Methodology Transfer of Transient Charge Air Cooler Thermal Cycle Analysis to FLUENT

Dr. C. Merkle, S. Larpent, Dr. W. Kühnel, Dr. T. Heckenberger, F. Guilbaud
Behr GmbH & Co. KG, Stuttgart, Germany
Dr. Y. Chen
Behr America Inc., Troy, USA

Abstract

A completely closed chain of simulation models for the transient aero-thermodynamic and structural analysis of charge air coolers was developed. Computational fluid dynamics (CFD) delivers full information on the spatially non-homogeneous and time-dependent flow and on the temperature distribution in the fluids and structural parts. The structural temperatures are transferred to a finite element analysis (FEA) model and used to simulate the strains and stresses at all points for complete transient cycles. This allows the determination of the critical points in the design and the critical operation modes. Besides delivering key analysis results, CFD and FEA help in gaining a thorough understanding of the physics behind the behavior of component designs. As different CFD codes are used at Behr due to customer requirements the method was first developed in another commercial CFD code and then transferred to Fluent.

1. INTRODUCTION

The engine performance can be increased by improving the combustion with high air flow rates into the cylinder. This is achieved by compressed air with a turbocharger. Beside the compression it is beneficial to reduce the temperature of the charge air for further improvement of the combustion process. Nowadays almost all turbocharged engines are combined with a charge air cooler. The charge air can be cooled by air or by coolant. For direct cooling with air the charge air cooler is part of the cooling module in front of the vehicle. Indirect charge air coolers are integrated in the coolant system and are placed inside the engine compartment.

Since driving a vehicle is a highly dynamic process, all parts of a cooling system face unsteady variations of internal pressure and temperatures. The design of the heat exchanger has to consider all of these transient loads so that the cooler will withstand all loads over the required lifetime with high reliability.

Copyright ANSYS, Inc.
At Behr simulation is a fixed part for validation in the development process. In early phases of development, simulation is the only possibility to evaluate concepts and designs. Therefore validated simulation methods are required. For predicting the thermal stress in heat exchangers, methods have been developed for various coolers. Basically the process of the simulation method for the various heat exchangers is the same: with computational fluid dynamics (CFD) the non-homogeneous and time-dependent flow and temperature distribution is predicted. The structural temperatures resulting from the CFD are transferred to a finite element analysis (FEA) model and used to simulate the strains and stresses at all points for complete transient cycles. With this, critical points in the structure and critical operation modes can be determined and measures for improvement derived. Simulation helps to get a deep understanding of the physical phenomena of the complex transient temperature cycling process.

The results of a coupled CFD and FEA analysis for a radiator thermal cycle test and its validation are described in [1]; the analogue process for an exhaust gas recirculation cooler is published in [2]. The first results of transient CFD simulations for the charge air cooler considered in this paper are reported in [3]. The completely closed chain of simulation models for the transient aero-thermodynamic and structural analysis of charge air coolers is described in [4]. This paper focuses on the transfer of the CFD methodology for charge air coolers to Fluent.

2. CFD SIMULATION METHODOLOGY

The method is based on the coupling of CFD and FEA used for charge air coolers for the prediction of a thermal shock load in a transient operation mode. Accuracy and efficiency were two basic prerequisites for the integration of the method in the development process of charge air coolers. Primarily, the methodology was implemented using the commercial finite volume code Star-CD v3.26.

To reduce both meshing and calculation time, the heat exchanger fins have been modeled as porous media with a heat sink/source. The pressure drop and heat transfer characteristics of the homogenized fins were determined using 1D-data from the BEHR system database.

Fig. 1 shows some details of the CFD model used for the validation. The mesh of the solid parts was carried over from an existing FEA mesh. Since the development of the method the mesh generation code Harpoon has also been commonly used to mesh the complete geometry (solids and fluids). The temperature distribution on the solid parts is then mapped from the CFD mesh to the FEA mesh for predetermined time steps.

To decrease the calculation time a so-called two-step methodology has been developed, switching sequentially from the fully coupled transport and energy equations to the unique resolution of the energy equation. The method has been validated using four different criteria:

1. Comparisons with infrared (IR)-measurements for an overall qualitative assessment of the temperature distribution
2. Time-dependent charge air outlet temperature as an integral quantitative system answer
3. Time-dependent measurements of local structure temperatures at 38 locations on the cooler (tubes and tanks) for the complete test cycle
4. Time-dependent measurements of strains at 44 locations on the cooler (tubes) for the complete test cycle

Copyright ANSYS, Inc.
Some examples of the validation of the method are shown in Fig. 2. The diagrams show the time-dependent measured temperatures at three locations of the charge air cooler in comparison to simulation results. The relative deviations remain small in all cases and even at the points with the largest absolute deviations the transient behavior is still reproduced very well. However, a global comparison of all locations shows that deviations might be caused by a scatter in the positions of thermocouples and undefined contact between the thermocouples and the tube wall. A further refinement of the model is not judged as sufficiently justified.

The comparison of the measured strains with the predicted strains from FEA similarly shows good agreement. Further details of the validation of the method are described in [4].
3. TRANSFER OF THE METHOD TO FLUENT AND RESULTS VALIDATION

To transfer the methodology from Star-CD v3.26 to Fluent 6.3.26 a User-Defined Function (UDF) has been developed. This UDF serves the following purposes:

- Adjust the heat transfer on the interior surfaces of the heat exchanger tubes. To reduce the size of the computational mesh, the interior of the tubes was modeled as a porous medium. The increased heat exchange due to the presence of interior fins was implemented using a user-defined function.

- Adjust the heat transfer between the outside air and the exterior charge air cooler fins. This part of the UDF consists of reproducing the approach used in Star-CD to accurately compute the heat exchange between the outside cooling air and the outside cooling fins of the charge air cooler.

- Convert the porous medium coefficients used in Star-CD (α and β) into inertial and viscous resistance coefficients (C1 and C2) as requested in Fluent. This conversion was carried out using the Fluent and Star-CD definition of a porous medium [5] [6].

In addition to the UDF a scheme file has been developed to switch back and forth from a fully transient mode to an energy-only transient mode. The validation has been completed on a heavy duty truck charge air cooler used for the development of the methodology in Star-CD. Since the methodology has been fully validated
within Star-CD, Star-CD results have been used as a reference for the validation of the methodology in Fluent.

The Fluent mesh was generated with the commercial mesh generation code Tgrid 4.0.24 (see Fig. 3). It is hexcore dominant similar to the one generated in ProAM for Star-CD. The number of cells as well as the mesh topology is comparable in both cases.

![Cross-section through the computational domain: ProAM mesh (left) used in the Star-CD simulation, Tgrid mesh (right) used in the Fluent simulation](image)

**Fig. 3: Cross-section through the computational domain: ProAM mesh (left) used in the Star-CD simulation, Tgrid mesh (right) used in the Fluent simulation**

The test cycle used during the validation is shown in Fig. 4. The cycle ramps up to full heated charge air side, along with low cooling air flow at ambient temperature. After 90 s, the cooling air flow is increased up to 180 s along with the hot charge air. At 180 s the latter is no longer heated and at 270 s the cooling air flow is again reduced to the lower level. At 360s/0s the cycle starts over.

![Thermal cycle on test bench](image)

**Fig. 4: Thermal cycle on test bench**

The initialization of the transient simulation is done with a steady-state solution. To ensure the convergence of both Fluent and Star-CD steady-state solutions, residuals and charge air outlet temperature were monitored (see Fig. 5).

The average outlet temperatures of the charge air predicted by the steady-state simulation in Star-CD and Fluent are comparable (see Fig. 5). Note the different initialization temperatures and scales. This is a first indication that the physical models have been implemented correctly in Fluent.
Fig. 5: Steady simulation – Convergence monitoring
Residuals: Star-CD (top left), Fluent (top right)
Charge air outlet temperature: Star-CD (bottom left), Fluent (bottom right)

Fig. 6 shows that the mass flow distribution through the tubes is similar for both solvers. The maximum absolute deviation is less than 0.5%.

Fig. 6: Mass flow distribution through the tubes of the charge air cooler; steady-state simulation

The transient inlet temperature and mass flow characteristics of the test bench (see Fig. 4) were used as boundary conditions in the CFD models. A relevant criterion for the FEA analysis is the temperature distribution in the solids at any time of the operation mode.
Fig. 7 shows the comparison of the structural temperature distribution for different time steps of the test cycle as predicted by Star-CD and Fluent. The small deviations observed can be in part explained by the different post-processing tools used to generate the pictures in Star-CD and Fluent. Overall both Fluent and Star-CD temperature distributions are in agreement.

![Fluent results](chart1)

![Star-CD results](chart2)

**Fig. 7: Temperature distribution in the solids at different time steps**

Star-CD (bottom), Fluent (top)

In order to have a quantitative validation criterion of the predicted solid temperatures in Fluent, virtual sensors were placed at different locations on the cooler surface (see Fig. 8). The deviation between the Star-CD and Fluent results is small. These results could be improved by further fine-tuning of the physical model to obtain a perfect match. Overall the Fluent and Star-CD results are in good agreement. Fluent can reproduce the transient behavior of a charge air cooler thermal cycle. This method is now used intensively in the development process at all Behr locations that use Fluent.
Fig. 8: Transient temperatures at four different locations for a complete test cycle

4. CONCLUSION

A method for predicting thermal stress in charge air coolers due to temperature cycling was developed and validated with a commercial CFD code. Because of customer requirements Fluent is used additionally at some Behr sites. Therefore the method was transferred, all necessary UDF were defined resulting finally in good agreement with the validated results from the other CFD code.

5. REFERENCES


