Optimized Charge Air Cooling for Diesel Passenger Cars with Respect to Upcoming CO2-Limitations & RDE-Use-Cases

Christian Doppler,

VIRTUAL VEHICLE Research Center

Gernot Hirschl,

AVL List GmbH

Gerhard Zsiga,

MAHLE Behr GmbH & Co. KG

Keywords: Indirect Charge Air Cooler, Co-Simulation, Engine Gas Exchange, CO2-Reductin

Abstract:

With the introduction of RDE in 2017 (EU6c measure), the controversial NEDC (New European Driving Cycle) will belong to the past and the OEMs will have to avoid harmful emissions in any conceivable real life situation. Further, in 2020, an EU-wide CO2-limitation of 95 g/km is scheduled for the average of an OEMs passenger car fleet. Taking that into consideration, additional improvement measures of the Diesel cycle are necessary in order to reduce fuel consumption and emissions while boosting, or at the least, keeping performance values at the same time.

The present article deals with the possibilities of an optimized air/water charge air cooler, also called iCAC (indirect Charge Air Cooler) for a Diesel passenger car amongst extreme-boundary conditions. The considered engine is a 2.4 litre EURO VI diesel engine with variable-geometry turbocharger (VGT) and low pressure exhaust gas recirculation (LP EGR). Object of study was the impact of charge air cooling on the engine working process at constant boundary conditions which could have been conducted with an available and validated engine model in AVL BOOST. Part load was realized with constant power and NO_x-emissions, whereas full load was accomplished with a lambda control in order to obtain maximum engine performance. The informative results were used to implement a simulation model in Matlab/Simulink which is further integrated into a full vehicle simulation environment via coupling with ICOS (Independent Co-Simulation Platform). Next, the dynamic engine behavior was validated and modified with load steps taken from the engine test bed. Due to the modular setup in the Co-Simulation, different CAC-models have been simulated quickly with their different influences on the working process. In doing so, a new cooler variation isn't needed to be reproduced and implemented into the primary simulation model environment, but is implemented quickly and easily as an independent component into the simulation entity. By means of the association of the engine model, longitudinal dynamics vehicle model and different CAC models (air/air & water/air variants) in both, steady state and transient operational modes, statements are gained

regarding fuel consumption, NO_x-emissions and power behavior.
Several extreme-boundaries - e.g. varying ambient temperatures or mountainous routes - that will become very important in the near future regarding RDE (Real Driving emissions) were subject of the investigation. The simulations showed a benefit in FC between 1 and 3% for the iCAC in real world conditions.
Beside of the simulative tests, there have been conducted also measurements on the engine test bench with the mentioned demonstrator engine, those who approved the simulation results very well.

Modification of Intake Port Shape for Increasing of Effective Parameters of Middle-Speed Diesel Engine

Authors: R.Z. Kavtaradze, A.A. Zelentcov

Company/ University, Department: Bauman Moscow State Technical University, Russia; Department of Piston Engines

Keywords: diesel engine, swirl number, intake port, effective and ecological parameters

Abstract:

Internal combustion piston engines remain the basis of power units for the road transport at present. Support of high effective and ecological parameters of piston engine is inseparably connected with improving of processes in combustion chamber. One way of improvement of mixture formation and combustion of air-fuel mixture consists of intensification of swirl (or tumble) motion of gas into the cylinder.

The objective of current investigation is rising of effective performance of middle-speed diesel engine with support of given ecological parameters by means of intake port modification. For the purpose of solving of assigned task the intake port shape with optimal swirl number *SN* is determined. After that the effective and ecological parameters of diesel engine with modified cylinder head are calculated.

In paper the different variants of diesel engine intake ports are considered: one variant allows obtaining higher swirl number than base intake ports, and another one allows to receive lower swirl number (the ports under consideration allows obtaining *SN* = 0.78; 0.99 (base variant) and 1.28). Three-dimensional simulation of intake process was carried out with imitation of conditions of test bed specialized for investigation of port throttling characteristic in case of maximum valve lift.

For the intensification of swirl motion in the cylinder of diesel engine it is recommended to use the intake ports with decreased angle of axis relative to horizontal plane (which coincides with the cylinder head plane) as they provide swirl number increase up to 1.28.

Three-dimensional simulation of in-cylinder processes allowed evaluating the influence of swirl motion in the intake ports of diesel engine on its effective and ecological parameters. As a result the increasing of effective parameters was obtained with the intensification of swirl motion (effective power N_e = 320 kW when *SN* = 0.78 and N_e = 329 kW when *SN* = 1.28). At the same time with rise of *SN* the concentration of nitric oxides increased too; with decrease of swirl number the inverse dependence was observed.

Thus the modification of intake ports geometry allows to obtain increasing either effective or ecological parameters of middle-speed diesel engine. Simultaneous decreasing of soot and nitric oxides with the acceptable level of effective parameters can also be achieved at the expense of using Premixed Charge Compression Ignition (PCCI) or application of aftertreatment systems.

A Multi-Site Kinetic Model for NH_3-SCR over Cu/SSZ-13

Authors: Louise Olsson[1,*], Kurnia Wijayanti[1], Kirsten Leistner[1], Ashok Kumar[2], Saurabh Y. Joshi[2], Krishna Kamasamudram[2], Neal W. Currier[2], Aleksey Yezerets[2]

Company/ University, Department: [1] Chemical Engineering; Competence Centre for Catalysis, Chalmers University of Technology,
[2] Cummins Inc.

Keywords:

Ammonia SCR; Cu/SSZ-13; Kinetic model; Multi-site model

Abstract:

We have developed a kinetic model for ammonia-SCR over a well-characterized Cu/SSZ-13 catalyst [1]. The model is based on micro-calorimeter data combined with flow reactor experiments. In order to describe the ammonia adsorption and desorption, ammonia oxidation and standard SCR over a broad temperature range, three sites were needed. Earlier studies in literature have proposed that there are two active copper species present and that the SCR reaction occurs mainly on the under-exchanged sites and the oxidation reactions on the over-exchanged sites. In addition, results from detailed characterization of Cu/SSZ-13 have shown that copper is first ion-exchanged into the six-membered ring, followed by introduction into the large cages. Our kinetic model therefore contains the following sites: S1, which represents copper in the six membered rings, where the SCR reaction mainly occurs and S2, which represents copper in the large cages, where the ammonia oxidation mostly take place and finally S3 where loosely bound ammonia can adsorb. In addition, ammonia is also stored on Bronsted acid sites, but in order not to complicate the model further, it was lumped together in the S1 and S2 sites. Ammonia oxidation on the Cu/SSZ-13 showed an interesting behavior, where the ammonia conversion was somewhat decreased at 400°C compared to 350°C. In the model, the main oxidation occurs at high temperature on S2 sites, but it was also necessary to introduce ammonia oxidation with a low rate on the S1 sites. When increasing the temperature the coverage of ammonia on S1 decreased, resulting in a decreased ammonia oxidation. For even higher temperatures, ammonia oxidation started to increase again, due to ammonia oxidation on S2 sites. In a similar way, ammonia SCR reactions were introduced on both S1 (main SCR site) as well S2 in order to describe the SCR activity over a large temperature interval. To conclude, the three-site model developed was capable of describing the experimental features for ammonia adsorption and desorption, ammonia oxidation, as well as SCR and N_2O formation over a large temperature interval (100 to 600°C).

[1] L. Olsson, K. Wijayanti, K. Leistner, A. Kumar, S. Joshi, K. Kamasamudram, N.W. Currier, A. Yezerets, Appl. Catal. B: Environmental. 174-175 (2015) 212.

Efficient use of detailed chemistry in AVL FIRE with FGM

Authors (please underline main author): Ferry TAP, Dmitry Goryntsev, Anton Starikov

Company/ University, Department: Dacolt International BV

Keywords: detailed chemistry, turbulence-chemistry interaction

Abstract:

The FGM combustion model allows to use detailed chemistry in CFD at minimal CPU cost. Additionally, the model accounts for turbulence-chemistry interactions, further improving the predictivity. In this work, the FGM model in AVL FIRE is applied to heptane spray flames from the Engine Combustion Network (ECN) and a Diesel engine test case.

The FGM look-up tables are generated with Tabkin®, Dacolt's dedicated software program for the generation of CFD look-up tables for advanced combustion models.

Engine Emission and Performance Optimization with Optimus

Author: Silvia Poles

Company/ University, Department: Noesis Solutions

Keywords:

Process integration, Design Optimization, Optimus, PIDO, multiobjective optimization, emissions reduction, fuel consumption

Abstract:

Optimus is a Process Integration and Design Optimization (PIDO) platform that allows to integrate and combine any engineering software tool into a single simulation workflow.

Once a workflow is defined, Optimus orchestrates the simulation process to automatically explore the design space, automates simulations, identifies the optimized solution and allows a valuable insight into the simulation model.

In this presentation we demonstrate how Optimus can easily integrate with AVL FIRE & AVL BOOST and how it can be used to save significant time in the design process. We describe the benefits of the process integration by means of two application cases.

The first case is a diesel engine emission multiobjective optimization using AVL FIRE in which the soot and the NOx emission are minimized simultaneously changing geometrical and physical design variables. The entire Optimus workflow has been easily exported from AVL and automatically built without requiring any specific knowledge about process integration. The final optimal solution of this first optimization case has led to a 24% reduction of the NOx and a 15% improvement in soot emission with respect to the nominal configuration.

The second case is an optimization of torque and fuel consumption using AVL BOOST. Also in this case, the problem is solved taking care of both conflicting goals in a truly multiobjective approach in which Optimus automatically identifies all Pareto-optimal solutions. Selecting the most preferable configuration, out of the entire set of optimal points, delivers a 16% power increase and reduces fuel consumption by 10%.

3D Numerical Simulation of CNG Direct Injection

Authors : A. Twellmeyer, F. Köpple, B. Weigand

Company/ University, Department: Robert Bosch GmbH

Keywords: compressed natural gas, direct injection

Abstract:

The usage of compressed natural gas (CNG) as a fuel in internal combustion engines is very promising in terms of efficiency and CO_2 emissions. Due to the lower carbon/hydrogen ratio in methane, which is the main component of CNG, carbon dioxide emissions can be reduced by about 25 % compared to conventional gasoline combustion. Furthermore the high octane number of methane permits higher compression ratios leading to enhanced efficiency of the engine.

To use the full potential of natural gas as a fuel, suitable mixture preparation and combustion concepts for direct injection have to be developed. The aim of this work is the numerical investigation of the mixture formation by means of 3D CFD and its validation against experimental data.

Two different concepts are employed to simulate direct injection of methane from different nozzle geometries using AVL Fire. Besides the detailed modeling, considering the internal nozzle flow, a simplified gasjet module based on the Euler-Lagrange approach was used. Given the low density of gaseous methane, high volume flow rates are necessary to provide sufficient fuel charge during the short time frame of stratified operation. Therefore a supersonic gas jet is formed including a Mach disk and high gradients of the flow parameters in the expansion zone close to the orifice. The modeling of these flow discontinuities is numerically challenging and requires a full resolution of the nozzle geometry which makes the simulation demanding in terms of computational resources. The gasjet module provides a simple Lagrangian alternative for modeling the fuel injection with less computational effort since the nozzle geometry doesn't have to be discretized.

The simulation results are compared to optical data from Schlieren and tracer-based 2D laser-induced fluorescence measurements. The gas jet behavior in the nozzle vicinity simulated using the detailed approach of modeling the internal nozzle flow was found to be in good agreement with experimental results. The localization of the Mach disk as well as the periodicity of the expansion waves could be simulated with good accuracy. Furthermore the transient jet propagation was simulated satisfactorily. The gasjet module was found to be an appealing option for the flow modeling from single-hole injectors in the area further down the nozzle, especially regarding the comparatively small computational effort.

Investigation of the flame propagation in a large gas engine with LES

Authors: Werner Holly, Thomas Lauer, Peter Priesching, Jürgen Schneider, Reinhard Tatschl

Company/ University, Department: TU Wien, AVL List GmbH

Keywords:

Abstract:

Due to their high efficiency, large gas engines are widely used for stationary power plants as well as for ship propulsion. These engines operate with very lean air-fuel mixtures to ensure low raw emissions. Therefore advanced ignition concepts are necessary to achieve a stable combustion. A widely used example for such an advanced ignition concept is the pre-chamber spark plug. The combustion in such a prechamber and its interaction with the main combustion chamber is very complex. Therefore CFD-simulations were set up to investigate the charge motion and the flame propagation in the pre-chamber and cylinder.

The mesh used in the computational study consists of 6 to 10 million cells and the combustion model is based on the well-known ECFM-family. Further, two different modelling approaches for the turbulence were used, namely the k-zeta-f RANS model and the LES-CSM.

The investigation revealed, that the RANS simulation is not sufficient to represent the complex turbulence and flame behaviour in the prechamber and therefore the experimental observations could not be represented. In contrast, the LES approach was able to describe the flow field and the combustion process with high accuracy. As a result, a good correlation between the measured and simulated flame propagation velocities in the prechamber and cylinder were achieved. It was possible to study the combustion process in the prechamber, the torch penetration in the main chamber and the flame propagation in the cylinder. Further, the differences between the RANS and LES results were analysed and will be presented in the paper.

Due to the fact that the LES approach resolves the cyclic variations, a high number of consecutive cycles were simulated. Based on these cycles, the sources of cyclic variations were investigated and will be presented.

The topic of this paper is the combustion simulations in a large gas engine with pre-chamber spark plug. Based on the LES-approach it was possible to simulate the heat release rate in good agreement with the measurements. Thereby, this model is a very useful tool for the further development of combustion processes.

Numerical investigation and realisation of optimised valve timing for an OHV cogeneration engine

Authors: Jürgen Bauer, Denis Neher, Maurice Kettner

Company/ University, Department: Institute of Refrigeration, Air-Conditioning, and Environmental Engineering (IKKU), Karlsruhe University of Applied Sciences

Keywords:

Valve timing optimisation; cogeneration engine; OHV-engine

Abstract:

In times of rising primary energy prices and tightening emission standards, significant efforts in development of cogeneration engines are required to maintain their profitability. Theoretically, the simplest approach of improving engine efficiency is to rise the compression ratio. For real engine operation, however, the chance of knocking combustion increases.

At Karlsruhe University of Applied Sciences, investigations are carried out to increase the compression ratio of a four-cylinder cogeneration engine, while maintaining knocking tendency at the level of the baseline engine. For this purpose, various knock reducing measures, such as gas exchange optimisation, are applied. This paper demonstrates the use of AVL BOOST and AVL EXCITE for valve timing optimisation and the subsequent realisation of the determined valve lift curve in cam grinding contour.

In a first step, a detailed numerical model of the natural gas engine was created and adjusted for the reference operating point. Valve opening and closing timings were varied in a wide range and analysed with respect to gas exchange, in particular to residual gas content. It exhibits a major influence on the in-cylinder temperature, whose history, along with the one of the in-cylinder pressure, is the characteristic parameter of knocking tendency. The optimisation study was conducted at a nominal engine speed of n = 1500 rev/min and a series relative air-fuel ratio of λ = 1.55. The combustion was realised by implementing a constant normalised rate of heat release, which was derived from the measured in-cylinder pressure trace of the test bed engine. The optimised valve timing reduces the residual gas content of up to 2.7 %-points, causing a drop in in-cylinder temperature at the beginning of the high-pressure phase of up to 15K.

In order to realise the optimised valve lift, a multibody simulation model of the OHV configuration was created by considering stiffness and inertia of each component. When comparing calculated and measured valve lift of the series cam contour, only a minor deviation can be observed. In the next step, the optimised cam contour was realised using AVL CamDesign. Cam and follower were designed while taking the occurring Hertzian pressure, ramp speed and minimum tip radius into account.

Based on the determined cam grinding contour, a prototype camshaft will be manufactured to investigate its influence also experimentally.

Combustion Prediction for Large Diesel Engines using 3D CFD Simulation

Authors: Jan Vystejn, Gert Taucher, Engelmayer Michael

Company/ University, Department: Large Engines Competence Center, TU Graz

Keywords:

CFD, diesel combustion, large bore engines

Abstract:

The theoretical capability of a CFD code to precalculate the influences of various hardware configurations (i.e. piston shapes, cylinder heads and injectors) on the combustion of a diesel engine has always been around, but it was mostly the inaccuracy of the results together with rather small achievable gains that prevented its effective use during the engine development stage. With always strengthening emission limits and also constantly improving CFD codes the question, whether such simulations can be carried out accurately enough in order to save time and resources otherwise needed for experiments, is being asked repeatedly with an increasing emphasis.

The presentation gives a short summary of what is needed to for a successful simulation of a high speed large bore diesel engine combustion that is accurate enough to significantly reduce the amount of experiments needed for a hardware configuration optimization.

Two different engine hardware configurations are discussed and thoroughly investigated. At the beginning the process of mesh generation is stated, where the influences of the various meshing approaches are discussed. Setup of the spray and combustion models follows together with its validation through optical investigations means. At the end simulation results of several engine operating conditions are compared to the measured results from a one cylinder research engine.

Modelling of Cyclic Variability in Combustion of Spark-Ignition Engine Using the Cycle-Simulation Model

Authors (please underline main author): Momir Sjerić, Darko Kozarac, Ivan Taritaš

Company/ University, Department: University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture, Chair of IC Engines and Motor Vehicles

Keywords: spark-ignition, combustion, turbulence, cyclic variability

Abstract:

The cycle-to-cycle variations (CCV) in combustion of SI (spark-ignition) engine are initiated by the stochastic nature of in-cylinder flow field and by the imperfect mixing of fuel, air and residual gases that oscillates from cycle-to-cycle. The presence of CCV in combustion causes that the individual combustion progress deviate from the cycle-averaged result increasing the fuel consumption and occurrence of knocking cycles. Variations of these factors are shown in the initial kernel growth rate, in the shifting of the flame center from the spark plug location and in the turbulent burning rate. For the correct prediction of CCV in combustion all mentioned effects should be taken into account. For the modelling of in-cylinder turbulence the developed k-ε turbulence sub-model is applied while the spark-ignition associated phenomena are described by using the newly developed quasi-dimensional ignition sub-model (QDIM) that is integrated into the modified quasi-dimensional combustion sub-model of AVL BOOST. The modified quasi-dimensional combustion sub-model is based on the fractal theory and it includes the ignition sub-model, modified transition from laminar to fully developed turbulent flame and two-zone turbulence. The newly developed sub-models for turbulence and combustion were calibrated with the available 3-D CFD results for several SI engines. After that, the cycle-simulation was applied to simulate the cyclic variability in combustion by comparing the cycle-simulation results with the experimental data. The CCV in combustion were triggered with the random and independent variations of in-cylinder turbulence level and the flow angle at the spark plug from cycle-to-cycle. The CCV study was performed on the single cylinder and naturally aspirated SI engine fuelled by gasoline. For each operating point at full load conditions 300 cycles were analyzed and simulated with the application of single set of constants. The simulation results are in a very good agreement with the experimental data for several operating points. The presented cycle-simulation model that includes developed sub-models for the turbulence, ignition and combustion represents the fast and well promising tool to simulate the real performance of SI engine.

Study of Atkinson and Miller Cycles Based on Vibe and Fractal Combustion Model

Authors: Din-Sun Ju

Company/ University, Department: Engine Performance Development Department, China Engine Corporation

Keywords:

Atkinson cycle, Miller cycle, Fractal combustion model, Vibe combustion model

Abstract:

The fuel consumption potential and performance influence with Atkinson cycle and Miller cycle are evaluated for turbo aspired engine. In concept stage of engine development, cam profile is one of the most important designs. Atkinson cycle working with late-intake-valve closing (LIVC) and Miller cycle working with early-intake-valve closing (EIVC) are common strategies to improve fuel economy by reducing pumping loss. However, the applications of theses cycles cause a decrease in performance due to worse combustion.

In this study, it focuses on the part-load fuel consumption and full-load performance with Atkinson and Miller cycle based on Vibe and Fractal combustion model. AVL Boost is utilized to analysis the fuel consumption and performance. Atkinson cycle is implemented by retarding the intake-valve-closing. The cam with shorter duration and decreasing lift is designed for Miller cycle. The benefits, performances, and combustion phenomenon with different combustion model are showed and discussed.

Research and Optimization to Improve Distribution Uniformity of EGR

Authors (please underline main author): Xiaochun Zeng

Company/ University, Department: Jiangling Motor company (JMC)

Keywords: engine emission, intake manifold system, EGR rate , CAE coupling analysis, structure improve, test verification

Abstract:

EGR technology is an effective approach to reduce engine emission, one big issue of this technology is how to realize the even distribution of EGR for each cylinder. In this study, the gas flow and mixing procedure in intake manifold and EGR pipes was simulated through 1D-3D coupling analysis and EGR rate of each cylinder was obtained and analyzed. Based on the study of the results, optimization of the intake manifold system design was guided. Compared with the original plan, for plan B, a "steady flow chamber" was added at the entrance of EGR pipe to the intake manifold, which played as a buffer and achieved a more steady flow when exhaust gas flowed into the chamber. This would lead to a steadier and stronger mixing of exhaust gas with fresh air. Further tests of prototype with this plan B also proved that this design improved the uniform distribution of EGR for each cylinder and reduced the emission.