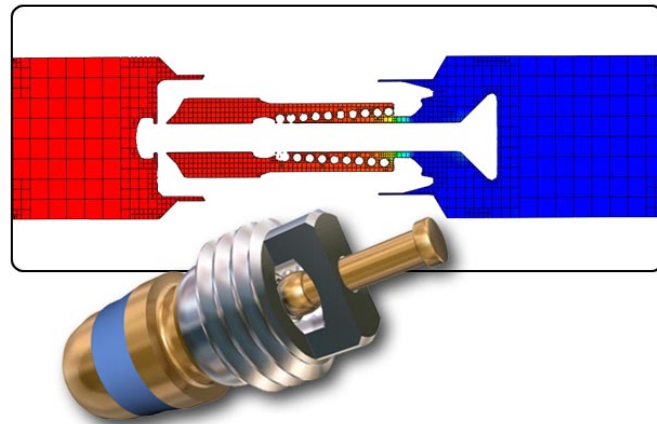


CATIA V5-Embedded CFD Helps Save Four Months in Design of Automotive Valve

The current move from hydrofluorocarbon refrigerants to CO₂ presents a major challenge to designers of valves used in automotive air conditioning systems. The new refrigerant is required to operate at much higher pressures and flow rates which in



turn makes it critical to substantially reduce the pressure drop through the valve. "In the past, we would have had to build and test at least 50 valves in order to get the design right," said Peter Pfaffenwimmer, Project Manager for VENTREX Automotive GmbH, Graz, Austria. Daniel Gaisbacher, also a Project Manager for VENTREX, added that: "CAD-embedded CFD makes it possible to determine simulation results nearly as fast as we can change the design. The result is that we were able to improve the flow rate of our new CO₂ valve by 15% while eliminating about 50 prototypes and reducing time to market by 4 months."

About a decade ago, most automotive conditioning systems converted from CFC-12, a refrigerant that harms the ozone layer, to HFC-134a, a refrigerant that does not harm the ozone layer. Even while this conversion was being made, it was understood that HFC-134a was only a short-term solution because it tends to promote global warming. Now automobile manufacturers are beginning the process of converting their air conditioning systems to CO₂. CO₂ is a refrigerant that exists in nature so it does not damage the ozone layer. Although carbon dioxide is a global warming gas, it has less than 1/1000 the effect of the hydrofluorocarbons it is replacing. But for CO₂ to be effective,

the system needs to operate at pressures 7 to 10 times higher than HFC-134a based systems and this requires redesign of many air conditioning system components.

Need to redesign air conditioning valves

One of the components of the air conditioning system most affected by the change to CO₂ refrigerant is the valves used to evacuate and charge the system. When the system is charged, vapor-phase refrigerant flows through the valve into the air conditioning system. Existing valve designs are not able to operate under the conditions required when CO₂ is used as the refrigerant. Most important, the pressure drops seen in existing valves are too high to enable the required flow rates. Valves need to be redesigned to remove constrictions and improve their efficiency.

Reducing the pressure drop of valves is challenging because of the complexity of internal flow passages caused by the presence of components used to open and close the valve. In the typical valve, the geometrical features in the flow passages create obstructions that increase the pressure drop of the valve. But the complexity of the flow means that it is rarely possible to determine simply by looking at the geometry which features are the most significant from a pressure drop standpoint and what is the best way to correct them. In the past, engineers would make design changes based on educated guesses. For each design change they would need to build and test a prototype of the valve. This process was time-consuming and expensive and the test results did not provide diagnostic information that would help engineers determine whether or not the design change had the intended effect.

CFD provides diagnostic information

CFD provides a potential solution to this problem by simulating the flow within the valve. CFD makes it possible to build a software prototype of the valve that can be solved to determine the pressure drop of any particular design iteration without having to build a prototype. A key advantage of CFD is that the simulation also provides diagnostic information such as the flow velocity and direction at every point in the flow field. This type of information helps engineers evaluate the entire flow field to determine which geometric features are providing the greatest obstacles to flow and offering insights into what type of changes might be able to overcome these obstacles.

Up to now, however, CFD has been primarily used to design high value added products that depend heavily on fluid flow such as airplanes and automobiles but has been used only infrequently by valve manufacturers, which tend to be small and medium sized companies. The greatest obstacle to the use of CFD has been that traditional CFD codes require the user to have a deep understanding of the computational aspects of fluid dynamics in order to be certain of obtaining accurate results. For example, users need to know how to translate their computer-aided design (CAD) model into the CFD environment, then “reverse” the model so that empty flow space rather than the solid product is modeled, create a mesh with the right properties, determine boundary conditions, select the right physical models, tweak solver settings to ensure convergence, as well as other tasks.

CAD/CFD interplay cuts design time

But in the past few years a new generation of CFD software has become available that by extends CAD functionality to include fluid flow and heat transfer simulation within a single user environment. VENTREX has long used CATIA CAD software so they selected Flomerics’ EFD.V5 software, a full-featured general purpose fluid flow and heat transfer analysis software specifically

developed to work inside CATIA V5. "We selected EFD.V5 because it simplifies the process of performing fluid flow analysis to the point where it can be accomplished by any engineer," said Gaisbacher.

Johannes Holzer, also a Project Manager, performed the analysis on the new CO2 valve. He began by calling up the initial concept design in CATIA V5. Then without leaving CATIA he executed a menu pick that invoked the EFD.V5 CFD software to simulate the native CATIA data. The CFD software analyzed the CAD model and identified the voids within the valve where fluids could flow. An automatic mesher created a grid for analysis within these voids. Holzer then attached the boundary conditions directly to the CAD model, in this case an inlet pressure of 59 bar and an outlet pressure of 1 bar which rises to the system pressure of 59 bar.

Iterating to an optimized design

"Within a few hours we had completed the simulation of the initial concept design and we were able to turn our attention to improving it," said Pfaffenwimmer. "We viewed the simulation results within CATIA mapped onto our original model. These results included vector plots that showed the direction and velocity and flow of the refrigerant at every point and color-coded contour maps that showed properties such as velocity and pressure. The simulation of the existing design showed us how the refrigerant was flowing through the valve and helped us identify areas where flow was being constricted."

"Rather than use our intuition to propose design changes," Gaisbacher said, "we zeroed in on the precise areas that the simulation identified as the source of the pressure drop. We started by the relatively simple and inexpensive step of putting chamfers on the corners that were associated with the pressure drops. We made the changes in CATIA and then pushed a button to regenerate the mesh. Then we re-ran the simulation. We could

see not only the effect of our change on the overall pressure drop and flow rate and also get a better understanding of the sensitivity of the design to changes in a particular parameter.”

“The ability to make our design changes in CATIA and then have these changes automatically ripple over to the CFD simulation reduced the time from when we developed the design change to when we got the results to an hour or less,” Pfaffenwimmer said. “In some cases we discovered that smoothing out the corners alone did not have a major impact on the pressure drop. In those cases, we experimented with more extensive remedies such as increasing the size or changing the shape of flow remedies. The key advantage of CFD in these cases was that it provided a window into not only the overall performance of the design but even more important the performance in the local areas that were directly affected by the latest design change. We took full advantage of this benefit by re-simulating and checking the results after each design change to determine whether or not the change had the desired effect.”

“It would not have been practical to use conventional CFD software in this application because by the time we could have gotten simulation results the design would have changed to a degree that would have made them obsolete,” Gaisbacher concluded. “On the other hand, by using CFD software that is embedded into our CAD software we could evaluate the performance of each new design iteration almost as fast as we could conceive it. This made it possible to quickly improve the performance of the design. We reduced pressure drop to the point that flow rate improved by about 15% in the final design at any given pressure. We reduced the number of prototypes that were required during the design process by about 50, which saved a considerable amount of money but most importantly let us bring the product to market faster. We have already shipped this product to customers and they have verified that it performs nearly exactly as predicted by the simulation.”