Numerical Analysis of Heat Transfer in Water Channels of the Opposed-Piston Diesel Engine

Authors : Michal Bialy, Marcin Szlachetka, Mateusz Paszko

Abstract : This paper discusses the CFD results of heat transfer in water channels in the engine body. The research engine was a newly designed Diesel combustion engine. The engine has three cylinders with three pairs of opposed pistons inside. The engine will be able to generate 100 kW mechanical power at a crankshaft speed of 3,800-4,000 rpm. The water channels are in the engine body along the axis of the three cylinders. These channels are around the three combustion chambers. The water channels transfer combustion heat that occurs the cylinders to the external radiator. This CFD research was based on the ANSYS Fluent software and aimed to optimize the geometry of the water channels. These channels should have a maximum flow of heat from the combustion chamber or the external radiator. Based on the parallel simulation research, the boundary and initial conditions enabled us to specify average values of key parameters for our numerical analysis. Our simulation used the average momentum equations and turbulence model k-epsilon double equation. There was also used a real k-epsilon model with a function of a standard wall. The turbulence intensity factor was 10%. The working fluid mass flow rate was calculated for a single typical value, specified in line with the research into the flow rate of automotive engine cooling pumps used in engines of similar power. The research uses a series of geometric models which differ, for instance, in the shape of the cross-section of the channel along the axis of the cylinder. The results are presented as colourful distribution maps of temperature, speed fields and heat flow through the cylinder walls. Due to limitations of space, our paper presents the results on the most representative geometric model only. Acknowledgement: This work has been realized in the cooperation with The Construction Office of WSK 'PZL-KALISZ' S.A. and is part of Grant Agreement No. POIR.01.02.00-00-0002/15 financed by the Polish National Centre for Research and Development.

Keywords : Ansys fluent, combustion engine, computational fluid dynamics CFD, cooling system
Conference Title : ICAMAME 2018 : International Conference on Aerospace, Mechanical, Automotive and Materials Engineering
Conference Location : Kyoto, Japan
Conference Dates : April 26-27, 2018