

# Battery Pack Temperature Distribution Simulation with COMSOL and MATLAB®

M. Masomtob<sup>1</sup>, C. Schaeper<sup>1</sup>, W. Zhou<sup>1</sup>, Dirk Uwe Sauer<sup>1</sup>

<sup>1</sup>Electrochemical Energy Conversion and Storage Systems Group, Institute for Power Electronics and Electrical Drives (ISEA), RWTH Aachen University, Aachen, Germany

## Abstract

This work is to present an implementation of Finite Element Method (FEM) for the cooling water system of a battery pack based on the physics in COMSOL. The M-file of a model of the battery module from COMSOL 4.2 was exported to run in MATLAB® and managed by using another M-file. The advantage of running FEM in MATLAB® R2011b is that each part in system build in COMSOL 4.2 or MATLAB® can be combined together and simulated at the same time in MATLAB®. Moreover, the results are also easy to monitor and collect in MATLAB® thanks to MATLAB®'s functionality. The CAD file in this work, built in SolidWorks® 2011 and exported to COMSOL 4.2, is 3D and includes solid and fluid parts. Therefore, not only is heat conduction used with several kinds of solids, namely, copper, thermal interface material (TIM) and aluminium plates, but also heat convection is used in fluid domain as coolant. The main boundary conditions are composed of generated heat in each single battery cell as function of time, whereas inlet coolant temperature of the first battery module is kept constant. Outlet coolant temperature of the first battery module as a function of time is then assigned for the inlet of the next battery module in series (10 battery modules). The running process is managed by another M-file that can set up some boundary and collect the data in form of .txt while the M-file from COMSOL is running. The results in this simulation are the temperatures of the water outlet of each battery module as well as the temperatures at the top and the bottom of each battery cell.