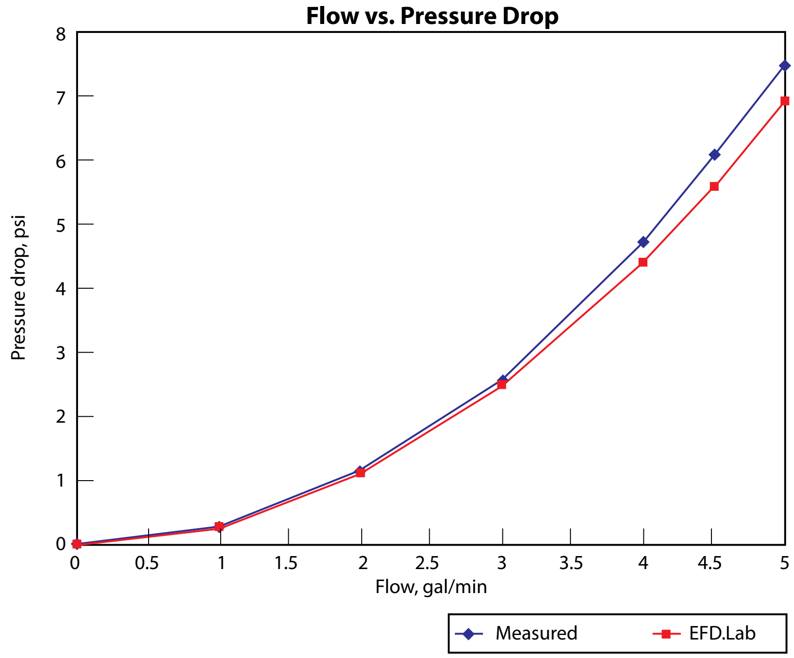
Computers in engineering

Fluid simulation reduces valve prototype development time

Aerospace valve manufacturer **Shaw Aero Devices** recently faced a challenge from one of its customers to build a solenoid valve with a pressure drop of 0.75 psi at a flow rate of 4.45 gal/min, while its standard valve measured out at 6.09 psi.

The valve that Shaw was asked to design was an airborne fuel control solenoid type for a UAV. Six of these valves are used in a manifold that routes fuels



The simulation results of the original valve were compared with test results in EFT.Lab and the results are shown above.

Shaw Aero Devicesí standard solenoid valve has a pressure drop of 6.09 psi; however, the company was challenged to design a valve with a pressure drop of 0.75 psi at a flow rate of 4.45 gal per min.

to different tanks in order to keep the aircraft in balance as the fuel is depleted. The valves are also used during refueling to direct incoming fuel to the appropriate tank. The OEM of the UAV wanted to increase the payload of the craft by using a smaller fuel pump in the aircraft, and this design decision necessitated a dramatic reduction in the pressure drop of the solenoid valves.

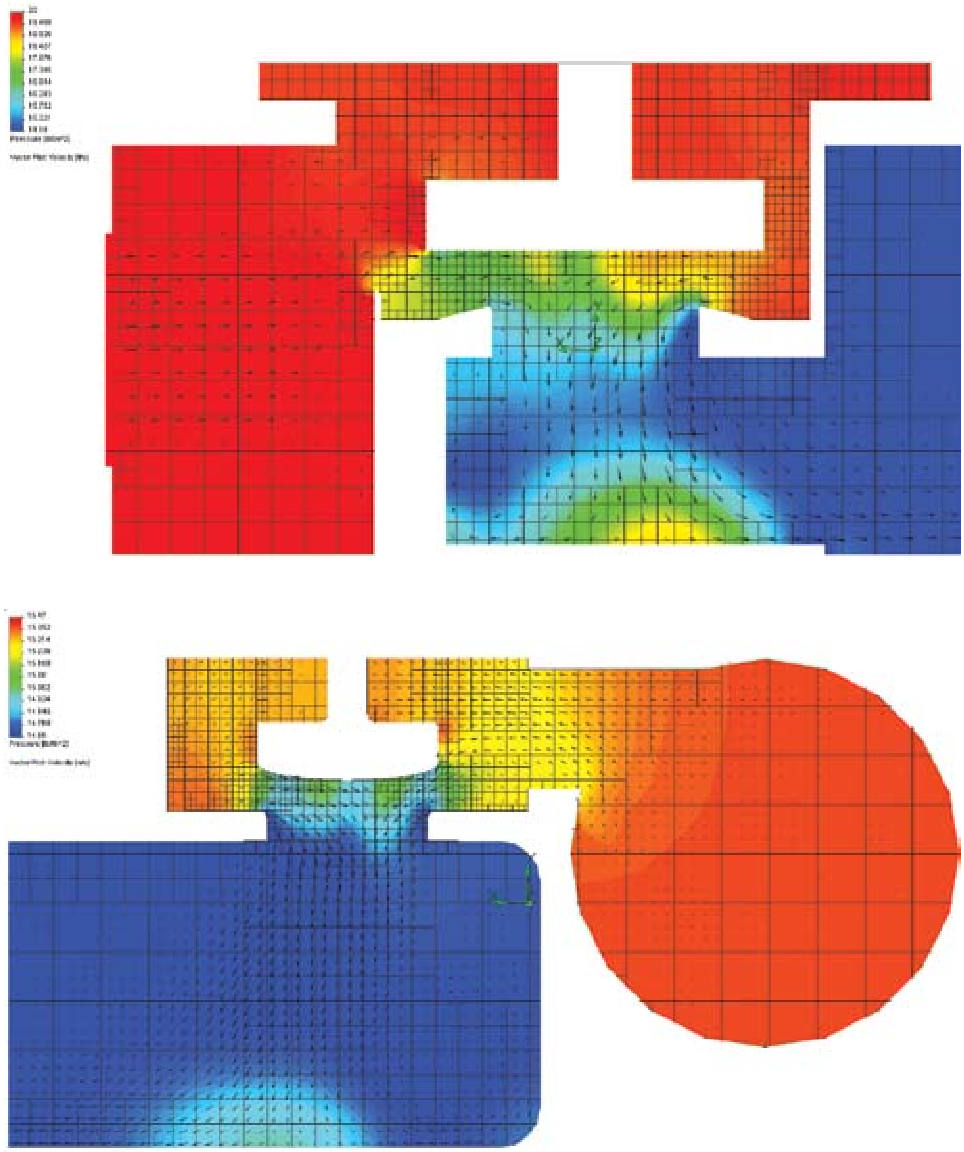
In the past, Shaw would have had to make an educated guess on what was raising the pressure drop in the valve. Each time a guess was made, a prototype would be built to see if the hypothesis was correct. Each prototype would have cost about $3000 and taken about a month to machine, assemble, and test. One of the weaknesses of the build-and-test approach is that physical tests determine the pressure drop of a design iteration but provide very little additional information that provides assistance in diagnosing a problem, such as determining which areas of the valve are constricting fluid flow.

Shaw has for some time been using CFD to address problems of this type as software prototypes without the need for building hardware or running wind-tunnel tests. Until very recently, CFD has been used very little by small and mid-size manufacturers such as Shaw because CFD codes have required the skills of a specialist and could not be performed as part of the design process. The user has been required to know how to translate his or her 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.

But in the last few years, a new generation of CFD software has been introduced that addresses all the major reasons for the relative lack of use of CFD software. The use of native 3-D CAD data, automatic gridding of the flow space, and managing the flow parameters as object-based features eliminates the need for engineers to understand the computational part of CFD and instead enables them to focus on the fluid dynamics of the product, which is already their responsibility to understand and master. The development of user-friendly intuitive CFD codes such as EFD.Lab from **Flomerics** makes it practical for companies of any size to do CFD as part of the design process.

Shaw engineers began by modeling the original standard product design. The design geometry was transferred from the AutoCAD software to EFD.Lab. EFD.Lab reduced the time required to simulate flow by analyzing the model and automatically identifying fluid and solid regions without user interaction. Then the automatic mesher created a mesh with 415,072 cells while maintaining a high level of quality, eliminating the need for any manual intervention. A series of simulations were then run at

Computers in engineering



When compared with the original valve (top), the cut plot of pressure gradient for the final configuration (bottom) showed that pressure gradients were substantially reduced in the transitional area.

flow rates from 1 to 5 gal/min and EFD.Lab calculated the pressure drop of each flow rate. The simulation predicted the pressure drop with an error of less than 10% at every point, not far from the measurement error involved in physical testing.

The CFD simulation also provided valuable diagnostic information. The most useful information was a cut plot of the pressure gradient across the old valve. This plot showed that the majority of the pressure drop was across the gap between the solenoid poppet and the valve seat. The new solenoid needed to have a significantly lower pressure drop than the original design, 0.75 psi vs. 6.09 psi. So the first design change was to increase the diameter of the outlet hole. This modification was made to the CFD model and the simulation was run again. The results showed that the pressure drop was reduced to 2.62 psi. In the cut plots of the pressure gradient, the fuel flows from the high-pressure opening on the left to the low pressure seen at the outlet of the valve exit on the right.

In viewing the cut plot of the pressure gradient for the first design run, it was noticed that the angled wing seat was causing a significant portion of the pressure drop. In order to address this issue, the new valve was modeled with a larger opening below the solenoid poppet and removed the angled valve seat. The CFD simulation showed that this change substantially reduced the pressure drop to 1.09 psi, which was, however, still too high.

The cut plot of pressure gradient for the second design run showed that the transitional area between the poppet and the seat had a high-pressure drop. The valve seat corner also contributed significantly to the pressure drop. The gap between the poppet and the seat could not be increased due to other considerations, so it was decided to try a plain radius for the valve seat and to radius the end of the poppet. When the new configuration was run, the pressure drop across the valve was down to an acceptable 0.71 psi. The slight changes made on this iteration reduced the pressure drop by 35%. The cut plot of pressure gradient for the final configuration showed that pressure gradients were substantially reduced in the transitional area.

CFD simulation dramatically reduced the time needed to meet the customer's specifications, moving from the beginning of the project to the development of an acceptable software prototype in only one day. Without CFD, this knowledge would have been hard won. A minimum of three prototypes, each costing $3000, would have had to have been built, more likely several more.

Simulation results were taken to the customer, and Shaw was awarded the contract to build the valve. Shaw is so confident that the new configuration will work it is not planning to build any prototypes.

Rob Preble, Project Engineer, Shaw Aero Devices, wrote this article for *Aerospace Engineering*.