# Multiphysics Simulation Optimizes Design of EGR Coolers

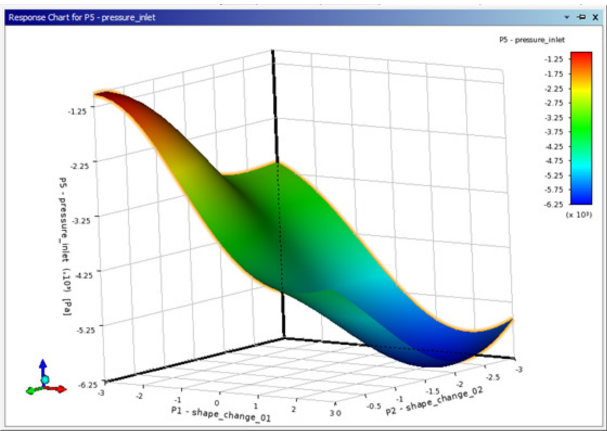
Exhaust gas recovery (EGR) coolers represent a very difficult design challenge, primarily because multiple physics are involved in evaluating their performance under thermo-mechanical cyclic loading. Using conventional simulation tools to simulate this condition requires a time-consuming manual or script-driven process to map data between simulation tools. ANSYS provides a complete suite of multiphysics simulation tools integrated together in the ANSYS Workbench environment that makes it possible to simulate the full range of EGR cooler operating conditions while data are seamlessly moved from one tool to the next. The end result is that EGR cooler designers can optimize the design to a higher level than was possible in the past while getting the design right the first time.

EGR coolers are typically shell and tube heat exchangers that use engine coolant to reduce the temperature of exhaust gas before it is recirculated through the engine's intake system for the primary purpose of reducing NOx emissions. The basic design goal is typically to achieve a given amount of temperature change in the exhaust gas while minimizing the cooler size and pressure drop in order to lower manufacturing and operating costs.

In addition, designers of EGR coolers must address some special challenges caused by the extreme operating environment. As the engine changes back and forth from idle to running speed, it generates cyclic thermo-mechanical loading which can lead to fatigue failure. In order to ensure the EGR cooler can withstand this duty cycle, it is typically subjected to accelerated thermal shock testing.

Overheating of the coolant circuit on the shell side can cause the coolant to boil, which can in turn lead to catastrophic failure of the EGR cooler. On the other hand, very cold conditions can cause the exhaust gases to condense and deposit sulfuric acid on the inner walls of the tubes. This phenomenon, known as fouling, corrodes the tubes, reducing their life and increasing back pressure, and may also drive water into the intake manifold.

Addressing these often conflicting design goals presents a considerable challenge. As with most any heat exchanger, gas should be distributed uniformly in all of the tubes to minimize pressure drop and manufacturing costs. Computational fluid dynamics (CFD) simulation can determine the flow distribution for any proposed design. The design can be optimized by using a shape optimizer to drive the CFD simulation, usually by focusing on the inlet diffusor.

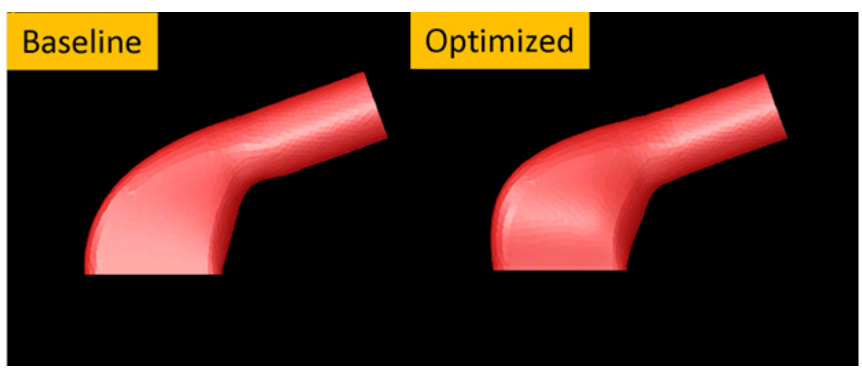



Response surface shows effect of three shape parameters on flow distribution uniformity.

In addition, designers need to ensure the design can withstand extreme operating environments. Typically, CFD is paired with conjugate heat transfer (CHT) to predict the temperature of the metal components. The temperatures are then mapped as boundary conditions to a thermal mechanical simulation tool that calculates thermal stresses. At each time step, the temperatures at each node need to be exported as text files and imported to the thermal-mechanical software. This is not difficult for a single time step but many time steps are typically required to evaluate a duty cycle. The mapping process can take a considerable amount of time for each design iteration and many design iterations often need to be evaluated during the design process. Each mapping operation has the potential for errors that could in the best case require the operation to be done over, or in the worst case produce unreliable results. This process is so difficult that most EGR cooler designs are not simulated for cyclic thermo-mechanical fatigue, so they often fail testing and require expensive additional built-test-fix design loops.

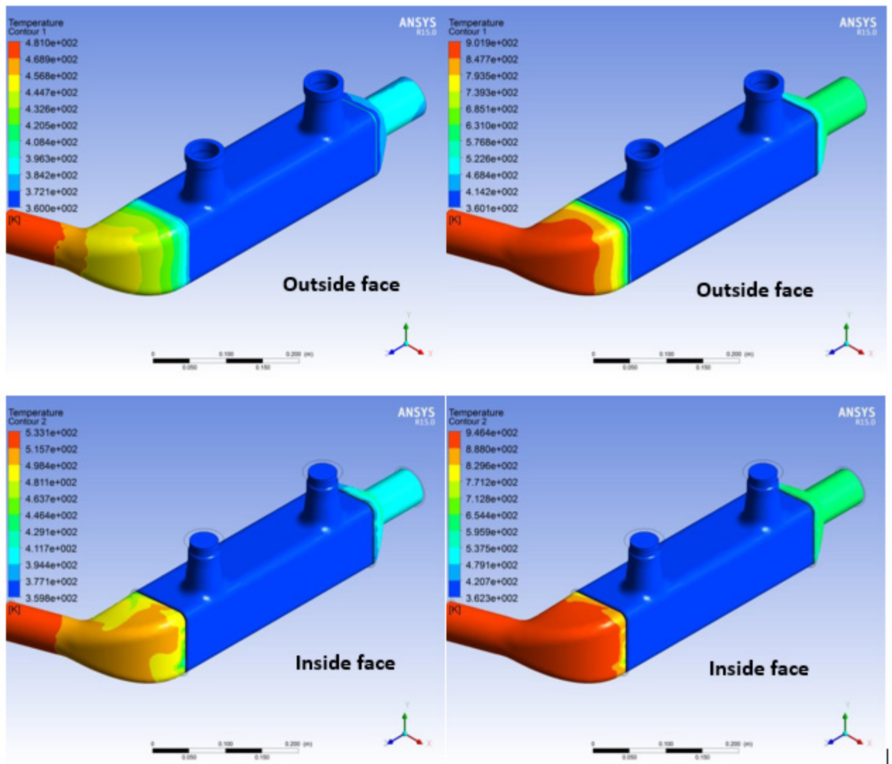
**Solution**

**ANSYS provides a multiphysics solution that integrates all of the different** functionality required to simulate and optimize the design of an EGR cooler within the ANSYS Workbench environment, where the output from one software package can be coupled as input to another with a single drag and drop operation. Further, you can use a range of different parametric analysis and design optimization tools to simultaneously drive any or all of these software packages to optimize the design with all of the physics involved in the product taken into account.



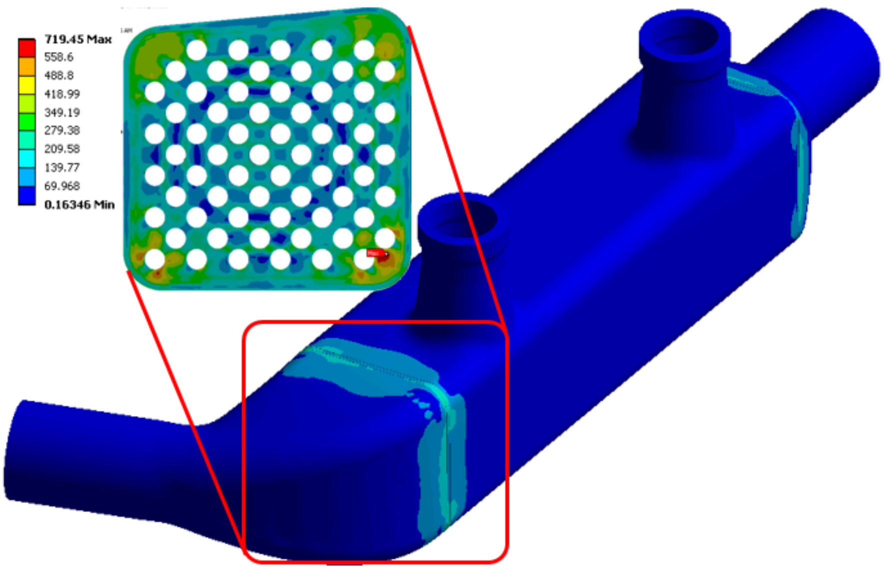

Optimized inlet diffusor geometry provides a 25 percent improvement in uniformity with marginal reduction in pressure drop.

The first step is usually to use ANSYS shape optimization technologies such as the mesh morpher and optimizer and RBF-Morph to drive the CFD simulation and even out flow through the tubes to maximize heat transfer and minimize pressure drop.

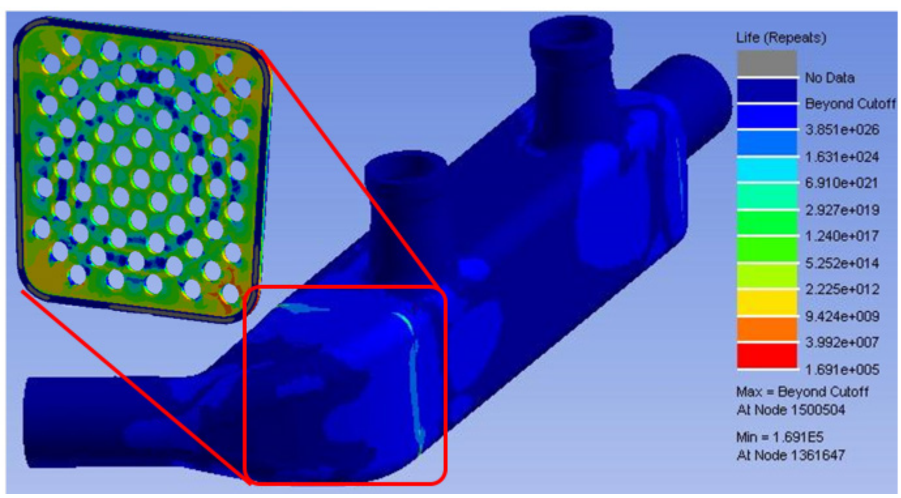



CFD-CHT simulation results show temperature of metal components.

The next step is typically to evaluate the design against the thermal-mechanical fatigue life requirements. Solving transient CFD-CHT simulation for the load cycle could take a long time since the specific heat of the metal components is large. You can achieve substantial time savings by carrying out steady state CHT at hot and cold conditions. Using ANSYS Workbench, you can map the heat transfer coefficients and heat flux calculated from the CFD-CHT simulation as thermal boundary conditions in ANSYS Mechanical simply by dragging the CHT output to the ANSYS Mechanical boundary conditions.



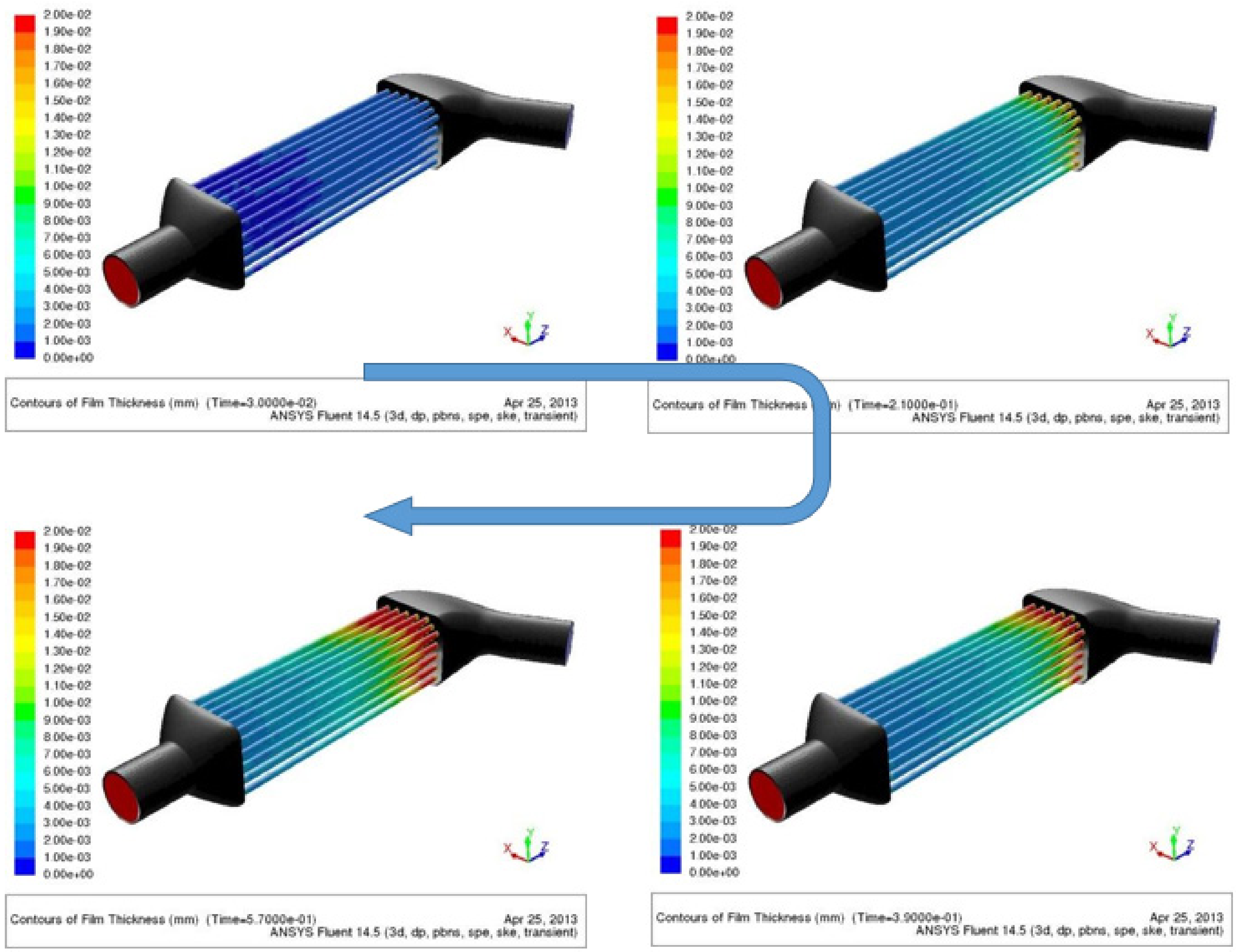

Max Von-Mises Stress Results




Fatigue analysis predicts minimum life of 1.691E5cycles.

The hot and cold boundary conditions are then used to construct a transient load cycle in the ANSYS Mechanical solver. Transient thermal simulation is performed until the thermal cycle stabilizes to a time-periodic cycle. The stable cycle is then used to solve a transient structural simulation that includes all of the other loads and constraints experienced by the EGR cooler. The thermal stress cycle data are then used to perform fatigue analysis to predict the stress life of the cooler. You can then use ANSYS DesignXplorer to optimize the design based on either thermal stress or fatigue life.

Boiling in the cooler can be simulated in Fluent using a semi-mechanistic boiling model based on Chen's correlation that requires relatively little computational effort and can be used with a multiphase mixture model in a steady-state simulation. The model has been thoroughly validated for both horizontal and vertical channels at different operating pressures and wall superheat. It can be used to design the EGR cooler and especially the inlet diffuser to avoid boiling under severe conditions.




Simulation of condensate formation

## Results

The ANSYS multiphysics suite of software enables engineers to accurately simulate EGR cooler design with respect to heat transfer and pressure drop and investigate performance under thermal duty cycles and very hot and very cold weather. This can be accomplished within the ANSYS Workbench environment which seamlessly transfers data from one simulation process to the next and also provides a full range of optimization tools. The end result is that the performance of EGR coolers can be optimized to a higher level than ever before while getting the product to market faster.

ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com