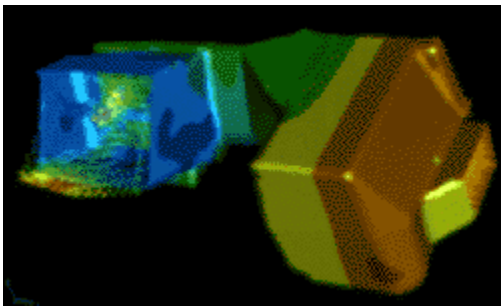


# Unstructured CFD Reduces Design Cycle for Automotive Air Conditioning and Heating Systems at the Climate Control Division of Ford

By George Anderson and Tim Hall, Product Design Engineers  
Ford Motor Company, Dearborn, Michigan



## Group Function

The responsibility of our group is to design air handling systems and components such as evaporator cases, fans, and defrosters for vehicle air conditioning and heater systems. One of the most important considerations in the design of these systems is to minimize the flow losses in the system caused by recirculation, turbulence, or sharp changes in the direction of air flow.

In the past, the only method to evaluate a design concept was to build a prototype and test it in the laboratory. The air handling components were placed on a test stand and conditioned air was supplied at the inlet and the air flow and temperature distribution at critical locations was measured. This approach takes a considerable amount of time and requires the construction of expensive prototypes. But the biggest problem is that even after the test was run, there was

little or no understanding of why the design performed the way it did. In particular, testing is unable to detect details of recirculating areas, turbulence, or constrictions that adversely impact performance and pressure loss.

## New Approach

Recently, we began to model our designs using RAMPANT from Fluent, Inc., Lebanon, New Hampshire. RAMPANT is the first computational fluid dynamics (CFD) software package that we have found to be suitable for use by design engineers, as opposed to analytical engineers who spend all their time doing analysis. As a typical example, RAMPANT reduced weeks of testing of a cowl box by showing why water under certain vehicle operating conditions was being drawn into the air inlet to the system and helped us evaluate alternate solutions to the problem. One of the key reasons that we did not select conventional cartesian or structured CFD codes is the geometric complexity of the components designed by our group. They typically include ducts that expand and contract, change from round to square cross-sections, and go through complex curves throughout their length. Furthermore, in many cases, the performance of the system needs to be evaluated in a variety of different configurations. For example, we need to know the performance of an evaporator case with its internal

doors in every possible position. To correctly determine system losses, accurate geometric representation is required. Minimizing the system losses allows the optimization of air flow in a given package space. This affects the power consumption and noise level of the blower motor as well as the overall performance of the heating and air conditioning systems in the vehicle.

In the past, it took as long to build a CFD model as it took to build a prototype. The modeling tools available were difficult to use by the design engineer within the timing constraints of our normal product cycle which means that the design engineer did not have the chance to use these programs enough to gain the familiarity of a dedicated user. Typically, any analysis had to be sent to our analysis group or contracted out which further increased lead-time requirements. By the time the analysis was complete, many other changes usually had been made to the parts and the results may have been obsolete.

The reason why conventional CFD codes are difficult and time-consuming to use is that nearly all of them use a structured mesh. The problem with these approaches is that each element has to be defined by hand. It usually takes weeks or months to produce a grid using the conventional structured grid approach.

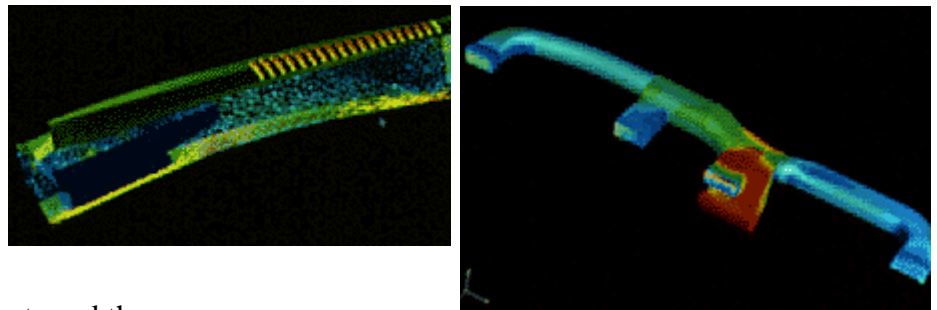
Unstructured grids offer the possibility of adapting the grid distribution in space to optimally solve a given problem. Adaptive mesh refinement also allows important flow features to be resolved locally while retaining a more cost effective coarse mesh in regions of smooth flow.

## Cowl Application

The vehicle cowl box, which is a structural member, sits at the base of the windshield and is used as an air inlet for the air conditioning and heating systems. When water rolls down the windshield some of it goes into the cowl chamber and then is supposed to exit through drains in the opposite ends of the cowl. Early tests for this particular component showed that when the car is sitting at idle or at low speed,

however, the blower would ingest water along with the air up into the blower inlet where it was forced into the air conditioner air handling system. The air flow of this particular component was far too complex to visualize by physical testing.

RAMPANT was used to model the inside of the cowl chamber. Because the part is basically symmetrical it was only necessary to model one half of the chamber. The geometry of the part was created using software from The MacNeal-Schwendler Corporation. We used MSC/ARIES to create a surface mesh of the part using the automatic triangular surface generator included with this system. We



exported this mesh, using the PATRAN neutral file format, into RAMPANT's mesh software, TGRID, to create a volume mesh of the cowl. Furthermore, after the initial analysis had been run, the mesh was refined to increase density in areas that were shown to be important. In this case, we selected several areas where the velocity of the air flow was particularly high. The entire process of volume meshing the component took under one hour.

Boundary conditions were established by measuring the suction created by the blower motor on our prototype and setting zero static pressure at all inlets because water ingest was worst at idle conditions. The model was run overnight on a Hewlett Packard HP735 workstation. The analysis showed that velocity was high in several areas on the outboard cowl and that these velocities were drawing water droplets up into the inlet. The results of the analysis made it easy to visualize what the problem was. Based on these results, the design engineers came up with the idea of adding a hole to reduce the velocity. In addition, a cover was added to the top of the rain hat which will deflect water from the air inlet. Changing the model and re-running the analysis showed that the hole and cover reduced the velocity

and helped to resolve the water ingestion problem.

This analysis demonstrated the viability of using RAMPANT to analyze vehicle air conditioning and heating systems and system components. We plan to all systems that represent a design challenge in a similar manner.

## Concluding Remarks

In the past, problems with a design became apparent only late in the design cycle after full vehicle prototypes had been built and tested. This frequently caused delays and high-cost crash efforts to resolve the problems. Working with a technician in the lab, it frequently took months to come up with a solution. Using RAMPANT, we are able to identify and solve many of these problems long before the first prototype is built. This reduces product development lead-time and engineering costs. An important point to note is that the design engineers themselves perform all of the analysis work with RAMPANT. The engineer can quickly modify the design based on the results of RAMPANT and his or her understanding of the design goals. This is a tremendous advantage because it eliminates the communications difficulties and delays caused by having to work with a separate analysis group.

Another advantage of the new approach is that it provides objective information on the effect of design changes. In the past, at times, all we had to go on was past experience without test data and the vehicle design group was reluctant to make changes without really knowing whether or not these changes would improve the design. Now we can show them what the problem is and how we intend to solve it. While we do not expect this tool to eliminate testing, it has already shown the ability to substantially reduce the

amount of testing that is required using it mainly as a verification tool. The eventual goal of this process is to eliminate one or more prototype iterations and reduce design changes. Currently, there are normally a number of prototype iterations in a vehicle program. Each prototype iteration of a heating and cooling system is very expensive and time consuming. This includes creating tooling and producing plastic prototypes. Eliminating just one prototype iteration will dramatically demonstrate the benefits of CFD in the design process.

