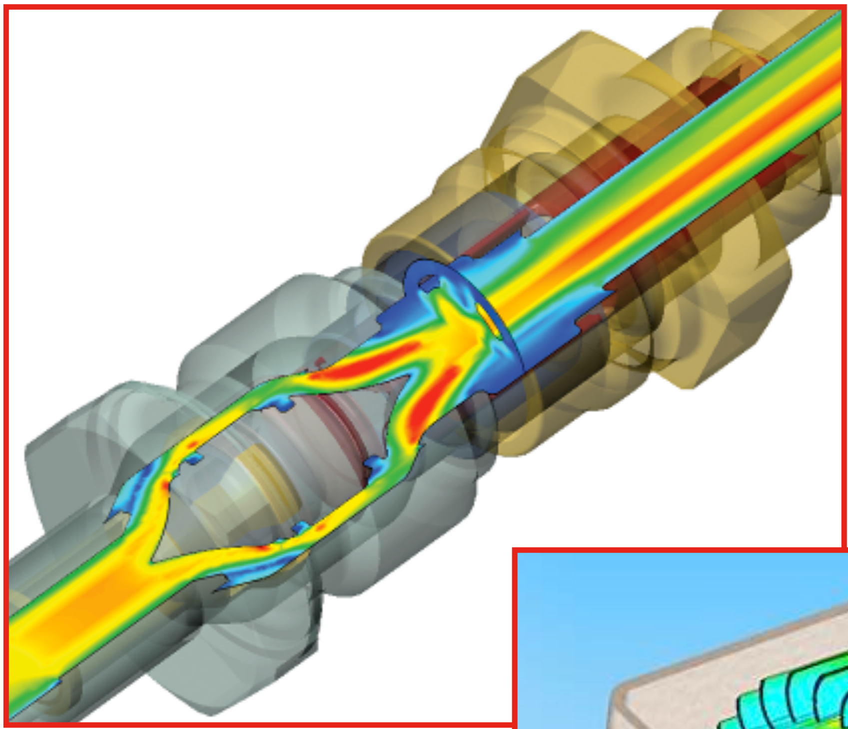
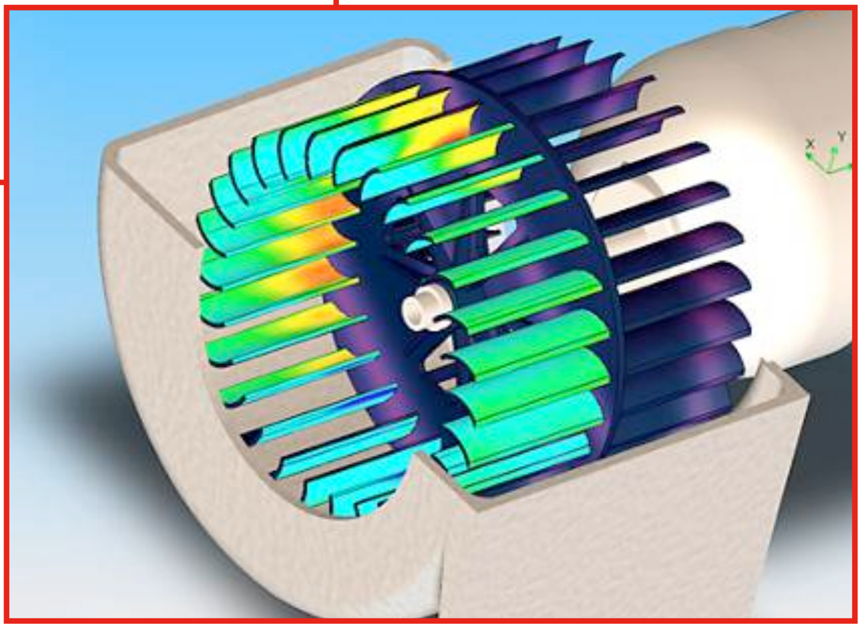
# How Embedded Flow Simulation Accelerates Mechanical Design Process

*SOLIDWORKS Flow Simulation White Paper*





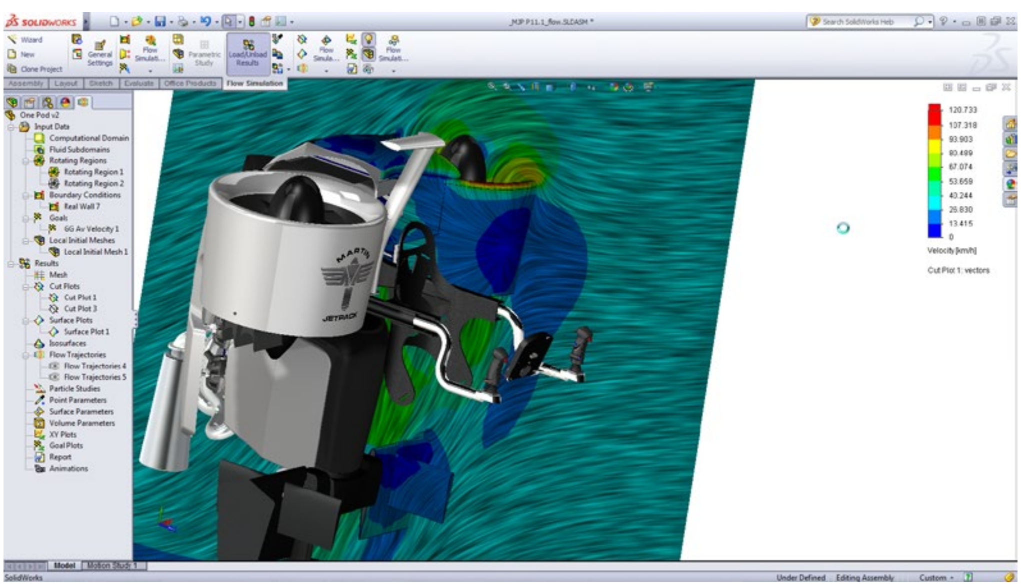
For several decades engineers and mathematicians have been trying to improve the understanding and prediction of fluid dynamics and heat transfer phenomena by means of Computational Fluid Dynamics (CFD) simulation.  The objective is clear - to optimize the design of products and processes that involve fluid flow while minimizing the experimental effort, time and cost expended.

However, it is only in the last ten years or so that CFD has gained widespread acceptance in industrial design, aided by huge reductions in the cost of computing power; better mathematical algorithms and more user-friendly commercial CFD codes.  The use of CFD has also been boosted by the transition from 2D drawing-based design to 3D design using solid models.

The concept of "Product Lifecycle Management" (PLM) has grown out of the desire to accelerate concurrent-engineering design by re-using the same comprehensive 3D data throughout the entire digital design and production process.  An essential step towards achieving this goal is to embed analysis and simulation results – describing operational modes, structural/mechanical behavior, fluid-flow/thermal behavior, etc – into the mainstream mechanical design process.  Today's concurrent-engineering approach often requires predictions about the feasibility and performance of a new design before important decisions are taken that affect design detail.  Simulation plays a crucial role in the design of these kinds of products.

Today's concurrent-engineering approach often requires predictions about the feasibility and performance of a new design before important decisions are taken that affect design detail. Simulation plays a crucial role in the design of these kinds of products.

Mechanical and structural analysis is already quite well embedded in mainstream mechanical design (MCAD) software.  However, the famous Navier-Stokes equations governing fluid flow and heat transfer processes are inherently more complex and non-linear and hence harder to solve mathematically than the equations governing mechanical stress and solid deformation.  Partly for this reason, companies developing CFD software have spent more effort on speeding up their mathematical algorithms than on embedding their software in the mainstream mechanical design environment.  Many commercial CFD companies claim to be integrated with mainstream mechanical design software, but when you look closely the "integration" leaves a lot to be desired and does not satisfy the requirements of fully-embedded concurrent engineering.




CAD embedded CFD Simulation

## A NEW APPROACH...

A new approach to fluid flow simulation is needed, providing cost-effective, process-oriented CFD software that is fully embedded within the mainstream design environment. Simulation results must be available as and when changes occur in the design cycle - only then can simulation play a central role in design decision-making and optimization. So, what are the characteristics and technical requirements for CFD software to fulfill this ambitious goal?

- Firstly, and most decisively, the CFD software must interact directly with the native 3D CAD data defined by the mainstream MCAD software in order to keep pace with ongoing design changes. Very few of today's commercial CFD software tools meet this requirement. Most CFD codes – even those that claim to be integrated with mechanical design (MCAD) software – actually take a copy of the 3D geometry, translate it via a neutral format such as Parasolid or ACIS and add boundary conditions to create a model for fluid flow analysis. Such an approach is fatally flawed and unable to provide fully-embedded flow simulation, because the very act of taking a copy and translating it introduces a "disconnect" between the mainstream mechanical design and the analysis version.
- Secondly, the CFD software must have the same "look and feel" as the mainstream mechanical design (MCAD) software and share the same assembly tree hierarchy, etc, so that the user is not forced to learn a new environment in order to operate the CFD software and is able to focus solely on the physical problem that he is trying to solve. Again, very few of today's commercial CFD software tools meet this requirement. Most vendors have chosen the translation option due to the cost and complexity of developing user-interfaces in several different MCAD software development environments.
- Thirdly, a particular problem for most CFD software tools importing solid models is that the (empty) flow space which must be used for the fluid-flow model does not exist as a discrete "object" in the original MCAD model. The standard method used by most CFD codes to deal with this problem is to identify and extract all the "cavities" from the MCAD model, add them to the feature-tree as extra phantom "objects", and grid them. This approach does not and cannot provide fully-embedded flow simulation because data consistency and 1-to-1 links with the original CAD data are lost.

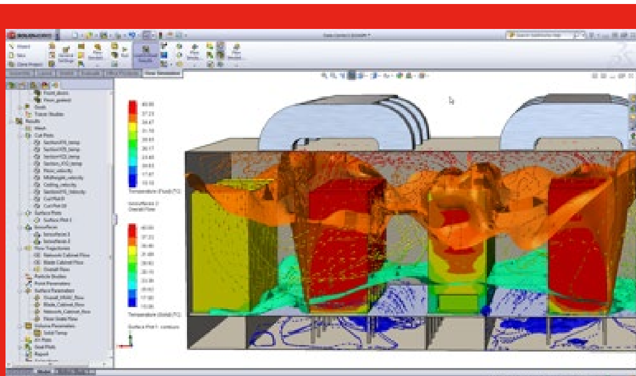
A new breed of CFD software known as Engineering Fluid Dynamics (EFD) has been developed specifically to overcome the problems described here and thus satisfy the requirements for fully-embedded concurrent flow simulation in 3D MCAD design environments.

SOLIDWORKS Flow Simulation product uses native SOLIDWORKS 3D CAD data directly for fluid flow simulations, without the need for any translation or copies.  SOLIDWORKS Flow Simulation software incorporates unique technology known as "Direct-CAD-to-CFD" - a software process that analyzes the CAD model, automatically identifies fluid and solid regions and allows the entire flow space to be defined and meshed in a single step without user interaction and without adding imaginary "objects" to the CAD model.

In most cases, the purpose of fluid flow simulation is a systematic search for the optimal solution to a particular design problem.  To do this, the engineer must simulate a large number of design variants that involve changes to geometrical parameters as well as input variables, temperatures, and flow conditions.  A modern MCAD environment such as SOLIDWORKS 3D CAD is the ideal platform for this, because parts and assemblies are already parameterized and structured in a manner familiar to the mechanical designer, and so design changes can be made very easily.  To make this work in reality, the SOLIDWORKS CAD model store not only geometrical parameters but also fluid flow parameters, temperatures, flow rates and so on.

The purpose of fluid flow simulation is a systematic search for the optimal solution to a particular design problem.

These parameters must be stored as object-based features, managed in the feature tree like other object-based data, and used directly to update the simulation software. SOLIDWORKS Flow Simulation software delivers this functionality and uses specific SOLIDWORKS parameter functions such as design configurations to support links to multiple design variants.  This approach enables a large number of design variations to be simulated automatically and cost effectively, keeping track of which results belong with which model.



**SOLIDWORKS FLOW SIMULATION FULLY EMBEDDED INSIDE SOLIDWORKS AS A CONCURRENT CFD APPLICATION**

SOLIDWORKS Flow Simulation has the same "look and feel" of SOLIDWORKS 3D CAD and shares the same feature tree and geometry model. All design changes are carried out directly in SOLIDWORKS using familiar solid-modelling functions.  All ancillary data required for flow simulations such as material properties and boundary conditions are associatively linked to the solid model and carried along with all design changes. Flow conditions are defined directly on the solid model and organized similarly to other design data in the feature tree.

All CFD simulations require a computational grid to be created in the flow space using some mathematical discretization methods such as the "finite-volume" or "finite-element" methods. The time spent creating a high-quality computational grid for a CFD simulation is often the greatest part of the total simulation time. This "total simulation time" is the critical factor for MCAD-embedded fluid flow simulation because, as we have seen earlier, simulation can only play a central role in design decision-making and optimisation if simulation results keep pace with design changes. So, it is essential for MCAD-embedded flow simulations to have a fast, automatic grid generator that provides high-quality optimized computational grids for flow simulations with no input effort by the user. The SOLIDWORKS Flow Simulation product suite meets this requirement by incorporating a fully-automatic, adaptive grid generator that uses basic functions in the SOLIDWORKS CAD system geometry kernel to optimize the computational grid according to the flow geometry.

Engineers often need to communicate simulation results to non-technical colleagues in order to influence design decisions and provide a good basis for discussion. To facilitate this kind of communication it is important that the flow simulations results are visualized directly on the 3D CAD model within the MCAD environment. Also graphics, charts and tables need to be generated automatically in the relevant Microsoft Office programs. To meet this requirement the SOLIDWORKS Flow Simulation product offer built-in tables, charts and reports that automatically show the simulation results on the surfaces, planes, lines, and edges selected in SOLIDWORKS software.

Today's mainstream mechanical designer is able to perform fluid-flow simulations directly on his own desktop, dramatically enhancing his design capability for new products.

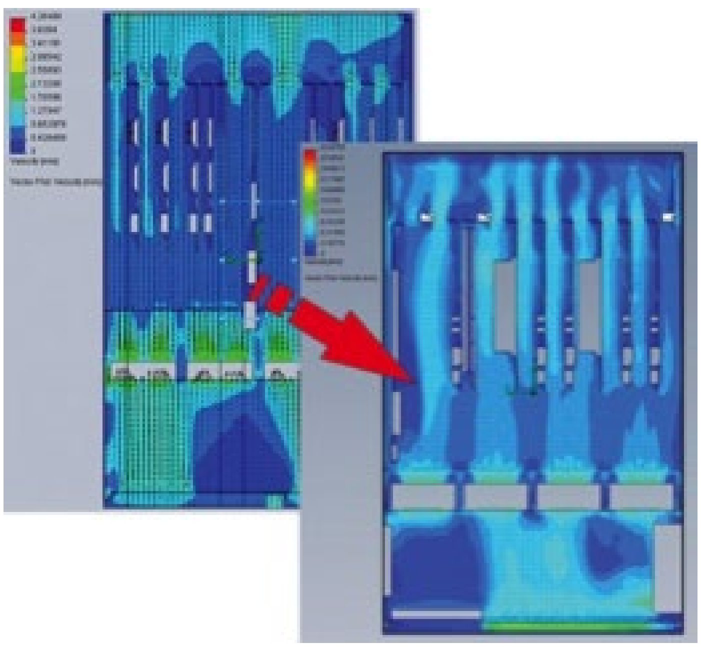
Engineering analysis including fluid flow simulations have traditionally been carried out in specialist analysis departments which are organizationally and sometimes commercially separated from mainstream design and development departments. However, this segregation is rapidly disappearing as PLM strategies are implemented successfully by more and more of today's leading companies. We are now clearly seeing the positive effects of embedding fluid flow simulation within the mainstream MCAD environment. Today's mainstream mechanical designer is able to perform fluid-flow simulations directly on his own desktop, dramatically enhancing his design capability for new products. Simulation results can now be incorporated into the design/development process at the right place and above all at the right time. By the same token, analysis specialists are relieved of the burden of routine design tasks and can focus on more complex projects in pure research and future product development. In this way, the full potential of fluid flow simulation as a product development and decision-support tool can be realized.

## INTEGRATED DESIGN AND ANALYSIS IN ACTION

The examples demonstrate how CFD has become embedded in the design process in many companies, enabling simulation to have a beneficial effect throughout the product design lifecycle. The key to these improvements is the availability of CFD software that provides results directly from the native MCAD data. According to a recent survey, this trend can be expected to continue. In response to "how important MCAD integration is for typical CFD users" 46% said it is "very important today" and an overwhelming 68% said it will be "very important in the future." When the results were confined only to users of mechanical CAD software, these percentages increased to 52% and 74% respectively. When asked what "CFD integrated with MCAD" means, most people chose the option describing the tightest level of integration: "A fully-embedded solution in a single environment. The CFD user interface looks and operates like the MCAD software and the CFD solver interacts directly with the "native" MCAD solid model data." Clearly the trend of using CFD at every stage of development seems likely to increase.

### POLYRACK TECH-GROUP

The German manufacturer is a leading provider of integrated packaging solutions for the electronics industry. Polyrack uses CFD to resolve heat transfer challenges related to its packaging designs.



**"SOLIDWORKS Flow Simulation not only improves our productivity and efficiency, but also lets us tackle heat transfer challenges that we would not be able to resolve without it."**

— Bernd Knab,
Development Manager

In evaluating flow analysis systems, POLYRACK determined that a CAD-integrated package was preferable. "It's better when the simulation takes place inside the CAD system," Bernd Knab, Development Manager, stresses. "It takes too much time when you have to write data to another format, and requires moving back and forth between the applications, and duplicating effort."

Bernd Knab explains: "SOLIDWORKS Flow Simulation has improved our development efforts by allowing us to better understand and address the heat transfer challenges that are inherent to our work."

With SOLIDWORKS Flow Simulation, POLYRACK can quickly simulate heat transfer behavior in packaging designs, 90 percent of which are customized for specific applications. These insights enable POLYRACK engineers to improve cooling performance while simultaneously saving time and reducing costs.

"The ability to simulate the effect of airflow characteristics with SOLIDWORKS Flow Simulation allows us to address heat transfer issues in software instead of through extensive and expensive prototyping," Knab explains. "Without simulation capabilities, optimizing the cooling system for this racked configuration of 10 boards would have taken three months or longer. With SOLIDWORKS Flow Simulation, we completed the work in just two weeks."

Because SOLIDWORKS Flow Simulation is integrated within SOLIDWORKS design software, POLYRACK can take advantage of design configurations to efficiently run heat transfer analyses on a range of different components. "We use configurations to run simulations on five different heat sink designs, for example, to determine which option will work best," Knab notes. "We only have to define the problem once, and then can run all five simulations at once, which saves a lot of time."

With embedded CFD in their 3D CAD software, Polyrack has been able to reduce development time from three months to two weeks, Cut two prototyping cycles as well as innovate with new approaches to electronics cooling system design.

### IDEX HEALTH & SCIENCE LLC

A division of IDEX Corp., IDEX Health & Science manufactures the precision microfluidics system components—including valves, pumps, manifolds, fittings, injectors, and tubing systems—that support sophisticated laboratory instruments.

**"Because SOLIDWORKS solutions are so intuitive, we can focus on our designs and not on the tools. This allows us to innovate more precise designs while simultaneously streamlining our development processes."**

Kevin Longley,
Mechanical Design Engineer

To achieve microfluidics breakthroughs, the company's engineers needed access to integrated design and simulation tools. IDEX decided to migrate to integrated SOLIDWORKS® solutions in 2001. "IDEX standardized on SOLIDWORKS because it's easy to use and facilitates training, yet provides powerful structural and fluid-flow simulation tools with SOLIDWORKS Simulation and SOLIDWORKS Flow Simulation."

As the complexity of IDEX designs has grown, so has its reliance on SOLIDWORKS Flow Simulation tool. "Much of what we do would be near-impossible to do by hand," Kevin Longley, Mechanical Design Engineer, stresses. "The tubing we use is 1/32nd of an inch in diameter, but the inner diameters are 0.004 inches. With such small passages and such high pressure driving such small fluid volumes through the system, simulation is a must. We conduct flow simulations on products that involve mixing"

Using SOLIDWORKS solutions, IDEX has shortened its design cycle by 50 percent while increasing innovation. In addition to saving time using SOLIDWORKS simulation tools, IDEX realizes time savings from SOLIDWORKS design configuration capabilities and the capability to compare these design configurations in SOLIDWORKS Flow Simulation.

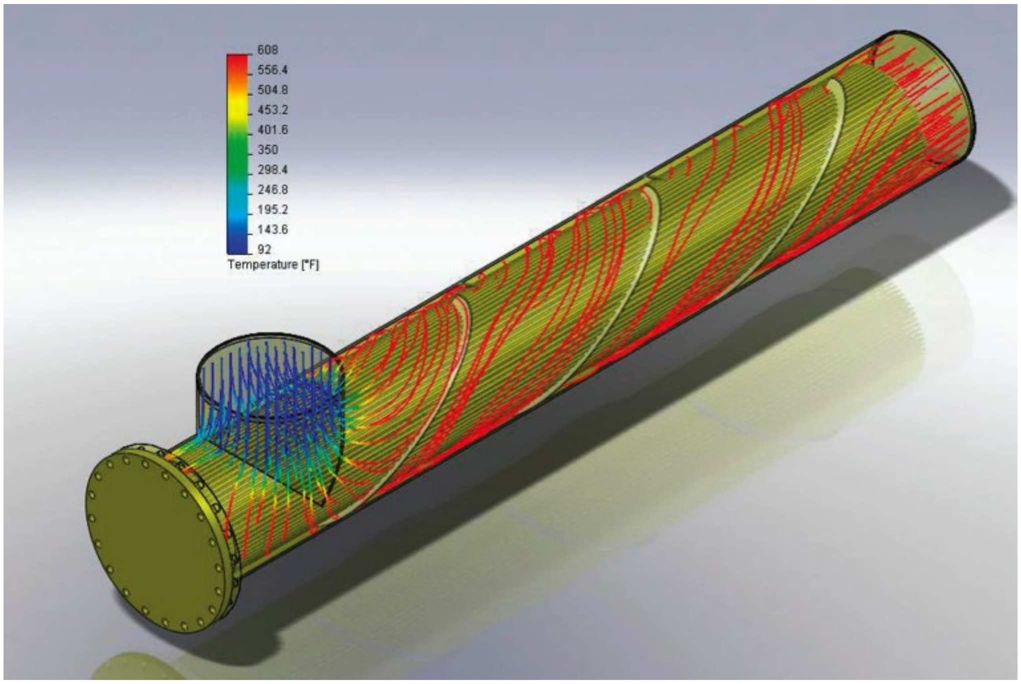
"Configurations help because our product is 99 percent components," Longley points out. "The liquid end of our valves is a stator. While the stator body remains the same, it could have many port configurations. Some have two ports; others have up to 25 ports, and the angles of the holes vary widely. With configurations, we can efficiently model all of the potential configurations from our initial design, which saves a lot of time."

With SOLIDWORKS Simulation and SOLIDWORKS Flow Simulation analysis capabilities, IDEX Health & Science has been able to increase design complexity and innovation.

## GAUMER PROCESS

Gaumer Process, Houston-based manufacturer, develops electric process heater, acquiring several patents for its electric process heaters, systems, and controls.

Given its deep commitment to quality, Gaumer Process often overdesigned and over engineered its heaters. Recently, however, market demands to accelerate system delivery, control costs, reduce energy consumption, and optimize material usage prompted the company to evaluate simulation technology.




**"Instead of using brute force and bloody ignorance to overcompensate, we can design our supports and braces using thinner materials in a way that more accurately meets the needs of the actual operating environment. The cost savings we realize help us to save our customers money while offering the optimum design at the same time."**

Craig Tiras,
Vice President of Engineering
and Design.

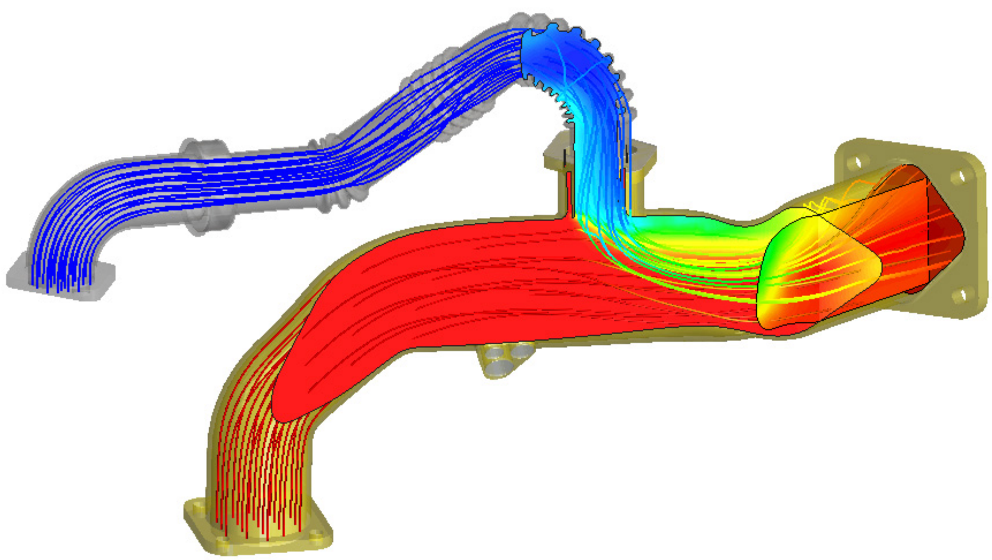
Gaumer Process uses SOLIDWORKS Flow Simulation to improve heat transfer performance. For instance, the company's engineers believed that an internal baffle design could enhance heat transfer within its electric process heaters.

Without SOLIDWORKS Simulation tools, Craig Tiras, Vice President of Engineering and Design, says Gaumer engineers most likely would have pursued a cross-baffle design—four times better, theoretically—and then would have worked through trial and error to optimize it. That process would have taken three years.

However, by using SOLIDWORKS CFD and thermal analysis software to simulate heat transfer in a variety of concepts, Gaumer was able to show that an optimized scissor-baffle design performed best.

"With SOLIDWORKS Flow Simulation software, we were able to study and test six different concepts and reach an optimized design in less than three months," Tiras points out. "We eliminated more than two years of costs, saved $100,000 on prototyping, and produced a patented idea for enhancing heat transfer. That's the kind of advantage that helps us beat our competition."

With SOLIDWORKS Simulation and SOLIDWORKS Flow Simulation software, Gaumer Process has reduced design times and cut costs on the development of its electric process heaters, while maintaining quality.



Fluid Mixing Analysis in CAD Embedded CFD Simulation solution



**Americas**
Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA

**Europe/Middle East/Africa**
Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

**Asia-Pacific**
Dassault Systèmes
Pier City Shibaura Bldg 10F
3-18-1 Kaigan, Minato-Ku
Tokyo 108-002
Japan