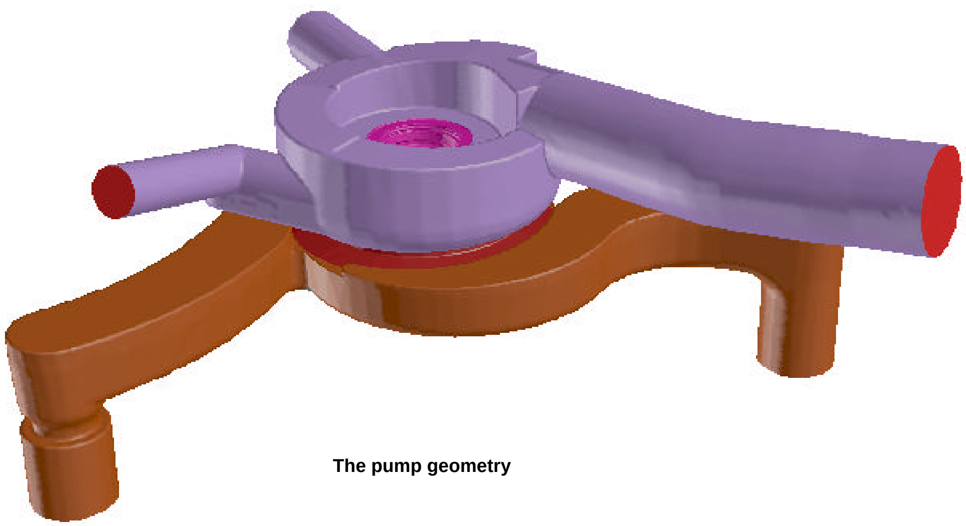
Simulation Helps Tesma Reduce Water Pump Manufacturing Costs

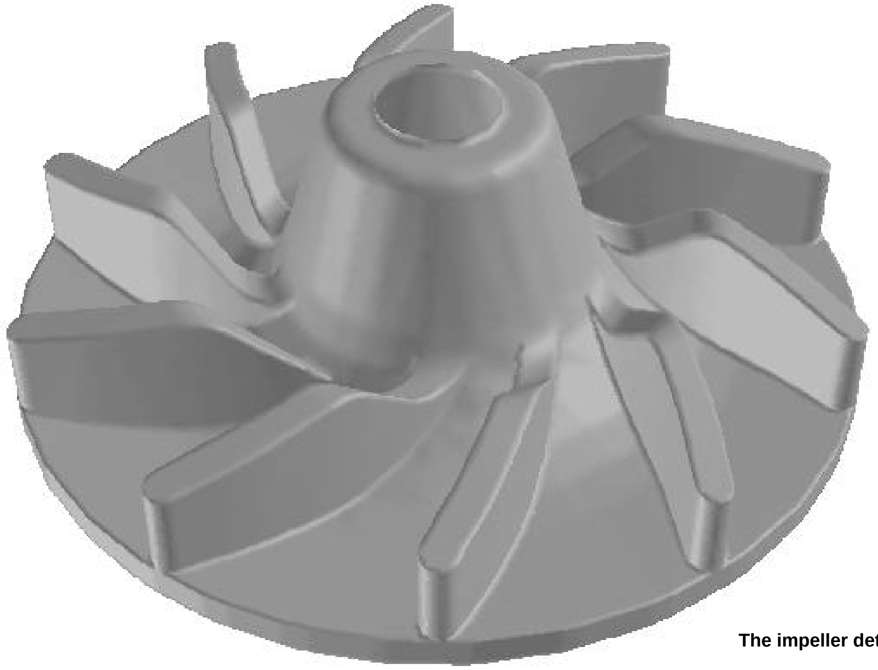
Wei Li
CFD Specialist
Tesma International Inc.
Concord, Ontario

Computer simulation made it possible for Tesma International Inc.'s Engine Technology Group (Tesma Engine Technologies, A Division of Tesma International Inc.) to design a new automotive water pump that met all design requirements while being considerably less expensive to build than the initial concept design. The relatively rough guidelines provided by the one-dimensional analysis design tool used in the past meant that a lengthy build and test cycle was usually required simply to meet performance requirements, making it impossible in most cases to investigate enough alternatives to optimize the design. For our latest program, we used computational fluid dynamics (CFD) to evaluate many different configurations of the impeller and shroud (housing and volute) in the pump, in order to investigate whether less expensive alternatives could meet critical performance metrics, such as pump head and hydraulic efficiency. We ended up going from blank paper to a finalized design in about four months while one engineer was spending only about half of his time on the project. The final result was that we were able to switch the impeller from plastic to a less expensive stamped metal design without sacrificing performance.



The pump geometry

Through working relationships with the six leading OEMs and other corporations in its sector, Tesma plays a global role in automobile production. Tesma employs over 4,600 employees in 22 manufacturing facilities and 2 research centers. Tesma Engine Technologies has operations located in Canada, Germany, and Korea. Their product offerings include front end accessory drive systems, front cover modules, engine oil pumps, cooling management systems, overrunning alternator decouplers, cam covers, variable camshaft phasing systems, engine oil pan assemblies, accessory and timing drive tensioners, pulleys, engine balance shaft assemblies, tubular drive shaft assemblies, and idler assemblies. As North America's leading water pump manufacturer, Tesma supplies over three million units annually. Its water pump and oil pump capabilities stretch to the Far East through a division in Korea.



The impeller detail

Need to continually generate new designs

Tesma must continually produce new water pump designs that are optimized to meet the performance requirements of its customers' new engines. We previously relied on a one-dimensional analysis program developed in-house to provide rough design guidelines. The program accepts as input basic design parameters such as the impeller outside and inside diameters and exit angle, the number of blades, the pump flow rate, and rpm. The program then estimates the head and efficiency of the design. The problem with this software is that it doesn't take the detailed geometry of the pump into account, so it provides only rough performance estimates. Another disadvantage is that this method is not capable of providing the detailed information on flow within the pump that would be required to gain an understanding of why one design works better than another.

As a result, in the past we had to rely largely on the testing of prototypes for detailed performance information. Prototypes were expensive because they required time not only on the part of the design engineer, but also from detailed designers, prototype builders, and test technicians. It normally took three to four weeks to build and test a single prototype. And while this approach offers accurate performance measurements, it provides little or no information on why a particular design performs the way it does, because it's impossible in most cases to obtain more than the most basic information concerning the critical flow parameters inside the pump.

Advantages of software prototyping

CFD provides fundamental improvements by simulating the performance of the pump while taking the full geometry into account and providing detailed diagnostic information. Our goal in moving to CFD was to reduce our design leadtime by performing design cycles in software rather than hardware. We wanted to improve product performance by evaluating more design alternatives while at the same time responding to customer requests to get our products into production in less time. A CFD analysis provides fluid velocity, temperature, chemical concentrations, and other relevant variables throughout the solution domain for problems with complex geometries and boundary conditions. As part of the analysis, a researcher may change the geometry of the system or the boundary conditions and view the effect on fluid flow patterns or concentration distributions. CFD also can provide detailed parametric studies that can significantly reduce the amount of experimentation necessary to develop a device and thus reduce design cycle times and costs.

We selected FLUENT CFD software from Fluent Inc., Lebanon, New Hampshire for this analysis because it simplifies the process of creating the model required to analyze a concept design. Boundary conditions, material properties, turbulence models and particle tracking are specified through the use of tables and drop-down menus rather than by entering information on a command line. In most cases, a user can be initiating an analysis in as little as 30 minutes after importing the analysis model or mesh. Another advantage of this software package is that it provides powerful post-processing capabilities that can easily be used to create graphical output that often clarifies the design problem and helps engineers move quickly to a solution. FLUENT is also capable of handling a wide range of applications in many industries, including aerospace, chemical, power generation, and electronics cooling, for example.

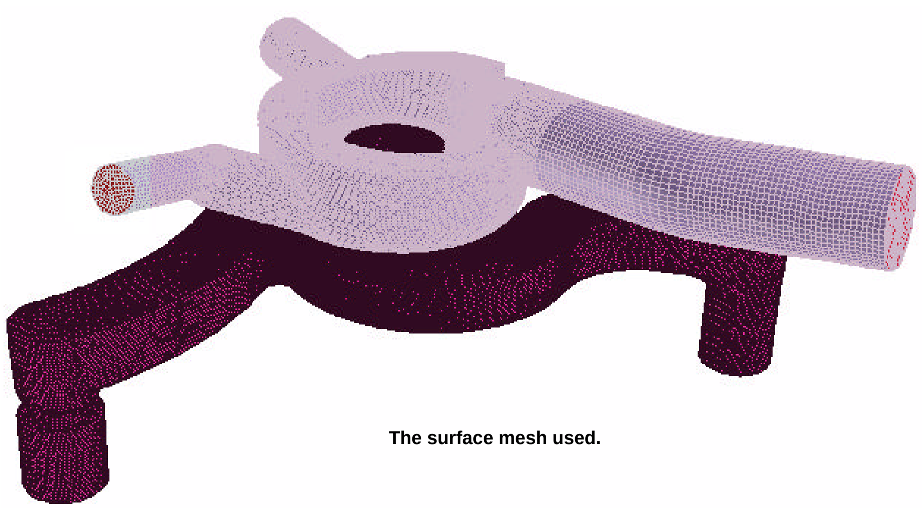
Modeling a recent concept design

On the recent project described above, we began by exporting the initial design concept from the SRDC I-DEAS computer aided design (CAD) system in the proprietary FTL format. We also regularly import geometry from the Unigraphics, PRO/Engineer, AutoCAD, and CATIA CAD systems. In this case, imperfections in the CAD geometry made it necessary to perform some touch-up work. Some edges of surfaces were not connected and other edges overlapped. This does not affect the appearance of the CAD file but must be corrected in order to provide accurate fluid flow results. We then used the GAMBIT preprocessor from FLUENT to generate a surface mesh over the problem domain while controlling the type and density of elements to provide the best mix of performance and accuracy. Tetrahedral elements, selected for fast mesh setup, were used for the majority of the domain while hexahedral elements were chosen for critical areas such as the gap between the teeth of the impeller and the shroud. We then exported the mesh to another Fluent program called TGrid, where we checked the quality of the surface mesh and made corrections where necessary. We then generated the volume mesh and checked its quality.

In order to account for the rotation of the impeller, we used the multiple reference frames (MRF) model in which the impeller is at rest in a rotating frame, and the shroud is in a stationary frame. The solution proceeds with a steady transfer of information across a pre-defined interface between the two frames. We used a mass flow inlet and pressure outlet as the boundary conditions for the model and the k-e turbulence model with standard wall functions. We then solved the model and viewed the results, particularly the pump head and hydraulic efficiency. The results of the analysis showed that the initial design met the performance requirements of the application. Rather than concluding the design process, as would have happened in the past when the cost of building and testing another prototype was high, we decided to investigate whether he could reduce the cost of the design while still maintaining the required performance levels.

Iterating to an optimized solution

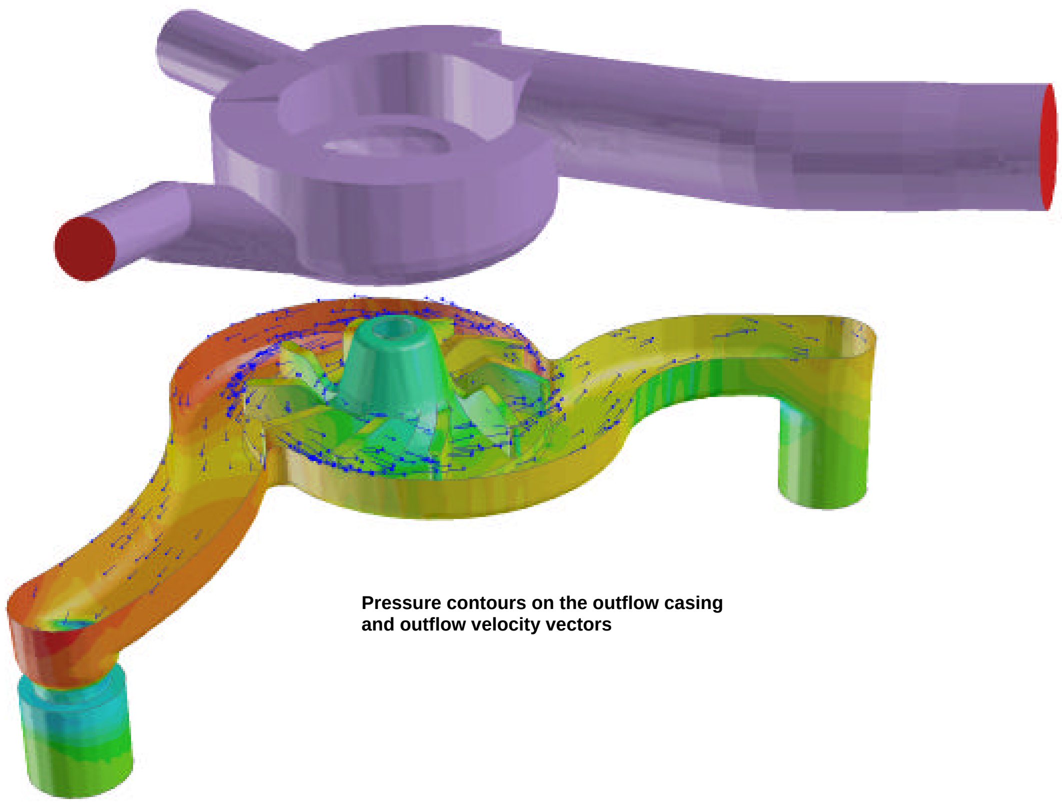
The original design used a plastic impeller because it was thought that the greater geometric flexibility provided by this approach was needed in order to meet the tough performance requirements. In an attempt to reduce the cost while maintaining the same performance levels, we evaluated a number of simpler shapes that could be produced through less expensive metal stamping methods. Ten different designs were investigated in only four months during a period when the engineer performing the analysis was able to spend only about half of his time on the project. We finally settled on a design that met all of the customer's performance requirements while being



The surface mesh used.

considerably less expensive to manufacture than the initial concept design. The design was released for prototype building. Testing results matched the analysis predictions very closely. The difference between analysis and testing is typically between 5% and 10% (usually below 10%) and the difference is stable enough that it is possible to predict the performance of a proposed design with even higher levels of accuracy.

The success of this project is far from an isolated incident. Since Tesma has been using CFD on a regular basis, we have completed four pump designs and in every case we were able to substantially reduce manufacturing costs from the initial design concept. Computer simulation with CFD allows design cycle time to be substantially reduced compared to traditional design methods which require building and testing a minimum of three prototypes - and often require as many as seven - to get the design right. Another advantage is that engineers are available to evaluate more possible design alternatives than was ever practical in the past, often making it possible to reduce manufacturing costs while still meeting performance requirements.



Pressure contours on the outflow casing and outflow velocity vectors