Ansys

CASE STUDY /

Ansys + STFS

"In our long-standing collaboration with Ansys in the field of combustion research, we have been able to build a robust simulation framework for combustion engines at our institute. The framework is largely based on customized solutions for engine simulation from Ansys CFX, which allow us to focus on the essential points of our research. Without this foundation, we would not be able to address current environmental challenges, such as climate change and increasing pollution, through sustainable combustion technology."

Prof. Dr.-Ing Christian Hasse
Head of the Institute / STFS — Technical University of Darmstadt

CASE STUDY

Institute for Simulation of reactive Thermo-Fluid Systems

The internal combustion engine is still considered as the most widely used propulsion system in the mobility sector. Nevertheless, particulates emitted from this method pose a health threat to the human body. They can cause respiratory and cardiovascular diseases, and are therefore strictly limited. Since these limits will likely be even stricter in the future, the design of next-generation engines requires a detailed understanding of the relevant physical and chemical phenomena inside the engine.

Challenges

Real Driving Emissions (RDE) measurements taken in real road traffic scenarios show that a large proportion of particulates emitted into the atmosphere during driving can be attributed to individual events. These events are characterized by highly transient engine operation and are strongly influenced by engine conditions, the vehicle's driver, and environmental factors. To further reduce particulate emissions from the vehicle fleet in the future, it is necessary to focus on the emission peaks that occur.

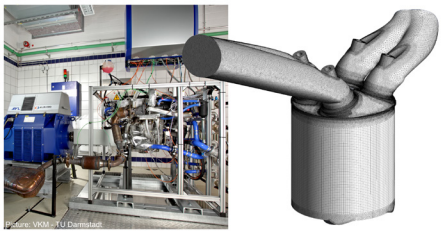

Figure 1: Combined experimental and numerical approach to investigate soot emission sources. Boundary conditions from a measured transient engine event are transferred into the virtual engine model for comprehensive 3D CFD calculations.

Ansys Products Used:

- Ansys CFX
- Ansys ICEM CFD

Engineering Solution

To investigate sources for particle emissions during RDE relevant transient engine operation, we worked together with our experimental partners to apply a combined experimental and numerical approach. Provided with detailed boundary conditions of the transient event, we utilized Ansys CFX coupled with our moment-based soot modeling approach to perform 3D computational fluid dynamics (CFD) in-cylinder simulations of consecutive transient engine cycles and collect insights about underlying emission-promoting effects. The tools and models for remeshing, multiphase flows, and combustion, together with additional engine specific models implemented in Ansys CFX, were crucial for this investigation.

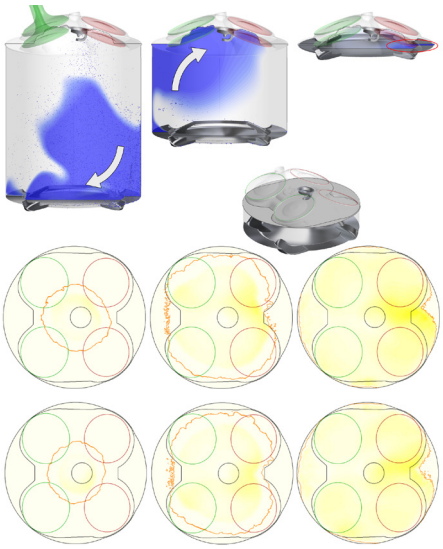

Figure 2: Mixture homogenization, combustion, and formation of particulate matter. The simulation methodology applied can represent the entire cause-and-effect-chain of the investigated engine cylinder. This makes it possible to identify potential sources of increased soot emissions over the course of several engine cycles.

Benefits

The simulation methodology using Ansys CFX allowed us exclusive access to the processes inside a state-of-the-art series combustion engine beyond the possibilities of any conventional measurement technique. We were able to identify several causes for increased soot particle emission in the investigated events. Further simulations suggest that a reduction potential by a factor of eight is possible for selected engine cycles.

Company Description

The Institute for Simulation of reactive Thermo-Fluid Systems (STFS) of the Technical University of Darmstadt focuses on modeling and simulating thermo-fluid processes in mechanical and chemical process engineering.

In the field of combustion engine research, the institute has been successfully collaborating with Ansys Germany for many years, which led to a significant number of publications and international visibility.