Automatic OpenFOAM Workflows for Thermal Management Applications @ MAHLE

Friedrich Brotz
MAHLE GmbH & Co. KG, Mauserstr. 3, 70469 Stuttgart, Germany
Phone: +49 711 501-47622, Email: friedrich.brotz@mahle.com

In our business unit Thermal Managment, CFD simulations on different levels of complexity are used to optimize HVAC, engine & electronic cooling components. Parametric meshing tools for fins & tubes in combination with OpenFOAM templates are applied to optimize new heat exchanger systems. Parametric tanks are generated e.g. in acquisition phase to enable early prediction of component pressure loss. For standardized applications, automatic simulation workflows are built based on OpenFOAM templates. For heat exchangers, we use such a process to enable pressure loss predictions even for non CFD experts. This tool was updated with the possibility of using not only simplified parametric geometry, but also CAD based geometry of inlet and outlet tanks. Geometries are described by closed volumes saved as *.stl from any CAD program. The solver buoyantSimpleFoam is used with a volumetric heat sink (based on local Reynolds number) in order to represent the cooldown of the fluid in the heat exchanger core. This helps to better predict the pressure loss with temperature dependent fluid properties, especially with strongly changing density of gas in charge air coolers or viscosity changes of coolant in radiators. The location of boundary conditions at the inlet & outlet or temperature at walls are based on faces defined by additional *.stl files. A Java GUI (Graphical User Interface) on Windows is used to select the *.stl files, input the core characteristics and the operating point of interest. By just a “push on a button” all input data is copied to a Linux server, batch meshing & simulation processes are started and finally, results are sent back to the Windows client by Email. Similar automated processes based on problem specific templates were adapted for other applications of thermal management. For example new dimple patterns of winglet tubes are optimized with respect to their heat transfer and pressure loss characteristics. Strong focus is placed on “Ease of Use” and robust processes. Especially high quality meshes with smooth boundary layers enable a good convergence behaviour and accurate prediction of heat transfer. With the enhanced capabilities of new OpenFOAM versions, we see strong potential to transfer more and more standardized simulation workflows from commercial solvers to OpenFOAM.

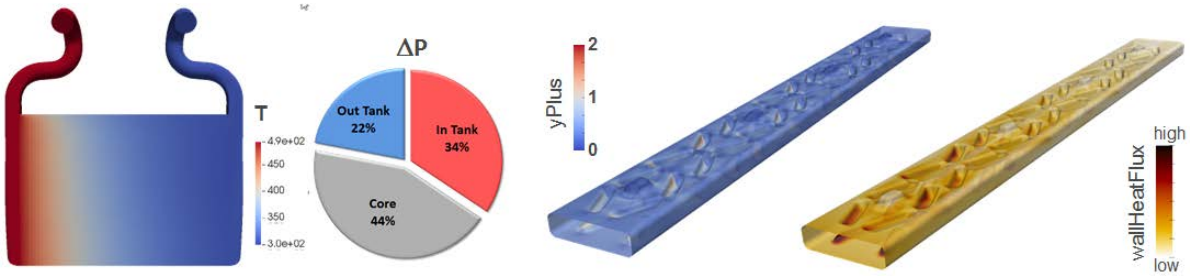

Fig. 1: Examples of Thermal Management applications (heat exchangers & winglet tubes)