Computational Analysis of Flow and Mixture Formation in DISI Engines

Priv. -Doz. Dr.-Ing. Philipp Adomeit, Dipl.-Ing. Oliver Lang
FEV Motorentechnik Aachen

Dipl.-Ing. Betina Vogt
Lehrstuhl für Verbrennungskraftmaschinen VKA, RWTH Aachen

1. Introduction

Direct Injection Spark Ignition (DISI) engines have a significant potential to reduce specific fuel consumption and CO2 emissions. In part load conditions DISI engines are most favourable operated in stratified charge mode, which combines the advantages of reduced pumping losses and lean combustion. In full load operation the in-cylinder charge is cooled by fuel evaporation. This reduces knock sensitivity and enables higher specific power output.

The above advantages are paid by a significant increase of combustion system complexity. In stratified operation the mayor challenge is the optimization of mixture formation. Several approaches are documented in the literature [1,2]:

1. Spray guided combustion systems, where the spray is directly targeted towards the spark plug with enlarged protrusion depth.
2. Swirl combustion systems, where swirl is used for mixture preparation. Frequently the spray is deflected towards the spark plug by the piston bowl.
3. Tumble combustion systems, where the in-cylinder charge motion is utilized for spray deflection and mixture preparation. These systems can be realized either air- or wall-guided.

In the present work air-guided tumble combustion systems are investigated. The tumble intensity is controlled by a continuous variable tumble system (CVTS) which enables controlled guidance of the fuel penetration towards the spark plug. An schematic of the combustion chamber geometry is shown in Figure 1.
In the air-guided DISI engine the mixture preparation is determined by both in-cylinder charge motion and injection, which interact in a complex manner depending on geometric and injection parameters. Hence, optimisation requires quantitative knowledge of the interaction of the individual parameters of influence.

The quantitative description of these processes requires the use of multidimensional modeling with numerical simulation tools. The quality of the prediction is directly dependent on the accuracy of the numerical flow and the injection model.

2. Numerical

The numerical calculations are performed using the program StarCD, which provides a methodology for moving meshes. Fully hexahedral meshes of intake and exhaust ports and combustion chamber are used here. Piston and valve motion are performed by cell activation and deactivation and supported by vertex motion routines. The full intake and compression cycle is accounted for in the CFD calculations.

The two-phase flow of gas and droplet phase is modeled by a lagrangian approach, which treats the droplets as a disperse phase, whose interaction with the continuous gas phase is determined by a two way coupling via adequate source terms. This approach has become a standard in fuel injection modeling and has been described and validated in a large number of publications.
3. Primary atomization model

The injection characteristics are of major importance for the mixture preparation. In the CFD calculation this is determined by the properties of the primary droplets. Hence a quantitative description of the primary droplets is a necessary condition for an accurate simulation of the momentum, energy and mass exchange between liquid and gas phase in the combustion chamber.

A direct modeling of the liquid disintegration requires to account for highly complex interfacial phenomena and thus is subject of ongoing research. In the present investigation a primary atomization model for high pressure swirl injectors is used, which is based on physical considerations but also takes into account empirical information.

The fundamental considerations of the primary atomization model, which bases on the model of Han et al. [3] are displayed in Figure 2. It is assumed, that the conical liquid sheet at the nozzle exit consists of coherent initial ligaments. These initial ligaments disintegrate into primary droplets at the location \( L_B \), which is determined by linear stability considerations. The primary droplet diameter is assumed proportional to the local liquid sheet thickness.

![Figure 2: Schematic of the atomization process of a high pressure swirl injector](image-url)
The determination of the primary droplet velocity is based on continuity and momentum considerations as given by Han et al. [3]. The input parameters of the atomization model can be easily determined:

- nozzle hole diameter $d_N$
- nominal cone angle $\alpha$
- injection pressure $p_{CR}$
- injected mass $m_{inj}$
- injection duration $t_{inj}$
- gas pressure and temperature $p, T$

Furthermore, it is necessary to account for the transient spray development. Due to the initially reduced swirl in the nozzle a pre-jet develops with significantly reduced cone angle. This process has been investigated numerically by Cousin et al. [5], which have shown that the duration for establishing a fully developed swirl in the nozzle corresponds with the development of a stable conical liquid sheet. Here, the pre-jet is accounted for by empirically prescribing the duration of the pre-jet $t_{pre}$ and assuming a temporal development of the pre-cone angle, which is based on experimental spray visualization results. Since the pre-jet affects both spray penetration and spray shape due to the deteriorated atomization and the correspondingly larger primary droplets, it is absolutely necessary to describe it in a quantitative manner.

Further droplet break-up is modeled by the approach of Reitz und Diwakar [4] as implemented in StarCD. This approach distinguishes between bag break-up, occurring at moderate relative droplet velocity and size, and stripping break-up due to strong droplet deformation at high droplet velocities and / or large droplet diameters.

The above atomization model has been validated by comparison to experimental results, which are described by Adomeit et al. [6] in detail. It was shown, that the model is capable to predict effects such as nozzle hole diameter, injection pressure, gas pressure and temperature without further adjustment of model parameter. This is an important condition for a well-aimed parameter variation to optimize the mixture formation.

4. Optimization of mixture formation in the DISI engine

The following investigation aims at the determination of the interaction between in-cylinder charge motion, combustion chamber geometry and injection and to deduce
approaches for their optimization. For a given combustion system the following parameters can be varied within a certain range:

- flow parameter: controlled by the CVTS system
- injection parameter: BOI, injection pressure, cone angle, etc.
- geometry of the piston bowl

In the present investigation a variation of flow parameters has not been taken into account, since a reduction of tumble intensity does not reveal potential at the relatively low engine speeds considered here.

The results of the mixture formation analysis are evaluated by a number of criteria based on characteristic quantities of mixture preparation and ignition conditions at the spark plug location. The following quantities are taken into consideration:

- the fraction of evaporated fuel $m_v / m_f$ which is a direct measure for the speed of mixture preparation
- the distribution of the fuel vapor on the relative air / fuel ratio classes $m_v = f(\Delta \lambda)$ which is a measure of the quality of mixture formation in stratified operation
- the local air / fuel ratio at the spark gap determines the ignition conditions in stratified operation
- the mass of liquid droplets in the spark gap may cause flame quenching effects due to evaporation cooling. The ratio of energy required for evaporation of the liquid droplets and the thermal energy of the ignition spark is considered as characteristic quantity here

Two different piston bowl geometries are investigated:

- Bowl A: central bowl with a slight displacement towards the outlet side
- Bowl B: enlarged bowl volume at the intake side

The compression ratio is equal in both situations as the bowl volume is kept constant in both geometries.

Results of the CFD analysis of bowl A in the operating point 2000 rpm and bmep = 2 bar are shown in Figure 3 in a temporal sequence.
Figure 3: Sequence of the mixture formation process in the air guided DISI engine with piston bowl A: CFD results (a) $p_{CR} = 60$ bar, (b) $p_{CR} = 100$ bar.

Figure 3(a) shows the spray penetration and mixture formation at an injection pressure of $p_{CR} = 60$ bar, Figure 3(b) at an injection pressure of $p_{CR} = 100$ bar. Starting point of the sequence is midtime of the injection event, ending point is the ignition time. The tumble charge motion causes a significant deflection of the fuel vapor cloud towards the spark plug. Especially with the lower injection pressure the mixture preparation can not be regarded as sufficient, since rather rich mixture is observed at the spark plug at the time of ignition.

Figure 4: Temporal development of the state of mixture preparation in the spark gap control volume with piston bowl A: (a) $p_{CR} = 60$ bar, (b) $p_{CR} = 100$ bar.
An analysis of the temporal development of the relative air/fuel ratio in a control volume at the spark gap gives a quantitative insight into this phenomenon and is shown in figure 4. At $p_{CR} = 60$ bar the mean and maximum value of the relative air/fuel ratio $\lambda$ is in the ignitable range. However, parts of the control volume are filled with very rich mixture below the lower ignition limit.

Raising the injection pressure to $p_{CR} = 100$ bar improves the mixture formation. This is partially caused by the earlier injection timing which becomes necessary to compensate for the enhanced spray penetration. Additionally, earlier injection causes a wider spray due to the smaller gas pressure. A further consequence is that ignition time has to be $7^\circ$CA earlier than at $p_{CR} = 60$ bar.

![Figure 5: Temporal development of the evaporated fuel mass in the entire combustion chamber: CFD results of piston bowl A](image)

At higher injection pressure the evaporation rate is enhanced due to improved atomization as seen in Figure 5. At the ignition point more than 86% of the fuel is evaporated at $p_{CR} = 100$ bar whereas it is only 72% at $p_{CR} = 60$ bar.

The quality of the mixture is shown in Figure 6 which displays the fuel vapor mass distribution. It is seen that rich mixture is more prominent at $p_{CR} = 100$ bar which is due to the enhanced evaporation rate.

The enlarged spray width at $p_{CR} = 100$ bar additionally causes a higher concentration of droplets in the vicinity of the spark gap. Figure 7 reveals that the energy necessary to evaporate these droplets is higher than the thermal spark energy shortly after end of activation EOA. Subsequently the droplet concentration at the spark gap diminishes and reaches a non-critical level at ignition point IP.
The effect of the piston bowl geometry on the penetration of the fuel vapor cloud is investigated by comparing the previous results to those of piston bowl B which are discussed in the following. The tumble generation during the intake phase remains almost identical. However, the bowl modification reduces the spray deflection due to the tumble charge motion in the late compression phase, as seen in Figure 8 (a). As a consequence the spray penetration into the piston bowl increases. Since spray penetration is strongly dependent on the pre-jet, the role of the pre-jet is investigated additionally by considering a hypothetical variant with reduced pre-jet. These results are shown in Figure 8 (b).
The state of mixture preparation in the spark plug control volume is seen in Figure 9. At the ignition point the mixture at the spark plug is less rich compared to piston A. With a fully developed pre-jet parts of the spark control volume are filled with very lean mixture above the upper ignition limit. With a reduced pre-jet this effect is avoided due to the larger radial spreading of the fuel vapor cloud. This effect is obvious by comparison of the shape of the fuel vapor cloud in figure 8 (a) and (b).
The effect of the piston bowl geometry on the mixture formation itself is rather weak. The evaporated fuel fraction at ignition point is slightly reduced from 72 % with bowl A to 68 % with bowl B, as seen in Figure 10. The standard fully developed pre-jet initially causes a slightly retarded evaporation which is a consequence of both the deteriorated atomization and the reduction of the effective spray angle.

![Figure 10: Temporal development of the evaporated fuel mass in the entire combustion chamber: CFD results of piston bowl B](image)

The effect of the pre-jet on the overall quality of the mixture formation process is rather weak. The fuel vapor mass distribution in Figure 11 shows that at early stages the reduced atomization in the pre-jet leads to a slightly richer mixture at early stages, 10°CA b. IP. However, at the ignition point IP almost identical vapor mass distributions are observed.

![Figure 11: State of the mixture preparation in the entire combustion chamber, piston bowl B: (a) standard injection, (b) reduced pre-jet](image)
Due to the reduction in spray deflection by piston bowl B, the concentration of liquid droplets at the spark plug remains independently of the pre-jet intensity below a critical level, as seen in Figure 12.

![Diagram showing energy required for droplet evaporation and thermal spark energy](image)

**Figure 12:** Temporal development of the heat of evaporation of liquid droplet mass in the spark gap control volume: CFD calculation of piston bowl B

All reported variants except the reduced pre-jet have been investigated experimentally on an engine test bench. Regarding the correlation between CFD results and experimental engine results, three most important findings are pointed out here:

1. The CFD calculation shows a rather rich mixture at the spark plug with piston bowl A at $p_{CR} = 60$ bar. This corresponds to the experimental observation of non-existing ignition window without misfiring.
2. Increasing the injection pressure to $p_{CR} = 100$ bar improves the mixture preparation in the spark volume with piston bowl A. This correlates to the experimental observation of an enlarged ignition window ($\Delta \alpha > 10^\circ$CA) without misfiring. However, the required earlier ignition is thermodynamically not optimal and leads to an increase in specific fuel consumption.
3. Piston bowl B improves the mixture preparation state in the vicinity of the spark plug significantly even $p_{CR} = 60$ bar. In the engine operation this correlates to a significantly enlarged ignition window ($\Delta \alpha > 20^\circ$CA) and a reduction in specific fuel consumption.

### 5. Conclusions

The interaction of charge motion, combustion chamber geometry and spray penetration is investigated by CFD calculations to deduce measures for optimization of stratified operation of DISI engines. Important characteristics of the mixture formation are identified, which are the vapor cloud penetration, the evaporation rate
and the degree of homogenization. The use of an validated injection model for high pressure swirl injectors [6] enables quantitative predictions of the in-cylinder mixture formation, which correlate well to engine test results.

As a conclusion it can be stated that numerical simulation by CFD not only improves the general understanding of the complex interaction of flow, injection and engine geometry, but also allows a detailed optimization of important parameters such as piston bowl geometry and injection pressure and timing. Multidimensional modeling by CFD calculation increasingly gains the ability to support both the efficient optimization of a given engine concept and feasibility studies of completely new concepts.

6. Literature


