

# Virtual Temperature Cycle Testing of Automotive Heat Exchangers by Coupled Fluid Structure Simulation

Friedrich Brotz and Wolfram Kühnel  
Behr GmbH & Co KG

Yang Chen  
Behr Climate Systems Inc.

Copyright © 2008 SAE International

## ABSTRACT

Temperature cycling is a major load for all automotive heat exchangers. A simulation method based on coupled fluid structure simulation was developed to analyze the thermal cycle behavior of the heat exchanger even in an early phase of the development process. The method is based on the modeling of transient heat transfer from the coolant and the air to the solid parts as well as heat conduction in the solid structure. In a first step, the temperature transients are simulated with Computational Fluid Dynamics (CFD). In a second step, the temperature distribution at specified time steps is mapped to a Finite Element Model (FEM) used for structural analysis. Compared with measurements, all relevant physical phenomena are well predicted in the simulations. This method helps to get a better understanding of the basic failure mechanisms. Finally, the calculated local stress values are the basis for subsequent lifetime prediction based on component Wöhler (S-N) curves or fatigue software tools.

## INTRODUCTION

The application of simulation tools within the product development process is getting more and more important for engine cooling components. The main reasons for this are the reduction of development time and costs compared to the traditional development process with prototypes and testing: tests such as temperature cycling of heat exchangers are very time consuming. Another important advantage of simulation tools is the comprehensive understanding gained of the mechanism for failures. Only with this knowledge it is possible to resolve the problems and to come up with improvements. This insight is more and more important to fulfill conflicting requirements like performance, weight, durability and costs.

Behr has developed a closed chain of simulation tools for the transient thermo-fluid and structural analysis of radiators, charge air coolers and exhaust gas coolers.

Computational Fluid Dynamics (CFD) delivers full information on the spatially non-homogeneous and time dependent flow and temperature distribution in the fluids, as well as the temperature distribution at the solid structure. The temperature distribution at the solid structure are transferred to a finite element analysis (FEA) model and used to simulate the stresses and strains in all points of the complete transient cycle. This enables the critical points in the structure and the critical operation modes to be found. Besides delivering key analysis results, CFD and FEA help in gaining a thorough understanding of the physics behind the behavior of the component.

Transient heat transfer analysis in a FEA code based on heat transfer coefficients evaluated with 1D and CFD simulations have been presented in [1, 2]. The focus of this paper is a method based on transient CFD conjugate heat transfer analysis with subsequent thermal stress analysis. By using CFD for the transient temperature calculation, the convective heat transfer for both fluids and the conduction in the solid parts are considered in one analysis. This is a big advantage compared to a heat transfer analysis performed in a FEA code, because the flow distribution can change dramatically in time, e.g. for radiator temperature shock conditions. With the changes in the flow distribution, the solid temperatures, especially of the heat exchanger tubes, are strongly affected. First results of this procedure and their validation have been described in [3, 4]. In this paper, the combined CFD and FEA method used at Behr for the temperature cycle simulation of radiators, charge air coolers and exhaust gas coolers is summarized. The basics of the method will be described in the first section for the example of a full aluminium radiator, showing the typical mechanism of a radiator tube failure. Afterwards, the details of temperature cycling method for charge air coolers and exhaust gas coolers will be explained.

## SIMULATION METHOD: RADIATORS

The first step in temperature cycle simulations is a transient CFD analysis in order to gain the

dependent temperature distribution in the component solid structure. This method is shown exemplarily for radiator simulations in Fig. 1. The CFD analysis is based on a full representation of the coolant and cooling air stream, the convective and conductive heat transfer in the fluid streams, the conjugate heat transfer between the fluids and the structure as well as the heat conduction in the solids. The solid temperatures from the CFD mesh are mapped to the FEA solid mesh at predefined time steps. Linear static analysis is performed for each of these time steps, yielding the strains and stresses based on the mapped solid temperatures. Efficient post processing algorithms have been developed to enable an in-depth analysis of the results: time plots of strains and stresses in particular points, overall deformation plots etc. The CFD method has been implemented using the commercial Finite Volume Code StarCD. The FEA analysis is performed within the solver PERMAS.

Suitable S-N curves and fatigue software tools may subsequently be used to assess component lifetime, based on the stress and strain results from the structural analysis.

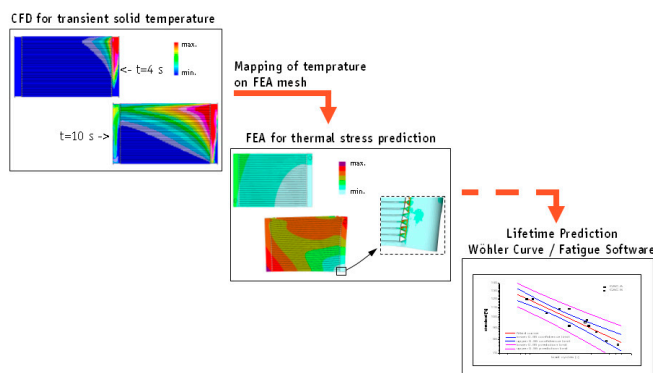


**Fig. 1: Method for temperature cycling simulation**

The geometry of a typical full aluminium radiator and some mesh details of the CFD model are shown in Fig. 2 and Fig. 3 respectively.

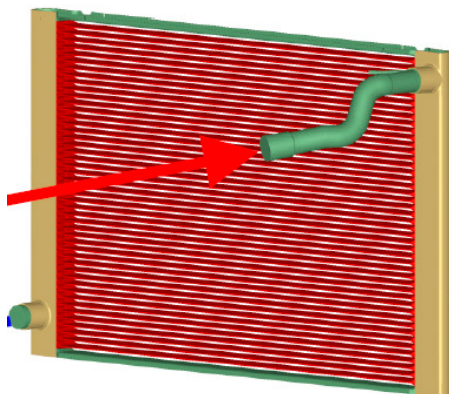


**Fig. 2: Simulation model of a full aluminium radiator**

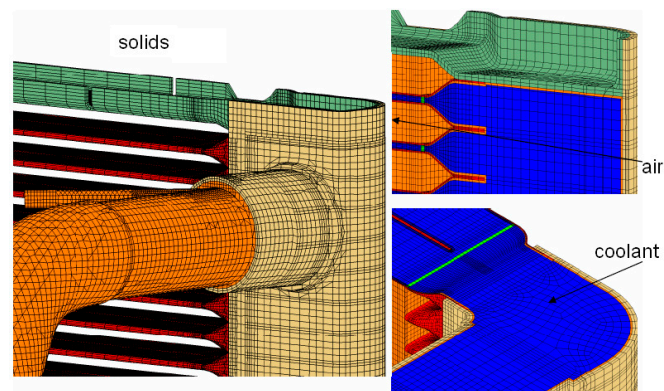


**Fig. 3: Mesh details of a full aluminium radiator CFD model**

The CFD mesh is predominantly built with hexahedral cells to ensure good convergence behaviour. The cooling air fins of the radiator block are modeled as homogenized material, while all other solid parts are modeled in full detail. The coolant flow has also to be resolved in detail in order to represent the strong influence of temperature dependent material properties, this is especially true for the viscosity of the coolant mixture.

The air fins are represented in a homogenized form because it is not possible to model the detailed fin structure for the full heat exchanger block. This is shown in Fig. 4: Modeling the finned region between the coolant tubes with overlapping cells, representing the cooling air as a porous medium and the fin solids as a homogenized material.

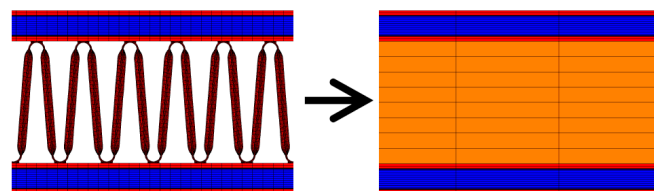


**Fig. 4: Homogenisation of louvered fins in the CFD model**

The porous medium for the cooling air accounts for the pressure drop effects. User defined subroutines describe the heat exchange between homogenized cooling air and fin solid with a so called dual cell method. The dual cell method enables the introduction of local sink and source terms depending on the local Reynolds number. The anisotropy of the fin conductivity due to their geometric structure is also accounted for. This feature is particularly important to represent the heat transfer perpendicular to the tube length axis. The porous media coefficients and the heat transfer characteristics of the homogenized fins have been determined by measurements. They are provided as dimensionless PSI- and Nusselt-Functions through a database used in a 1D simulation tool. Up front analyses have shown that a representation of radiation effects is not necessary, the main heat transfer effects are defined by conduction and convection.