6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11 – 15 June 2018, Glasgow, UK

Application of CFD Simulation to Optimize Combustion in Prechamber Gas Engines with Port Injection

Hubert Winter*[1], Eduard Schneßl[1], Gerhard Pirker[1], Jan Zelenka[1] and Andreas Wimmer[12]

[1] LEC GmbH, Inffeldgasse 19, Graz, Austria, hubert.winter@lec.tugraz.at, http://www.lec.at
[2] Institute of Internal Combustion Engines and Thermodynamics, Inffeldgasse 19, Graz University of Technology, Graz, Austria, wimmer@ivt.tugraz.at, https://www.ivt.tugraz.at

Key Words: *Gas Engine, Prechamber combustion concept, Port Injection, Flame Propagation, Knocking.*

The combustion engine technologies of the future must reduce fuel consumption and CO_2 emissions while maintaining high power density. In addition, large stationary gas engines have to meet high demands with respect to emission legislation, fuel flexibility and robustness. Cylinder-specific fuel gas admission (port injection) allows lean burn gas engines to meet these requirements. To further increase efficiency, it is key to avoid knocking combustion. With port injection, high efficiencies can be accomplished, but at the same time a high level of mixture homogeneity must be ensured.
This paper describes the use of CFD simulation to investigate and optimize combustion in a large prechamber gas engine. The focus is on a methodology that enables the evaluation of different mixing concepts and the interaction between the pre-chamber and main combustion chamber. Due to the three-dimensional nature of the mixing phenomena, CFD simulation is critical to understanding and optimizing these processes. Calculation methods and models are discussed and a methodology for the detailed assessment of the quality of mixture homogeneity is presented.
The CFD methodology is verified by assessing the combustion and knock behavior of a single cylinder research engine that includes optical combustion diagnostics. The flame front propagation is recorded by a large number of fiber optic sensors in the prechamber and main combustion chamber. A tomographic post-processing routine calculates local flame intensities, flame distribution in relation to crank angle degree and probable knock locations in the main combustion chamber.
Finally, the paper compares analyzed and simulated flame propagation in the prechamber and the main combustion chamber and presents the knocking locations identified in several example cases. The results confirm the statements derived from CFD simulation. It is shown that the application of CFD simulation is a valuable tool in the predesign of prechamber gas engines that contributes to a thorough understanding of the combustion process at the operating limits.