

Analysis of Cycle-to-Cycle Variations of the Mixing Process in a Direct Injection Spark Ignition Engine Using Scale-Resolving Simulations

Martin Theile
FVTR GmbH

Egon Hassel
University of Rostock

Dominique Thévenin
University of Magdeburg

Bert Buchholz
University of Rostock

Karsten Michels and Martin Hofer
Volkswagen AG

ABSTRACT

Since the mechanisms leading to cyclic combustion variabilities in direct injection gasoline engines are still poorly understood, advanced computational studies are necessary to be able to predict, analyze and optimize the complete engine process from aerodynamics to mixing, ignition, combustion and heat transfer. In this work the Scale-Adaptive Simulation (SAS) turbulence model is used in combination with a parameterized lagrangian spray model for the purpose of predicting transient in-cylinder cold flow, injection and mixture formation in a gasoline engine. An existing CFD model based on FLUENT v15.0 [1] has been extended with a spray description using the FLUENT Discrete Phase Model (DPM). This article will first discuss the validation of the in-cylinder cold flow model using experimental data measured within an optically accessible engine by High Speed Particle Image Velocimetry (HS-PIV). Afterwards, the parameterized spray model is validated using experimental data measured in a pressure spray chamber. Finally, results obtained with the combined model are discussed and used to analyze transient mixture formation and to give a detailed insight into cycle-to-cycle fluctuations associated with the turbulent mixing process in an internal combustion engine (ICE).

CITATION: Theile, M., Hassel, E., Thévenin, D., Buchholz, B. et al., "Analysis of Cycle-to-Cycle Variations of the Mixing Process in a Direct Injection Spark Ignition Engine Using Scale-Resolving Simulations," *SAE Int. J. Engines* 9(4):2016, doi:10.4271/2016-01-9048.

INTRODUCTION

Cyclic combustion variabilities (CCV) are one of the major issues encountered when optimizing gasoline engine combustion. CCV result in high pressure and velocity fluctuations in the combustion chamber, particularly during expansion phase. Figure 1 shows the behavior of the incylinder pressure in an SI Engine and will be used to describe the effects of CCV exemplary. Within a certain operating point with low EGR (black curve) one can see fluctuations of the incylinder pressure. The cycle averaged maximum peak pressure $p_{\text{max,mean}}$ is about 41.5 bar, together with a standard deviation σ_p of about 2.5 bar, which results in a coefficient of variance $\text{COV}_{p_{\text{max}}}$ of about 6%. Changing the operating condition to higher EGR, $p_{\text{max,mean}}$ stays relatively constant with 40.1 bar, but σ_p is increasing to 4.1 bar, which results in $\text{COV}_{p_{\text{max}}} = 10,2\%$. The same behavior can be seen by looking at the IMEP. The cycle averaged IMEP is almost constant

with $\text{IMEP}_{\text{lowEGR}} = 6.19$ bar and $\text{IMEP}_{\text{highEGR}} = 6.15$ bar. Since BMEP was set fixed, differences only occur due to different gas exchange efficiencies. The standard deviation of the IMEP is changing from $\sigma_{\text{lowEGR}} = 0.041$ bar to $\sigma_{\text{highEGR}} = 0.135$ bar, while the COV_{IMEP} is increasing from 0.66% to 2.19 %. This does not result in unstable operating conditions for this engine, but it shows clearly the increasing amount of cyclic variability due to different incylinder conditions respectively combustion methods.

Heywood [2] sets the limit of the COV_{IMEP} to 10%, before serious drivability problems occur. Different groups found unstable operating modes already at lower values of COV_{IMEP} (e.g. 2% [3] or 4% [4]).

The origin of this characteristic behavior of CCVs in general goes back to the turbulent in-cylinder flow, developed only after the ignition time, mainly at the intake stroke. The different flow fields

developing throughout the intake phase, affect the mixing process and result therefor in cycle dependent initial conditions for the ignition and combustion process. Since fuel efficiency and pollutant emissions both need to be optimized to fulfill new emission standards (EPA Tier 2, CARB LEV-II or Euro 6) with moderate exhaust aftertreatment effort, modern analysis tools must be developed for industrial applications. The coefficient of variance (COV) is an important optimization parameter when trying to fulfill consumer requirements. It can be obtained by CFD simulations when using scale-resolving turbulence models (SRS, in particular the Scale-Adaptive Simulation, or SAS, turbulence model) in combination with already established submodels (e.g., Lagrangian spray models). However, a careful analysis is needed to make sure that all models work properly in combination with each other, since most of the submodels were developed in a classical (U)RANS context.

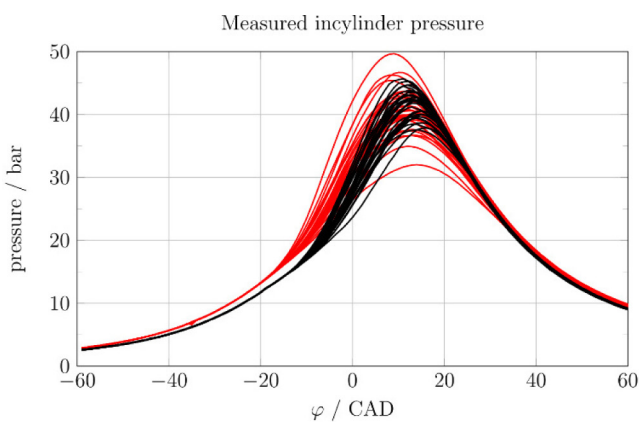


Figure 1. Example of cyclic fluctuations of the in-cylinder pressure measured at a spark ignited research test bench engine at University of Rostock at two different operating points (black line: low EGR, red line high EGR). For a better visualization, only 30 cycles per operating point are plotted.

In this project, a 3D CFD model has been developed to describe and analyze the impact of the turbulent in-cylinder flow on the cyclic fluctuations found during mixture formation in a direct injection spark ignition (DISI) engine. Variations observed concerning global mixture behavior as well as correlations between tumble flow and local fuel/air ratios are then analyzed. Local variations and cyclic fluctuations of the laminar flame speed in the vicinity of the spark plug will be discussed. This is of major importance, since the laminar flame speed controls flame kernel development in the early stage of ignition and combustion [5]. Finally, different regions of interest have to be analyzed carefully and modeled with high accuracy before obtaining a valid CFD model, able to simulate and predict correctly several complete in-cylinder flow cycles including injection and mixture formation.

A key feature of this CFD model is the usage of a hybrid turbulence model, the so called Scale-Adaptive Simulation (SAS) model. Classical turbulence models based on the Reynolds decomposition (RANS) with an eddy viscosity model lead to a calculated turbulent viscosity up to a hundred times higher than that suitable for resolving turbulent structures important for detailed in-cylinder flow analyses [6]. In the past, only cycle-averaged or integral values were considered, for which time or ensemble-averaged simulation results are sufficient. More recently, simulations relying on Large Eddy Simulations (LES; see e.g.,

[7, 8, 9]) have been increasingly considered in the academic research community. A good and still up-to-date overview is given by [10]. With LES, when using an eddy viscosity based subgrid-scale model, resulting values of the turbulent viscosity are at levels low enough for resolving turbulent structures much smaller than large-scale in-cylinder vortices (responsible for instance for tumble or swirl motion). However, standard LES simulations are often prohibitive in terms of computing time due to restrictions regarding the numerical mesh and numerical settings, which can influence the stability behavior of the solving process. A number of different alternative scale resolving turbulence models were developed in the past and used for incylinder flow modeling. This includes hybrid models like DES, DDES, IDDES, SDES or SBES, which all use a more or less complex blending function to switch between pure (U)RANS formulation in the near wall zone and LES turbulence modeling in the free stream part of the computational domain based on local grid information and wall distance. These models were successfully used (e.g. by [11, 12, 13]) to capture cyclic fluctuations in the flow field and to show the influence on turbulent combustion.

The need for easy to use turbulence models, which resolve as much turbulence as possible with the given numerical mesh resolution and time step size forced the development of additional global scale resolving turbulence models. Often, these models are not classified as pure LES models, since there is not a direct subgrid scale filter implemented, but the results show, that they are able to resolve nearly as much turbulence, compared to pure LES models. [14, 15] used a so called Partially-Averaged Navier-Stokes Simulation (PANS) model, which is originally based on a RANS $k-\epsilon$ -model. The eddy viscosity is calculated using additional transport equations for unresolved turbulence quantities (kinetic energy and dissipation rate). This approach was extended by [16] and was converted to the transport equation of the RANS $k-\zeta-f$ model. The adjustment of the eddy viscosity to resolve a higher amount of turbulence is done by using the ratio of grid size and integral length scale. This results in a flow field dependent value for the eddy viscosity. [15] showed, that PANS is able to produce more accurate flow fields and swirl numbers in comparison with a RANS- $k-\zeta-f$ model in a portflow application. Simulation results were compared with PIV and flow bench measurements. [14] showed, that the PANS model is able to resolve a certain part of the turbulence, but show an instable behavior in an application in real engine geometries. This behavior was explained by the highly decreased resolution parameter f_k , which leads to very small eddy viscosities in the valve gap. [17] developed a new scale resolving turbulence model, the Dynamic Length Scale Resolution Model (DLRM), which dynamically adapts the eddy viscosity as a result of a comparison between modelled and potentially resolvable length and timescales. The latter are estimated using grid and time step size information. In [17] a simplified engine geometry with a fixed, central positioned intake valve was used as a validation test case. DLRM simulations on a coarse (700 K cells) and fine (4 M cells) grid size were compared to Large Eddy Simulations using a WALE subgrid scale model on a 12 M cells mesh and with LDA measurements. The DLRM results showed a remarkable agreement in averaged velocities and rms fluctuations, partially with better agreement with the experimental data than the LES-WALE model.

An interesting alternative for predicting the transient in-cylinder flow in a similar context is the Scale-Adaptive Simulation (SAS) turbulence model. This turbulence model was first described by Menter [18, 19, 20] in 2003. It was shown that SAS delivers additional, fine-scale flow information like for a LES turbulence model, and leads to improved comparisons with experimental results compared to classical RANS models, like k- ω -SST. Previous studies with geometries close to real ICE confirmed the promising outputs obtained with SAS [12, 21].

The given examples regarding the usage of dynamically adaptive scale resolving turbulence models show, that they are interesting and already established alternatives to classical LES models. Time depended meshes and the time step sizes used in incylinder flow simulations are not always sufficient for pure LES models. The adaptive behavior of the discussed adaptive SRS models, regarding given mesh and time step sizes, makes them favorable for scale resolving incylinder flow simulations.

The final goal in this study is to develop a CFD model and a modeling and analyzation strategy that allows detailed analyzation of cyclic combustion variability for industrial applications. This includes the usage of an adaptive SRS model, which will be validated against PIV measurements. This ensures the prediction capabilities of the chosen SRS model in the specific case of this study.

NUMERICAL MODELS

Scale-Adaptive Turbulence Model

Most RANS-based turbulence models use an eddy viscosity approach, where the influence of the non-resolved turbulent motion on the time-or ensemble-averaged flow field is described using an assumption of turbulence homogeneity and isotropy. The unknown Reynolds stress tensor resulting from the Reynolds decomposition is then calculated based on the (known) mean velocity gradient tensor and on the eddy viscosity:

$$-\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right) - \frac{2}{3} \left(\rho k + \mu_t \frac{\partial \overline{u}_k}{\partial x_k} \right) \delta_{ij} \quad (1)$$

A common approach to estimate the eddy viscosity is to use at least one variable describing turbulence, which is calculated via its own transport equation. Nevertheless, most turbulence models currently employed for engine simulation use two-equation models, namely k- ϵ , k- ω -SST or k- ζ -f. The SAS turbulence model used in this study is originally based on the k- ω -SST model [22], so that the eddy viscosity is calculated following:

$$\mu_t = \frac{\rho k}{\omega} \frac{1}{\max \left[\frac{1}{\alpha^*}, \frac{SF_2}{\alpha_1 \omega} \right]} \quad (2)$$

Here, α^* is a low-Reynolds number correction and results in unity for high-Reynolds number flows; α_1 is a model constant ($\alpha_1=0.31$), F_2 is a blending function of the SST extension and S is the magnitude of the strain rate.

The difference between the standard k- ω -SST and the SAS model is a source term in the transport equation for the turbulence dissipation frequency (ω -equation). This important term involves the ratio between resolved and unresolved turbulent length scale, based on the von Karman length scale. The latter is derived from an exact transport equation describing the turbulence length scale. Further details are described in [18, 19]. Finally, the source term is defined as follows:

$$Q_{SAS} = \max \left[\rho \eta_2 \kappa S^2 \left(\frac{L}{L_{vK}} \right)^2 - C \frac{2\rho k}{\sigma_\Phi} Z_1, 0 \right]$$

$$Z_1 = \max \left(\frac{1}{\omega^2} \frac{\partial \omega}{\partial x_j} \frac{\partial \omega}{\partial x_j}, \frac{1}{k^2} \frac{\partial k}{\partial x_j} \frac{\partial k}{\partial x_j} \right) \quad (3)$$

with the model constants $\eta_2=3.51$, $\sigma_\Phi=2/3$ and $C=2$. The integral length scale L is defined as

$$L = \frac{\sqrt{k}}{\left(c_\mu^{1/4} \omega \right)} \quad (4)$$

with the model constant $c_\mu = 0.09$. The von Karman length scale L_{vK} is calculated via

$$L_{vK} = \kappa \frac{\sqrt{2 \cdot S_{ij} S_{ij}}}{\sqrt{\left(\frac{\partial^2 \overline{u}_i}{\partial x_j^2} \frac{\partial^2 \overline{u}_j}{\partial x_i^2} \right)}}, \quad S_{ij} = \frac{1}{2} \left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right) \quad (5)$$

The comparison of both length scales in this model results in a reduction of the eddy viscosity, when the formation of a resolved turbulent spectrum is detected by the SAS model, for small values of the von Karman length scale. Therefore, the model can act in a “scale-resolving” or “pseudo-LES” mode. This is particularly true for strongly unsteady flows, where fast variations in time can be introduced via boundary or initial conditions, or can develop spontaneously through non-linearity effects appearing in the Navier-Stokes equations in combination with complex geometric features. In the case of ICE, flow separation at the intake valves can trigger the model [1]. The potential of SAS for in-cylinder flow simulations has been already confirmed by different research groups [12, 21].

Engine Model - Cold Flow

The simulation domain is a full four-valve pent-roof combustion chamber with the characteristic bore of a passenger car. It incorporates the intake and exhaust ducts, which are numerically resolved up to 180 mm upstream the intake valves and 80 mm downstream the exhaust valves. Additional information is given in [table 1](#).

Table 1. Datasheet of the research engine used for CFD simulations

Name	value
Stroke	80 mm
Bore	74.5 mm
Connecting rod	144 mm
Compression ratio	7:1
Number of valves	4
Exhaust valve open	18.4° BBDC @ 0.2 mm lift
Exhaust valve close	43.2° ATDC @ 0.2 mm lift
Inlet valve open	42.1° BTDC @ 0.2 mm lift
Inlet valve close	13.5° ABDC @ 0.2 mm lift

The size of the numerical mesh employed in this study (see [Figure 2](#)) is 3.6 million cells in TDC and 6.3 million cells in BDC. The ICE Tool of the ANSYS Workbench was used for meshing after adapting it to the problem at hand. A hybrid meshing approach was used for the dynamic mesh in the CFD solver. To improve the accuracy of the prediction concerning flow separation at the intake valve, the mesh density was increased in this region. The average cell size was set there to 0.2 mm. In order to resolve a sufficient part of the turbulent spectrum, the cell size in the combustion chamber was set to 0.6 mm. The maximum cell size was limited to 2 mm. This value was only reached in the intake and exhaust ducts.

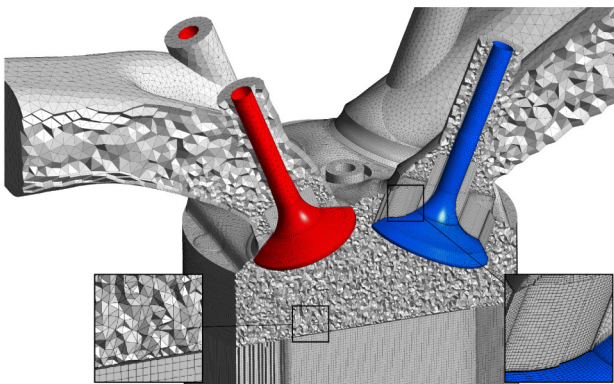


Figure 2. Employed numerical mesh consisting of ~ 3.6 million cells (TDC) and ~ 6.3 million cells (BDC).

The boundary conditions for the CFD simulation were taken from experimental data of individual cycles measured in the intake- and exhaust ducts of the test-bench engine, 120 mm upstream the intake valves and 120 mm downstream the exhaust valves (see [figure 3](#)). The spatial distances between these measurement locations and numerical boundaries were found to have only minor influence on the incylinder flow field. Assuming that the pressure wave moves with the speed of sound (roughly 340 m/s), the spatial distance between the numerical boundary and the real location of the pressure signal

measurements results in a shift of the pressure boundary signal of 1.5 CAD at 2000 rpm. The benefit of the chosen position of the inlet boundary is an increased duct volume, so that the influence of the scavenging air and resulting EGR (in combustion simulations), based on camshaft timings, can be directly resolved.

For each simulated cycle, specific boundary data have been used in terms of pressure curves (see [figure 3](#)). The COV of the intake and exhaust pressure curves are always below 0.5 % (see [table 2](#)). In combination with SAS, this modeling strategy leads to cycle-dependent flow fields as simulation results. Preliminary (U)RANS calculations with a $k-\omega$ -SST turbulence model showed only minor differences in the cylinder flow field. It can be stated, that the differences in the cycle dependent boundary conditions are not high enough to overcome the effects of the high turbulent diffusion of a common (U)RANS model. Since no turbulent fluctuations are introduced at the inlet boundary, an additional benefit of these kind of boundary conditions is simple implementation, when measurements are available.

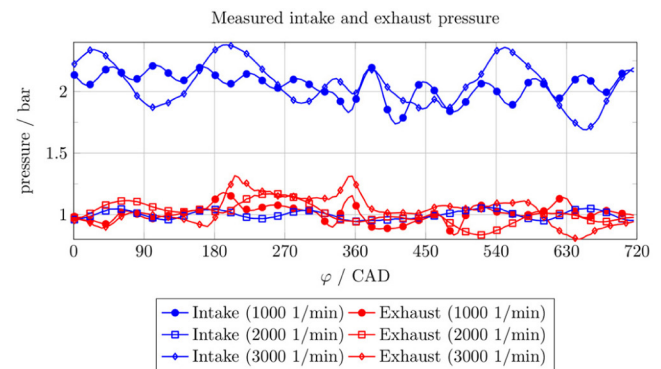


Figure 3. Example of pressure curves (cycle average of 200 cycles) measured at the intake and exhaust ducts of the test-bench engine, used as boundary conditions in CFD.

Temperatures for inlet and outlet boundary conditions were measured 180 mm upstream the intake valve and 180 mm downstream the exhaust valve. The measurements were not crank angle resolved, so that the inlet and exhaust boundary condition were set as constant values. Surface temperatures at the different boundaries (e.g. intake and exhaust ducts, cylinder head, piston, liner) were set constant with values between 303 K and 383 K. Initialization was done by first simulating a preliminary cycle with the use of a RANS- $k-\omega$ -SST turbulence model. Starting from this point, a different amount of multicycle simulations were started, each consisting of six consecutive cycles, whereas the first cycle was always dropped to ensure independence from initially prescribed values. The amount of the multicycle simulations is different for the analyzed operating points, resulting in different amounts of total cycles to analyze (see [Table 2](#)). For operating point OP 2, a total number of 30 cycles was reached by using six multicycle simulations. The dynamic mesh approach of FLUENT was used to describe the moving boundaries (piston and valves).

Multiple cycles of three different non-fired operating points have been simulated, compared and validated with experimental data measured at the test-bench engine. Data for the validation process

include incylinder pressure data measured in the combustion chamber, and cycle-averaged as well as rms values of the velocity, measured with High-Speed Particle Image Velocimetry (HS-PIV). Simulation and experimental results from HS-PIV measurements have been interpolated on congruent 2D post-processing grids for accurate analysis.

Three different operating points have been simulated with several individual cycles in order to gather statistics, focusing mainly on operating point 2. More details can be found in Table 2. The computational time needed for one cycle can be estimated by 4600 CPUh on Intel Xeon-E5 Cores. Comparisons with the same setup using DDES and WMLES turbulence models showed, that the convergence performance of the SAS model overcompensated the additional computation time for the additional transport equations respective more complex turbulence modeling procedure. Since convergence behavior and simulation stability was a major aspect, the decision was in favor of the SAS model.

Table 2. Description of the simulated operating points

Operating Point	Engine speed / 1/min	p_{intake} / bar	$COV_{\text{intake/exhaust}}$ / %	No. cycles
OP 1	1000	2	0.13 / 0.21	10
OP 2	2000	1	0.09 / 0.11	30
OP 3	3000	2	0.14 / 0.48	10

Spray Model for Fuel Injection

For simulating injection of gasoline fuel the FLUENT Discrete Phase Model (DPM) was used. This is a Lagrangian approach classically employed for modeling of diluted liquid or solid secondary phases. For a detailed description, see [23, 24, 25].

To model injection, spray breakup, and evaporation, different submodels were used. The injection process is modeled by using a blob injection model with a uniform diameter equal to the nozzle hole diameter. Primary and secondary breakup were modeled using the KHRT model [25]. This model considers Kelvin-Helmholtz instabilities at the surface of the liquid core of the spray to initialize breakup. This results in the generation of small droplets and ligaments forming out of the initial blobs. Additionally, Rayleigh-Taylor instabilities are modeled, generating smaller droplets in the diluted part of the spray. Turbulence production or damping at subgrid scale through droplets smaller than the grid size is considered using an implemented model described in [26].

In all simulations, a single-component surrogate was used instead of a multi-component fuel, which is a major but often used simplification. Gasoline is a mixture of more than hundred single hydrocarbons, which all have different vapor pressures. To describe evaporation [27], iso-octane was used as single component, since its vapor pressure lies near the average vapor pressure of the main components of gasoline fuel (see Figure 4). The properties of the fluid are computed using standard databases [28].

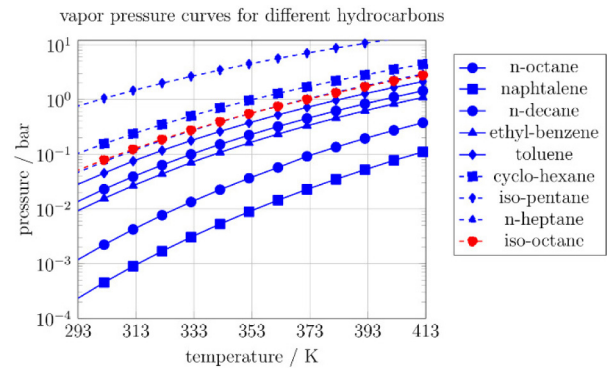


Figure 4. Vapor pressure of the main components of RON 95 gasoline fuel.

The injected mass flow-rate was evaluated experimentally. The injection velocities were calculated using mass conservation in the injector nozzle and assuming an incompressible fluid. The validation of the spray model was done using experimental data from measurements at an optically accessible pressure chamber. The geometrical features of the chamber were used to setup a CFD model for the gasoline injection process. Boundary conditions for temperature and initial values for the chamber pressure and temperature were set based on measured experimental values. Since there is no data available from the experiment regarding initial turbulence values in the chamber, values from a similar case [29] were applied. The numerical results were then compared with measurements. Available experimental data include High Speed Photography (HSP) and Phase-Doppler Anemometry (PDA) measurements delivering particle size distributions at 40 mm distance from the nozzle hole. The HSP data was post-processed to obtain the tip penetration length over time. Different operating points were analyzed, representing a wide range of possible engine operating points. In order to predict spray penetration and evaporation correctly, three operating points were used to fit the model, as described in Table 3.

Table 3. Operating points for spray injection in a constant volume chamber

Operating Point	p_{chamber} / bar	T_{chamber} / K	p_{fuel} / bar	T_{fuel} / K	ρ_{gas} / kg/m ³
OP _{spray} 1	2.10	373	200	363	1.961
OP _{spray} 2	0.62	373	200	363	0.579
OP _{spray} 3	0.85	373	200	363	0.794

RESULTS

Validation of the Engine Cold Flow Model

In-Cylinder Pressure

This first validation was done by comparing simulation results and measurements of the in-cylinder pressure and the velocity field in different planes within the combustion chamber.

As seen in Figure 5, there is a good agreement between prediction and measured in-cylinder pressure. Both results match very well during the gas exchange and early compression phase. In the timespan near the top dead center (TDC), there is a slight difference in comparison with the experiment. This can be explained by a loss of mass (blow-by) in the experimentally employed combustion chamber. At higher engine speeds, blow-by increases. The presented in-

cylinder model is optimized for operating point 2 (2000 rpm / 1 bar intake pressure), by adjusting compression ratio through a shift of the piston position to obtain the same peak pressure in TDC. The slight overshoot of pressure in the simulation results at TDC in OP 1 and the corresponding undershoot in OP 3 can be explained by the different engine speeds, since blow-by is a function of time, not of crank angle. The blowby of the real engine is decreasing with higher engine speeds, so that the pressure difference in comparison with the simulation is negative for high engine speeds, and vice versa for lower engine speeds. Blow-by is, in addition to wall heat transfer, also the reason why the maximum pressure is shifter to earlier crank shaft positions in the experiment. However, and in agreement with the scientific literature [30], the impact of blow-by on the velocity field is assumed to be minimal, so that blow-by is not considered at all in the CFD model.

The effect of blow-by can also be observed during the late expansion phase, when the measured in-cylinder pressure is lower than the volume-averaged in-cylinder pressure in the simulation. After the exhaust valves open (EVO), the pressure in the simulation drops to levels nearly identical to the measurements. In the gas exchange phase the pressure curves match again very well. At IVC the averaged pressure deviation between simulation and experiment is below 0.025 bar, which indicates, that nearly the same mass is contained in the combustion chamber. Possible differences concerning the really trapped mass can occur due to missing information concerning temperature at the boundaries. Since the optical engine has fixed operating temperatures for engine oil and cooling water, wall temperatures values were chosen based on in-house calculations and previous experience. Considering the obtained agreement, it is confirmed that this procedure works very well.

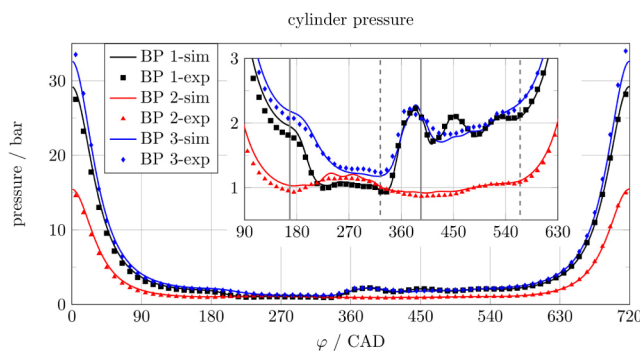


Figure 5. Comparison of the experimentally measured in-cylinder pressure with the simulated, volume-averaged pressure within the combustion chamber. Exhaust and inlet valve timings are marked (EV = solid line, IV = dashed line).

Velocity Field

Comparisons of the cycle-averaged velocity fields and of the rms values of velocity were carried out next. In order to understand the experimental procedure, a short description of the setup at the transparent test bench engine (see [figure 6](#)) is given first.

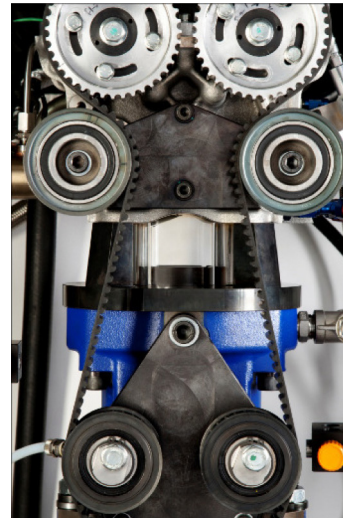


Figure 6. Optically accessible research engine with pancake piston and glass liner for High Speed Particle Image Velocimetry measurements

The High-Speed PIV System consists of two lasers (Quantronix Hawk II and Quantronix Hawk Pro) with an output wavelength of 532 nm. The images were captured with a LaVision HighSpeedStar 5 camera with a resolution of 512×512 pixel. The acquisition frequency was 2640 Hz at OP 1/OP 2 and 3960 Hz at OP 3. The time delay between one double image varied between 5 and 15 μ s. Peanut oil was used as seeding particles, which delivers best performance regarding atomization, flow following and vaporization behavior compared to other seeding substances like plant oil, magnesium oxide and propylene carbonate. The smoke number is above 433 K. In combination with a relatively low compression ratio of 7:1, vaporization occurs only at the time span around TDC. The seeding density was still high enough to calculate velocity vectors. The peanut oil was introduced and atomized 1 m upstream of the cylinder head. The raw images were post-processed with the commercial software LaVision Davis 8.16. For validation, images were analyzed every five CAD in 74 consecutive cycles from 450 CAD b. TDC to 30 CAD a. TDC.

The engine setup allows measurements as either front or side view through the glass liner, or as bottom view through the pancake piston. A schematic illustration of the optical setup can be seen in [figure 7](#).

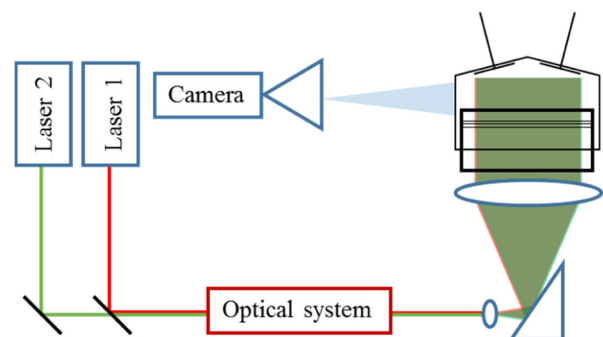


Figure 7. Schematic illustration of the High Speed PIV optical setup. A vertical measuring plane has been selected for this example.