is to extend the attention towards the formation of fuel sprays, when needle off-axis condition is encountered. In such a viewpoint, the development of each fuel spray has been modeled taking into account the hole-to-hole variations induced by the needle misalignment.

The investigation has been carried out basing on 3D-CFD campaigns, in AVL FIRE environment. The modeling of diesel nozzle flow has been interfaced to the spray simulation, initializing the break-up model on the basis of the transient flow conditions (fuel velocity, turbulence and vapor fraction) at each hole outlet section.

The current investigation has been performed on a complete geometry of a common rail multi-hole VCO nozzle. Simulations have shown how the needle misalignment is reflected in the formation of the fuel jets. Quantitative results have been provided on each fuel spray development, pointing out the hole-to-hole variations induced by the needle off-axis displacement.

1D-3D coupling simulations for EGR distribution in the cylinder

Authors (please underline main author): Jérôme Houstin[1], Olivier Davodet[2]

Company/ University, Department: Alten[1], PSA[2]

Keywords: EGR, 1D – 3D coupling

Abstract:

In a period of intense efforts to reduce pollutant formation by in-cylinder measures, the use of EGR is considered by the automotive industry as an efficient solution. In the framework of massive deployment of numerical solutions, PSA decided to focus on 1D – 3D coupling solutions to simulate unsteady phenomena, as the EGR process, with the final objective to integrate them in the design process.

As EGR modifies the combustion behavior, its impact has to be evaluated. To ensure good performances together with a high level of driving comfort and safety, it is essential to generate a good distribution of EGR with correct EGR rates in each cylinder. The coupling between 1D and 3D simulation tools allows to calculate the EGR rates and to visualize the internal aerodynamic phenomena, which is of great help to orientate the design.

The work is based on a coupling between a complete GT-Power engine model and a 3D computational domain containing the intake manifold and the connection with the EGR circuit simulated with AVL FIRE. For each simulation defined for specific operating conditions, both software tools calculate several engine cycles in order to ensure the convergence of the simulations, the data exchange between the 2 tools operating at user-defined crank angle positions.

The simulation approach as well as visualizations and results will be presented in detail.

Method for Predicting Li-Ion Cell Reactions due to Mechanical Crash Loads

Authors: Simon Franz HEINDL (VSI), Christoph BREITFUSS (VSI), Christian ELLERSDORFER (VSI), Florian FEIST (VSI), Wolfgang SINZ (VSI), Alexander GEIER (Audi), Reinhard TATSCHL (AVL), Clemens FINK (AVL), Zoran PAVLOVIC (AVL), Peter GOLLOB (AVL), Andreas BRAUN (AVL)

Company/ University, Department: Vehicle Safety Institute (TU Graz)
Audi AG
AVL List GmbH

Keywords: Simulation, mechanical, electrothermal, Lithium, Lithium-ion, Li-ion, battery, cell, critical, condition, prediction, vehicle, crash, hybrid, experiments

Abstract:

The automotive branch's latest developments have shown firm tendency towards electrification of drive trains. Besides the functional application of electrical energy storage systems (EES) in road vehicles a strong focus is on the integration of battery systems considering safety aspects. For this reason a hybrid simulation approach has been developed and applied. The main task of this tool is to predict the consequences of mechanical load passed to EES. The hybrid approach is composed of the three major parts explicit simulation, geometry interface and implicit simulation. Both for the explicit and the implicit solver proprietary software (ESI PAM-CRASH, AVL FIRE) is being used leaving the hybrid approach to innovatively combine those two strands synthesizing their outcome. The hybrid approach principle can be applied on internal as well as on external short-circuit situations but is shown here solely for the former. The explicit part solves the finite element code and depicts the mechanical stress in the cell's calculation elements. A dynamic stress limit or deformation threshold values enable the software to issue statements whether damage has occurred or not. A penetrated separator in this case is a synonym for an inner short-circuit and thus is likely to lead to thermal runaway. The information held in the calculation mesh is then transferred to the geometry interface tool which in turn generates the simulation grid for the implicit electrothermal solver. Cell elements which have been subject to damage are specially handled and implemented as so called short-circuit areas. The implicit electrothermal part then solves equation systems representing the battery's electrochemical processes and most important simulates short-circuit induced time dependent effects of the cell. Critical conditions (e.g. temperature rise above electrolyte's flash point) can be predicted. Basis for both explicit and implicit simulation models are especially designed experiments including dynamic three point bending, nail penetration and high temperature exposure tests. In summary, the hybrid approach allows explicit mechanical simulations on a millisecond time base and long-term (minute/hour based) electrothermal calculations being combined into a single tool giving us an excellent and promising foundation for further research.

Numerical and Experimental Investigation of the Flow Distributions in Fuel Cell Stack Manifolds

Authors: L. Feierabend[1], S. Burgmann[1], F. Schmieder[2], M. E. Kinaci[1], L. Büttner[2], J. Czarske[2]

Company/ University, Department: [1]Zentrum für BrennstoffzellenTechnik GmbH (ZBT), [2]Technische Universität Dresden, Prof. für Mess- und Prüftechnik,

Keywords: Fuel cell stack, flow distribution, multimode laser-light, fluorescent tracers, MM-LDV-sensor, CFD

Abstract:

Fuel cell stack performance strongly depends on the uniformity of the flow distribution over the individual fuel cells. Typically, basic engineering expertise and numerical flow simulations are used to design the small flow geometries within a fuel cell stack that should properly deliver the reactants to the active reaction zones. Depending on the operating conditions of the fuel cell stack, the flow conditions typically range from laminar flow everywhere inside the fuel cell stack to low Reynolds number transitional and turbulent conditions at the distributing manifold entrance and the collecting manifold exit. Therefore, the results of flow simulations inside a fuel cell stack strongly depend on the applied turbulence models and the specific wall modeling in combination with the corresponding grid resolution [1, 2]. Experimental data of the real flow conditions in the resulting stack geometries is mostly lacking. To measure the flow distribution within these small geometries, a specific Laser Doppler Velocimeter has been developed that uses light with low spatial coherence (multimode light) and fluorescent tracer particles. A spatial resolution of less than 100 µm even close to the wall is achieved. Numerical simulations with laminar flow conditions and low Reynolds number turbulent conditions applying the k-ε- and k-ζ-f-turbulence models have been performed and were compared against the experimental data obtained with the developed MM-LDV-sensor and classical PIV measurements for a simple model manifold setup. Comparing the results revealed a very good agreement of the experimental data. However, the numerical results are strongly dependent on the setup (i.e. steady or transient, turbulence modeling, wall modeling, and grid generation). In conclusion, experimental reference data are crucial to finding the appropriate setup for numerical modeling of the specific flow conditions in the manifold geometries. Then, the tested and validated sensor was applied to the flow within the real geometry of a fuel cell stack manifold to gain a deeper knowledge of the flow configuration and to generate a benchmark for CFD-based fuel cell design.

The Development of JMC Engine Simulation Platform and Its Applications

Authors (please underline main author): X. Zeng
Company/ University, Department: Jiangling Motor company (JMC)

Keywords: Independent engine simulation platform, establish, CAE applying, the beacon light

Abstract:

Nowadays, CAE analysis is becoming more and more important for engine independent development, which runs through the whole engine development process and plays a key role in determining the engine design level. For JMC, conventionally, the development of a brand new or derivative engine was mainly based on the experiences, reverse design , outsourcing and so on, which would cause higher-cost, longer-period and confidential issues. After 6 years' learning, researching, innovation and accumulation of the CAE technologies, JMC CAE TEAM has established an independent engine simulation platform. This platform consists of the methodologies as bidirectional fluid-structure coupling analysis, cooling jacket analysis, 3D&1D coupled simulation for cooling system, calibration method of engine oil consumption, calibration method of automatic tensioning wheel, etc . Based on this, now, JMC CAE TEAM can independently undertake the tasks of project P3 levels and parts of P4\P5 levels, and supply a strong and effective support to these engine development projects, like shortening the development cycle, cutting costs more the ten millions Yuan for the company. These remarkable contributions gain the leaders and other department's consistent compliments, and provides the guide for the automobile engine development. In this paper, the platform development process &innovation points, the analysis method & process, JMC CAE database development and successful application cases are introduced .