Simulation Helps Adapt Intake Manifold for Multiple Models, Saving Millions

Ed Hernandez, CFD Supervisor

Ford Motor Company

Dearborn, Michigan

Computer simulation helped to adapt an intake manifold design to meet the varying requirements of several different vehicle models, saving millions of dollars in manufacturing and logistics expenses. Engineers faced the challenge of creating a design compact enough to fit the envelope of the smallest model while being quiet enough to meet the noise/vibration/harshness (NVH) requirements of the most luxurious model in only eight weeks. It would have been impossible to achieve these goals with physical testing alone because it takes a week or two to build and test a single design concept, and testing alone provides little or no information that can be used to improve the next design iteration. Instead, computational fluid dynamics (CFD) simulated airflow through a series of proposed designs while providing complete information on flow velocity and pressure at every point inside the manifold, making it easy to detect problems and correct them. By getting all the major players involved in the design process to review the CFD results as soon as they were available, it was possible to conceive, detail, and analyze each successive design change in only one day, a tenfold improvement over the traditional approach. Engineers were able to complete twelve design

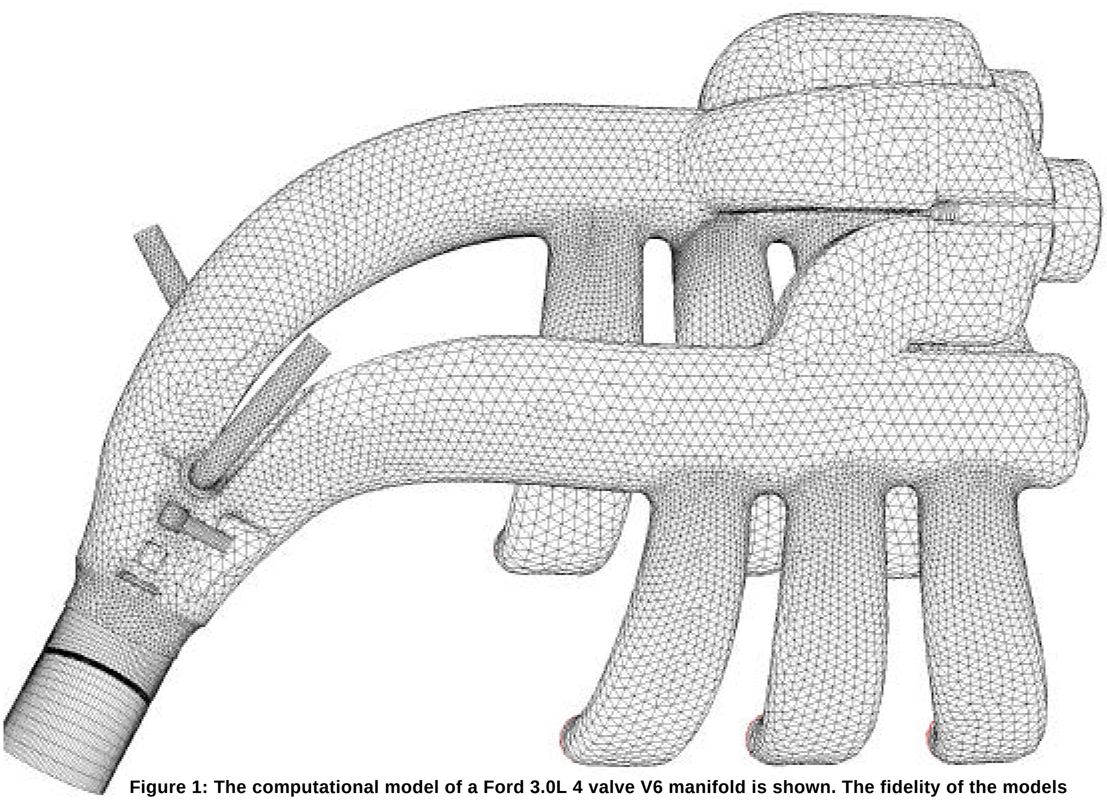

Figure 1: The computational model of a Ford 3.0L 4 valve V6 manifold is shown. The fidelity of the models analyzed by Ford include the throttle body and plate, exhaust gas recirculation(EGR) passage and Positive Crankcase Ventilation(PCV) tube, along with the plenum and intake runners. This is a 4 mode active manifold, so the model includes two balance tubes with tuning valves in their closed positions

iterations during the project life cycle, each providing a major leap in performance over the one before, because it was designed to address a specific flow condition revealed in the analysis. The final design is expected to meet all of the requirements for every model, generating huge savings by eliminating the need for a separate machining line, mold, and inventory.

Every automobile manufacturer is trying as hard as possible to use common components for multiple product lines because of the dramatic savings that can be achieved. Yet compromising on the performance, NVH restrictions, or other requirements of the different lines is never an option. In this case, the decision was made to adapt an existing design to a new model under very tight time constraints. The manifold had to fit within the space constraints of the new vehicle, which had a very aggressive hood line. It was also to have continued use on several luxury vehicles with very strict NVH requirements. One of the most common ways to make a manifold fit into a tighter space is to flatten the design. Flattening the design tends to amplify sounds, however, so a rounder design was needed, even though such a design would naturally tend to take up more space.

The traditional approach

The traditional way to address this problem would be to reduce the design to a size that would meet the requirements, build a prototype, run it on a test bench, find out that it didn't provide the desired flow, and then create a new design to try and fix the problems. This build and test approach would have taken far too long, however, resulting in a failure to meet the project deadline. It usually takes about two weeks to build and test a single design, and the measurements are not very indicative of why the design isn't working, or what might be done to fix it. If one design fails to work properly, the only option is to try to determine the cause of the problem, and then start over with another prototype and see if the new design works any better. Needless to say, many months would be required to meet the project objectives using this approach.

Fortunately, a new design method had been developed and validated at Ford Motor Company that made it possible to optimize a design much more quickly than before. This method uses CFD to model the interior of a manifold and determine the flow and pressure at any point inside the model. While building an initial CFD model can take almost as long as building a physical model, the CFD model can usually be changed in a couple of hours to conform to a new design. Building a new physical model, on the other hand, requires the same investment each time. In addition, each simulation provides far more information than physical testing, making it possible to visualize the flow conditions throughout the manifold and, in most cases, zero in on the issues that are preventing the design from performing in the best possible manner.

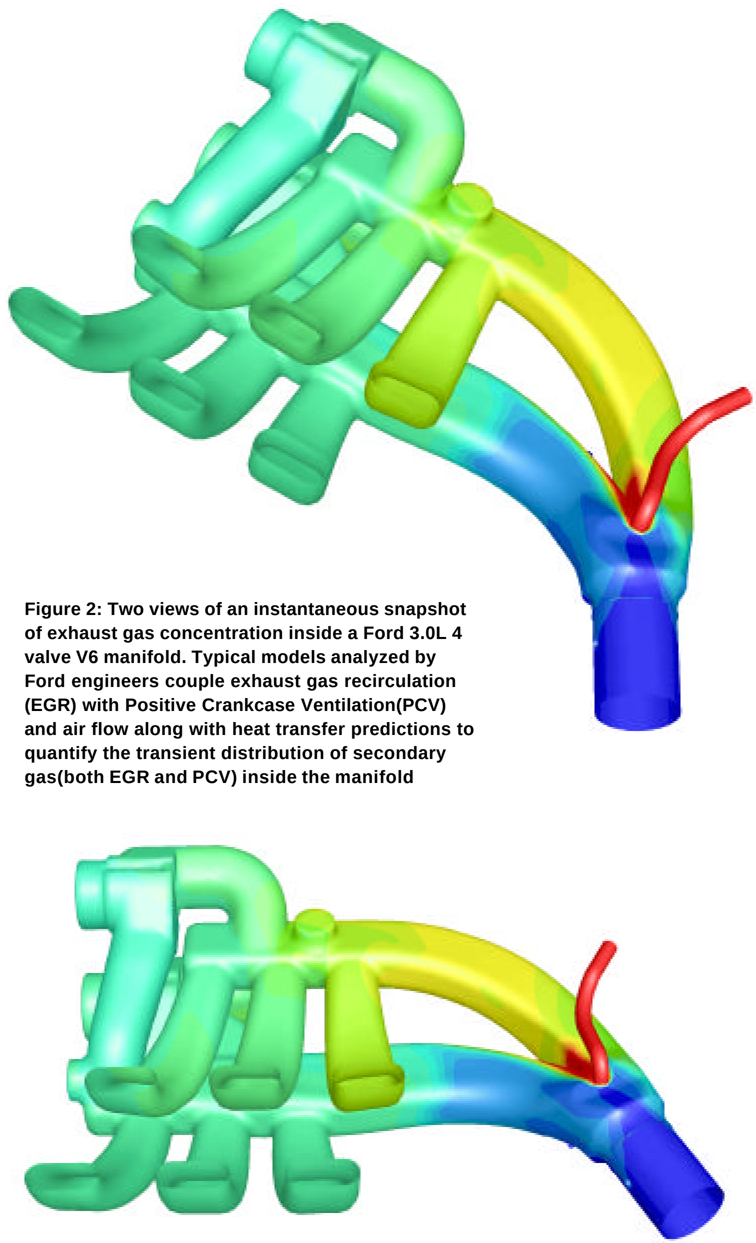

Figure 2: Two views of an instantaneous snapshot of exhaust gas concentration inside a Ford 3.0L 4 valve V6 manifold. Typical models analyzed by Ford engineers couple exhaust gas recirculation (EGR) with Positive Crankcase Ventilation(PCV) and air flow along with heat transfer predictions to quantify the transient distribution of secondary gas(both EGR and PCV) inside the manifold

The simulation approach

The process began by importing the geometry created in the I-DEAS computer aided design (CAD) package into the GAMBIT and TGrid pre-processors from Fluent Incorporated, Lebanon, New Hampshire. GAMBIT was used to generate a surface mesh, then TGrid was used to generate a volume mesh, which consisted of a few prism layers along the boundaries where gradients tend to be high, and tetrahedral elements throughout the rest of the flow domain. The first physical model was completed at about the same time as the CFD model, so the test conditions were duplicated in the simulation in order to validate the model. The CFD solution was performed using FLUENT software on a cluster of dual-processor Hewlett-Packard workstations in about four hours. The initial version of the CFD model matched the test results quite closely, so the results were carefully analyzed to see how the design could be improved.

All of the key players in the project met to discuss the results. They included the design engineer responsible for meeting cost and weight targets, the CAD designer who generates the product geometry, the experimental engineer in charge of physical testing, the NVH engineer responsible for meeting noise targets, and the CFD engineer who performed the analysis. The analysis results clearly showed an area of secondary flow in the plenum of the manifold, where the runners come together. It is normally challenging from a design standpoint because all of the different flow paths come together in a small area, and this has a tendency to create unusual flow patterns that add to pressure drop. In this particular case, a recirculation zone had developed perpendicular to the primary flow path, and was wasting a considerable amount of energy. The results also showed that the pressure drop was significantly higher in one of the runners than it should have been.

Simulation integrated into the design process

With the collection of project engineers in one room viewing the comprehensive information provided by the analysis, fast decisions could be made. A number of different changes to the plenum were discussed that had the potential to solve the problem. Each member of the team weighed in with thoughts on how the different changes would affect their area of specialization. A design change was subsequently mapped out, and the CAD designer was asked to create the modified geometry. It took the CAD designer an hour or two to create the detailed design, which the CFD engineer meshed in about one more hour. The entire process was completed by the end of day, and a solution for the revised model was obtained in an overnight run.

On the next day, the CFD results from the new model and the test bench were compared. They showed a dramatic improvement over the initial design, but they still didn't meet the performance targets. So the process from the previous day was repeated. The CFD results were analyzed, the team of engineers met to discuss different alternatives to improve the flow conditions, the design was modified, and the analysis was run again. At the same time, physical models were built for certain designs, and an acoustic analysis was performed on others. The first five designs created by this process demonstrated such marked improvement that the last was sufficient to meet the design requirements. At this point, only about half of the time allotted for the project had been used, so seven additional design iterations were investigated so that smaller performance improvements and cost reductions could also be implemented.

The final design was as quiet as the original design and provided a slight improvement in flow performance while fitting into the considerably tighter space requirements of the new model. This application highlights the enormous impact that computer simulation can play when it is integrated fully into the design process. It was easy to create and evaluate a new design iteration every day, and each iteration showed considerable improvement over the last because it was developed with detailed knowledge of the previous flow and pressure fields. A critical component of the success was the use of CFD software that is fast, easy to use, and highly reliable. FLUENT has consistently met all of these requirements. The end result was that an existing component could be used on a new vehicle, generating a seven-figure savings for the company.