WHAT THEY DIDN'T TEACH YOU IN ENGINEERING SCHOOL ABOUT 3D PRESSURE DROP ANALYSIS

MENTOR GRAPHICS

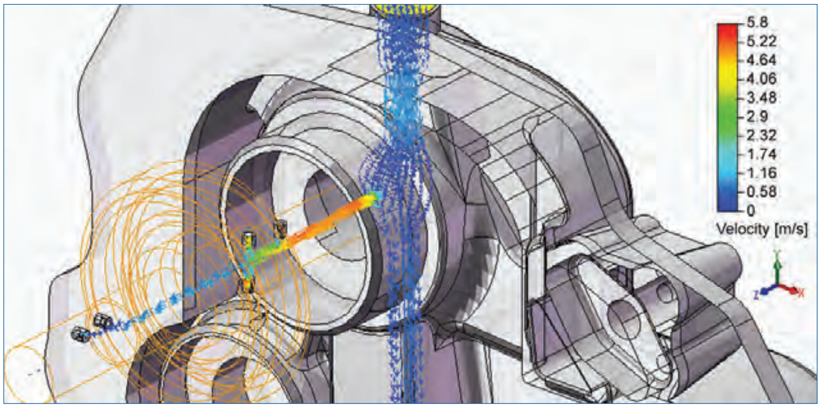
MECHANICAL ANALYSIS

WHITEPAPER

www.mentor.com

INTRODUCTION

Computational Fluid Dynamics (CFD) analysis is no longer a discipline reserved only for highly trained practitioners. A new class of CFD analysis software known as "Concurrent CFD" is proving to be greatly effective at performing pressure drop analysis, enabling mechanical engineers to accelerate key decisions at their workstations, without the need for CFD specialists. Embedded into the MCAD environment, this intuitive process allows designers to optimize a product during the design stages, reducing manufacturing costs across a wide range of mechanical designs and systems.



FloEFD, the industry's first concurrent CFD solution, embeds in leading MCAD environments and accepts device geometries directly.

Until recently, the commercial software available for CFD has typically been geared towards specialists, limiting its widespread use. Besides being expensive, these tools have either been difficult, cumbersome or time-consuming to use. As a result, engineering analysis for applications such as pressure drop traditionally have been carried out by specialists in analysis departments, separate from mainstream design and development departments.

To test or verify their designs, mechanical engineers therefore had to rely on creating physical prototypes and testing them on a flow bench or test rig. But this labor-intensive approach often led to incomplete results, limited to readings at discrete locations, making it difficult to thoroughly understand and characterize the underlying flow.

Fortunately, new tools have emerged that embed a complete range of flow and heat transfer analyses including pressure drop simulation within mainstream MCAD toolsets including CATIA® V5, Creo™, and Siemens NX™. The FloEFD™ design/analysis technology offered by leading simulation software company Mentor Graphics is aimed specifically at the mechanical design engineer. With FloEFD there is no need to hire or train CFD specialists, outsource analysis to consultants, or conduct tests on expensive multiple physical prototypes.

Instead, a design engineer with standard training and working in any size company can use his or her existing knowledge to successfully perform pressure drop analyses, entirely within today's familiar leading MCAD environments. FloEFD can improve design productivity and may dramatically reduce the number of physical prototypes needed. Equally important, it encourages engineers to explore many more 'what-if' scenarios to perfect their designs.

Certainly there will always be a few very demanding applications where more advanced CFD knowledge is needed to fine-tune the meshing and solver settings in order to converge to a solution. But taking CFD out of the exclusive domain of specialists and bringing it into the mainstream with FloEFD enables design engineers with no specific training in CFD to analyze problems in about 80% to 90% of all cases. This amounts to a fundamental breakthrough in design efficiency.

PRESSURE DROP ANALYSIS IN THE CAD ENVIRONMENT

Mentor Graphics FloEFD CFD simulation software combines all phases of pressure drop analysis in one package, from solid modelling, to problem set-up, running, results visualization, validation, and reporting. Typical pressure drop applications include flows through valves, manifolds, heat exchangers, filtration systems, electronics enclosures and ducting; in fact any system where the goal is to reduce the amount of energy required to move flow or to maximize its capacity.

With FloEFD, designers can focus on analyzing in detail why the flow of gas or liquid may be at a higher or lower pressure than that allowed in the technical specification. They can run ‘what-if’ scenarios and then optimize their design’s geometry within the MCAD tool. All the designer needs is knowledge of the MCAD system and the physics of the product. After installation of FloEFD, all the menus and commands necessary to run a full CFD flow analysis are created in the CAD package’s menu system. This close interaction between the MCAD system and FloEFD makes it extremely easy to use. In fact, most designers are ready to use FloEFD with less than eight hours of training.

The most common engineering task for fluid flow applications is to minimize the pressure losses in a system as a fluid flows from point A to point B. The basic engineering challenge is to either maximize the flow rate for a given pressure drop or minimize the pressure drop for a given flow rate. If the flow is driven by a pump or fan, then understanding the pressure drop enables the designer to optimize the size of the fan or pump.

The starting point of any flow analysis is to clearly describe the geometry of the mechanical system. FloEFD lets a designer take advantage of existing MCAD models for analysis, without having to export or import additional data, saving significant amounts of time and effort. The embedded FloEFD toolset can use newly-created or existing 3D CAD geometry and solid model information to simulate designs in real-world conditions. There is no need to create a separate model of the fluid region, a tedious process in traditional CFD tools. FloEFD recognizes the appropriate fluid region based on the empty internal spaces within the solid model, where the designer has placed boundary conditions.

FloEFD can also analyze a range of fluids. This includes gases—starting with the subsonic regime up through transonic, supersonic, and hypersonic flow—liquids and non-Newtonian fluids such as plastic flows, as well as flows for food processing applications. Even steam flow can be simulated. There is also a two-phase cavitation model and a combustible mixture simulation.

Once a model is created, it needs to be meshed. Developing a mesh is one of those skills that previously separated CFD specialists from mechanical engineers. With FloEFD, the base mesh is created automatically in a matter of minutes rather than requiring hours of tedious proportioning of regions and cells. FloEFD actually creates a solution-adaptive mesh that increases the cell density (increasing the resolution of the analysis) to ensure more accurate simulation results in critical flow regions of the model, as shown in Figure 1.

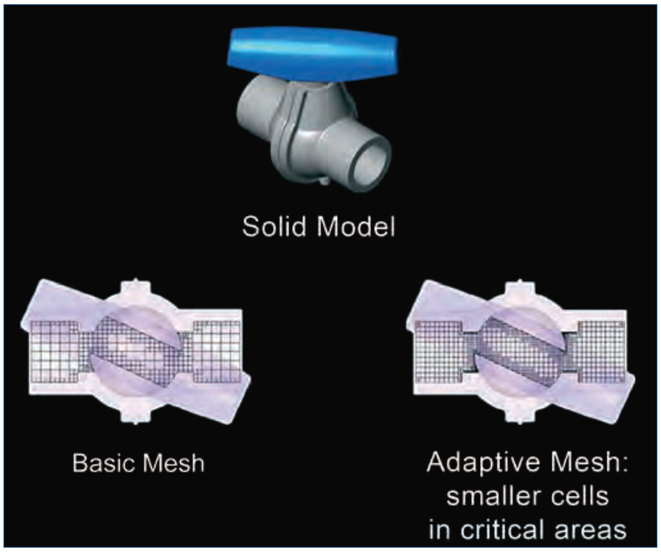


Figure 1: Using a rectangular adaptive mesh, the FloEFD concurrent CFD tool can automatically adjust cell size to deliver better resolution anywhere it is needed.

BENEFITS OF CONCURRENT CFD

Concurrent Computational Fluid Dynamics (CFD) is a breakthrough technology that enables design engineers to conduct up-front, concurrent CFD analysis throughout the product's lifecycle, using the familiar MCAD interface, thus reducing design times by orders of magnitude compared to traditional methods and products. Concurrent CFD can reduce simulation time by as much as 65 to 75 percent in comparison to traditional CFD tools, and enables users to optimize product performance while reducing physical prototyping and development costs without time or material penalties.

In **traditional CFD**, the model geometry is first exported from the CAD system. The geometry then needs to be re-imported into the user's CFD tool, meshed, solved, the results post-processed and finally reported back to the design team. The work is usually done by a specialist analysis group, or outsourced, so it is necessary for the design team to explain what needs to be done. By the time the results are in, the analysis model has become 'stale', as the design team often has moved on, making it difficult to act on the results.

Upfront CFD attempts to improve this situation by streamlining the interface between the CAD and the CFD tool. Although this results in a much cleaner import of the geometry, the analysis is still performed outside of the CAD system. The frequent transfer from the CAD and CFD software can result in a degradation of information.

In addition, both of these approaches require the creation of a 'cavity' to represent the flow space. Most conventional CFD meshing tools work by meshing a solid, so there is no such thing as an empty space. To work around this limitation, the designer must create a solid object that represents the flow space and then use Boolean subtraction to remove the dummy model from an encapsulating solid. This is usually done in the CAD system and this inverted flow space then is transferred to the CFD system for meshing. Obviously, this is a labor-intensive process that can easily introduce errors into the design and analysis.

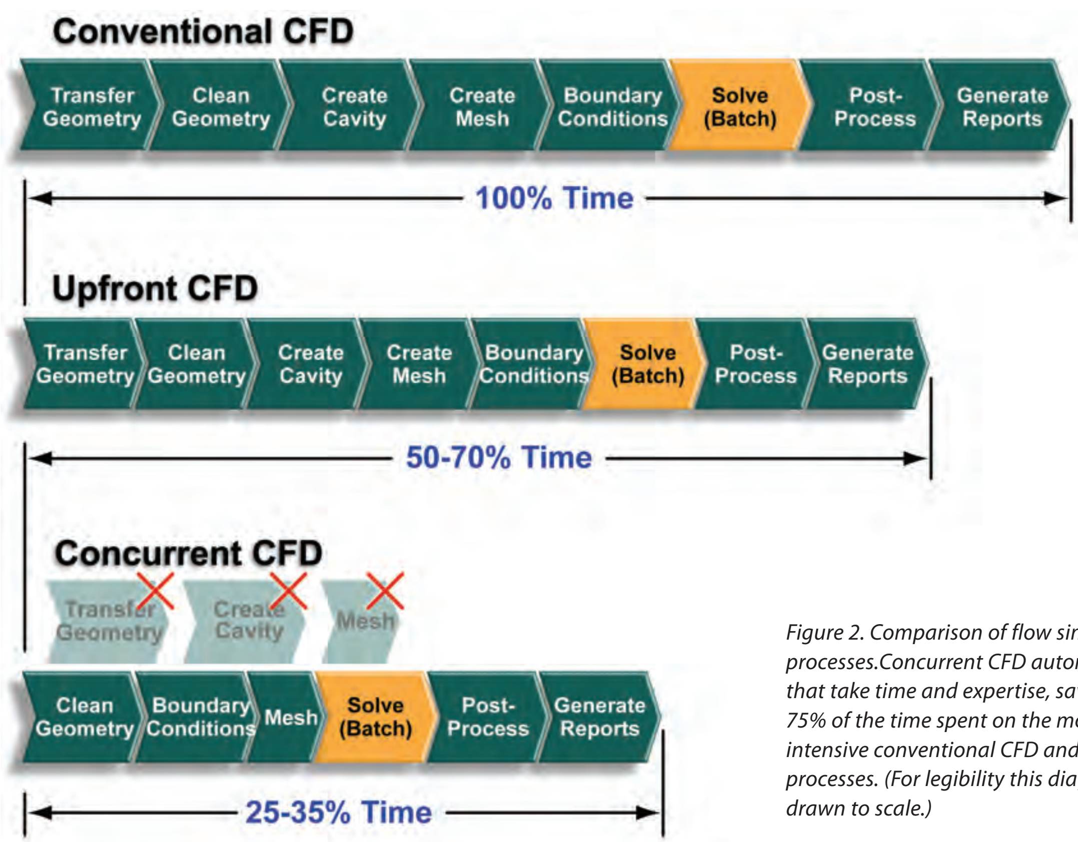


Figure 2. Comparison of flow simulation processes.Concurrent CFD automates steps that take time and expertise, saving up to 75% of the time spent on the more labor-intensive conventional CFD and upfront CFD processes. (For legibility this diagram is not drawn to scale.)

Concurrent CFD operates very differently. It is CAD-embedded CFD so the work is done within the designer's familiar MCAD environment. Design changes necessary to achieve the desired product performance are made directly on the MCAD model, so the design is always up-to-date with the analysis. Preparing a model for analysis is very easy with FloEFD. Unlike traditional CFD programs that require users to create additional solid parts to represent the fluid (empty) regions, FloEFD automatically differentiates between the MCAD geometry for internal and external flows and automatically creates the fluid domain. As a result, engineers are able to concentrate on their project as opposed to creating extra geometry in their CAD system, minimizing confusion and saving them time and effort.

SOLVING ADVANCED PRESSURE DROP CHALLENGES

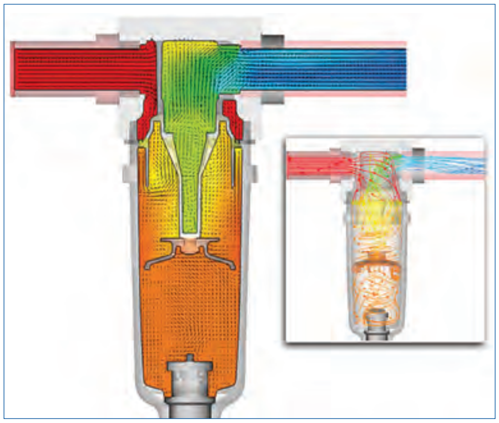
FloEFD provides an extensive ability to visualize what is happening to a design's flow, giving the engineer valuable insight that can guide design decisions. The visualization capabilities allow users to interrogate the design more thoroughly and visualize the flow field.

For example, when analyzing pressure drop there are often many flow passages much smaller in scale than the majority of the device. A valve design, for instance, might include a perforated insert with small holes that the flow must move through. Capturing this somewhat complex geometry and re-meshing it between successive design iterations would be a tedious task using a traditional CFD tool and would require advanced meshing knowledge. In contrast, using the auto-mesher in FloEFD the designer can easily enter the size of the perforated holes to guide the mesher in creating the correct size of the flow channels. A quality mesh that will give accurate answers will be automatically generated, enabling the designer to efficiently evaluate the impact on the overall performance of the system.

A two-dimensional way to examine the flow field in FloEFD is to use a cut plot, which depicts the flow on a plane through the model. A cut plot of results can be displayed with any results parameter and the representation can be created as a contour plot, ISO lines, or as vectors. It can also be created in any combination such as velocity magnitude, and velocity vectors. In addition to cut plots, a 3D surface plot can be easily displayed for any particular face as well as automatically for the entire flow domain.

FloEFD is also offers a powerful way to examine yet another parameter of interest in pressure drop analysis: total pressure. In real viscous flows, there are losses in total pressure as the fluid flows through a design contours. So areas of total pressure gradients indicate places where there are viscous losses of energy that cannot be recovered. Using FloEFD, designers can easily pinpoint these sections and focus their efforts on improving them to optimize the overall flow of the design.

Solving any of these pressure flow problems is an iterative process. After seeing the initial analysis results, most designers want to modify their models to explore different scenarios to see if they can optimize the flow. FloEFD makes it easy to conduct these "what-if" analyses. Designers can explore design alternatives, detect design flaws, and optimize product performance before detailed designs or physical prototypes are created.



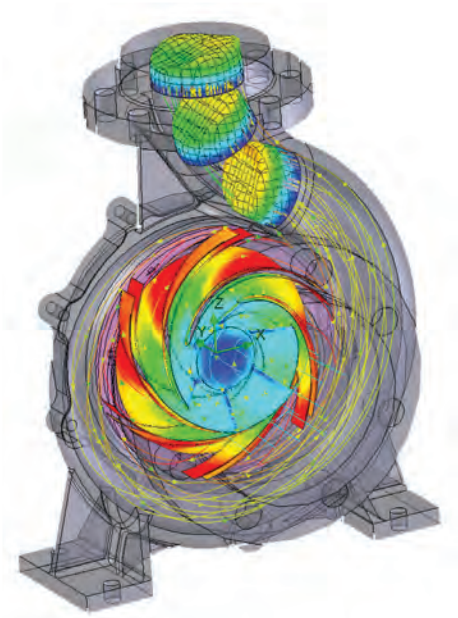
FloEFD handles challenging filter simulations with multiple filter modeling options, complex filter housings and comprehensive 3D visualizations.

This allows the design engineer to determine quickly and easily which designs have promise, and which designs are unlikely to be successful.

To examine alternatives, the designer simply creates multiple clones of the solid model in FloEFD that automatically retain all analysis data such as loads and boundary conditions. When the engineer modifies a solid model, he or she can immediately analyze it without having to re-apply loads, boundary conditions and material properties.

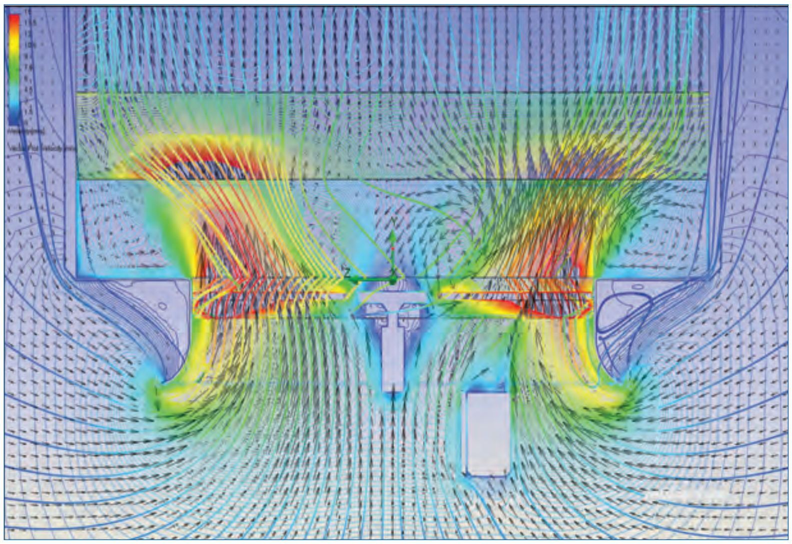
With traditional CFD software, after each geometrical change it is necessary to re-create the mesh which usually involves time-consuming manual intervention. In contrast, FloEFD software operates immediately on the modified geometry, creating a new mesh automatically and working with the previously defined boundary conditions. Thus, the step from a changed geometry to running the solver and examining results is greatly accelerated.

The software also aids in parametric optimization—for example, automatically running a design-of-experiments with various wall thickness scenarios to determine the optimal thickness. In these ways, FloEFD accelerates the iterative design process, allowing engineers to quickly and easily incorporate knowledge gained in an analysis into an improved design.



FloEFD can be used to determine pump efficiency and to optimize the design of components such as the housing, impeller, inlet and outlet sections. (Image courtesy of Johnson Pump).

FloEFD provides robust verification capabilities for validating designs. Before releasing a new version of FloEFD, Mentor Graphics’ engineers validate the release with a suite of 300 tests. Based on this rigorous verification suite, FloEFD offers 26 tutorial benchmark examples ready for immediate use. For example, designers could use these tutorials to validate the flow in a 2D channels with bilateral and unilateral expansions and parallel walls. Or they could verify the classic pressure drop benchmark for CFD: the flow in a 90-degree bend of a 3D square duct.



With its sophisticated multiple reference frame models, FloEFD can simulate complex fan characteristics, depict the flow and pressure fields surrounding a fan unit, and identify potential areas for improvements in energy efficiency and noise output. (Image courtesy of Bronswerk Heat Transfer BV).

Figure 3 highlights the strong correlation between FloEFD verification results and experimental data for secondary turbulence flow field activity in this type of design. Once the results are available, the design engineer needs to report his/her findings to others. FloEFD is fully integrated with Microsoft® Word® and Excel®, allowing engineers to create report documents and collect important data in graphical form from any FloEFD project. In addition it automatically creates Excel spreadsheets summarizing the outputs of an analysis.

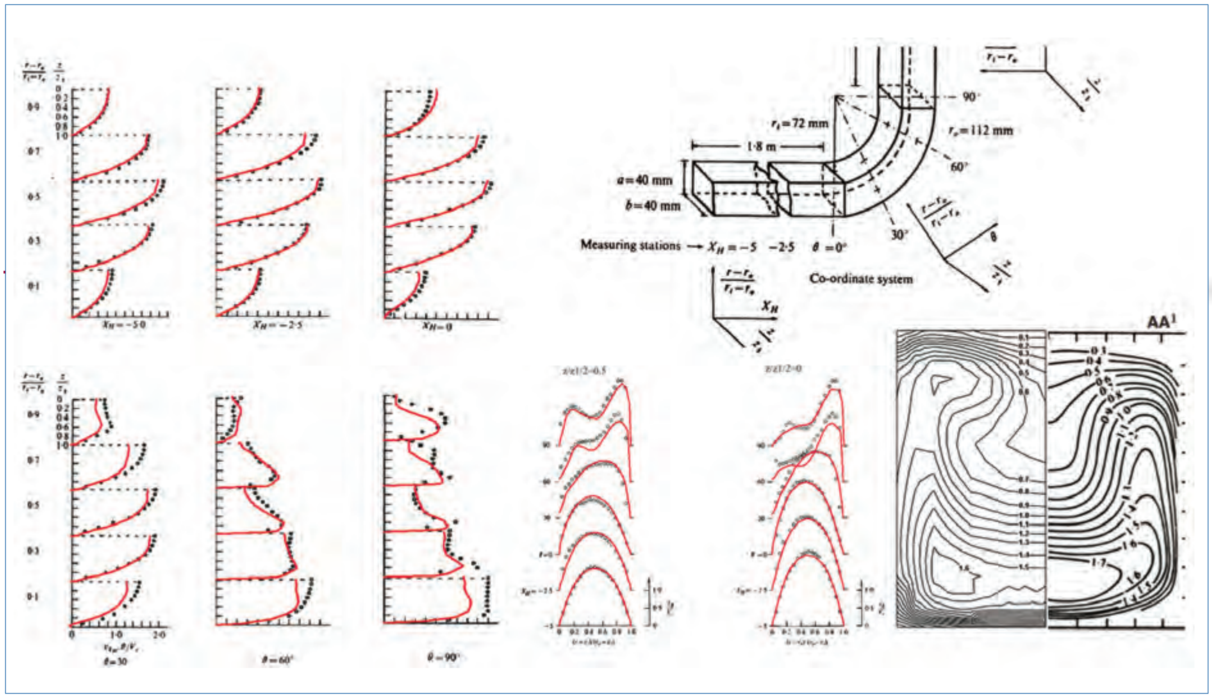
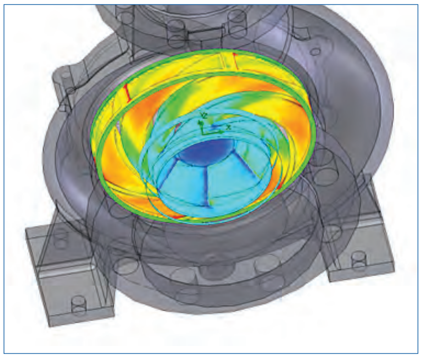
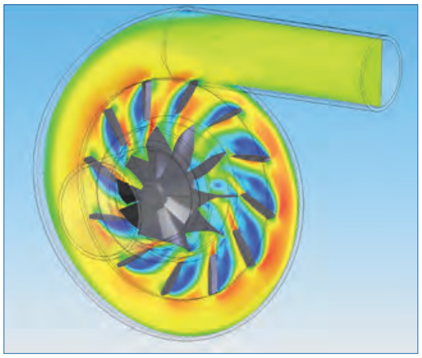


Figure 3: Velocity profiles at different cross-sections in different longitudinal planes. FloEFD validation and experimental results correspond closely in this example showing flow in a 90-degree bend of a 3D square duct.

REAL WORLD DESIGNERS AND PRESSURE DROP ANALYSIS

With FloEFD, designers can focus on improving product performance and functionality without requiring them to become full-time fluid dynamics specialists. The following real-world examples, accumulated over the past few years, demonstrate the effectiveness of FloEFD in helping designers meet tight deadlines, achieve higher quality results and/or minimize costs.





MARENCO AG USES FloEFD TO REDUCE EXTRACTOR HOOD DESIGN TIME

Marenco AG used FloEFD to design a new extractor hood for one of their clients. The combined design and analysis time required only half the time when compared to traditional engineering methods. Placed over a cooking range, extractor hoods remove grease, smoke and odors from the kitchen. They rely on a combination of a fan and a filtration system to extract and or clean the air. The new extractor hood designed by Marenco AG is not only more efficient but also operates at a lower decibel level than others. In consumer environments, a premium is placed on these extractor hoods as they create less noise.

"Using FloEFD enabled us to develop this new product in record time and significantly reduced the number of physical prototypes we needed to test."

Hans-Peter Keel,
Mechanical Design Engineer,
Marenco AG

"Using FloEFD enabled us to develop this new product in record time and significantly reduced the number of physical prototypes we needed to test" said Hans-Peter Keel, Mechanical Design Engineer at Marenco AG. "Because our team was able to develop this model so quickly our customer was able to take their product to market much faster than their competitors."

As a design specialist, Marenco AG relies on a combination of Siemens NX for solid modeling and FloEFD for computational fluid dynamics analysis (Figure 4). FloEFD proved especially helpful in their latest project as it is extremely difficult to see inside the exhaust enclosure and how the various components interact with one another. "When you have airspeed over nine meters per second you get a fair amount of noise. FloEFD helped us find the point where the noise originated so we were able to modify our design to reduce noise," Keel explained. Also, extractor hoods rely on a filtration system to remove smoke and fat particles from the air. "Through simulation we predicted the most impacted area of the filter and the best geometry for the porous filter material."

The entire design, simulation and physical testing process took half the time it would have taken using traditional design processes. Until now, Marenco AG had tested their designs by building a physical prototype and comparing it with the simulation results. However, FloEFD simulation helped them reduce the number of physical prototypes drastically and still get reliable performance information very quickly. Keel sums up his experience saying, "With FloEFD, I can do what I need to do faster and better by locating the critical problem areas quickly. It can be very difficult to investigate problems on a physical prototype but with FloEFD you can put your finger right on the problem and fix it immediately."

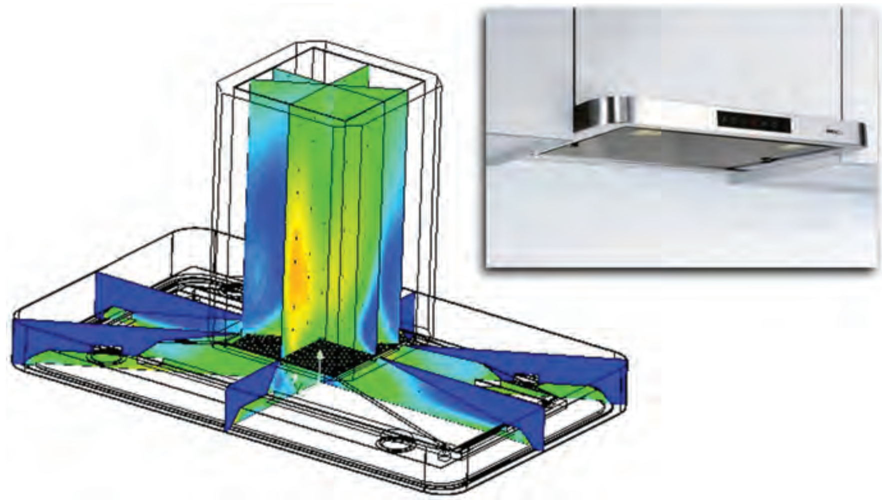


Figure 4: Marenco AG used FloEFD to design a new kitchen extractor hood for a client. The result is a hood that is not only more efficient that past designs but also operates at a lower decibel level than others, meeting the client's requirements. (Image Courtesy of Marenco AG)

FloEFD HELPS STURMAN INDUSTRIES BRING SPACE TECHNOLOGY TO EARTH

FloEFD is helping commercialize technology that once helped land men safely on the moon. Feet firmly planted on the ground, Woodland, Colorado-based Sturman Industries is using FloEFD to simulate fluid flow and refine high-speed, energy efficient, valve designs.

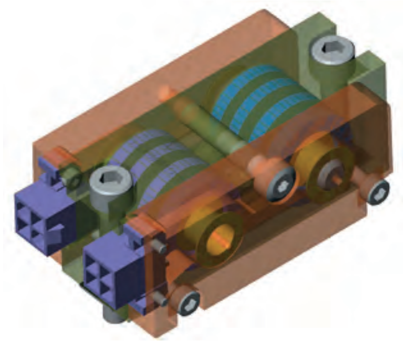


Figure 5: Sturman Industries magnetically latched fluid valve.

Sturman Industries' co-founder and namesake Eddie Sturman invented digitally- controlled, magnetically-latching valves the Apollo Space Program. By creating a magnetic latch on either side bar of a specially designed spool (Figure 5), and controlling the latch with sophisticated electronic controls, the spool is passed back-and-forth at tremendous speeds and remarkable precision to ensure accurate fluid control. Industrial applications for the technology range from automotive and trucking projects to irrigation and even carpet manufacturing.

Steve Massey, a Mechanical Engineer with Sturman Industries, explains: "The reasons for our growth are our technology, which is very unique, as well as a great relationship with International Trucks, who is using our products for both diesel fuel injection and a camless application that handles the hydraulic actuation of intake and exhaust valves. You can't vary how much mechanical valves open but, by manipulating our digital valves, a truck driver could theoretically turn cylinders completely off to slow the truck or increase fuel economy."

Massey conducts a wide variety of analysis and engineering support for Sturman Industries' design team and had been searching for a product like FloEFD for months. "There is another engineer on staff who is an analysis expert and who spends virtually all of his time conducting CFD simulations. My primary responsibility is design and I was immediately drawn to FloEFD because I want to do design verification rather than spend all day setting up a simple flow problem."

According to Massey, FloEFD has a very short learning curve and Massey found it so intuitive that he didn't even complete the training tutorial. "Programs like FloEFD are all about ease of use. They are tailored for designers, not CFD specialists. This is very important because most of the stuff that we do is design verification of non-transient, incompressible flow problems."

> "I would say that FloEFD has definitely paid for itself ten times over."
>
> Steve Massey,
> Mechanical Engineer, Sturman Industries

Design optimization is, indeed, the key application of FloEFD on Sturman Industries' HVA (Hydraulic Valve Actuation) product line. Massey's HVC II valve design received several hundreds of hours of FloEFD analysis (Figure 6) and was the first design in which the tool was absolutely crucial. "FloEFD probably drove 80 percent of the HVC II design because we had to maximize flow by minimizing radial and axial forces. I would say that FloEFD has definitely paid for itself ten times over."

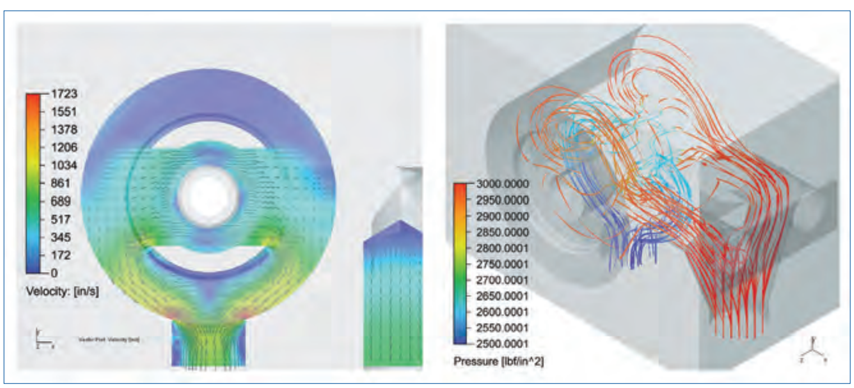


Figure 6: Sturman Industries designers relied on FloEFD for more than 80% of design for its Hydraulic Valve Actuation (HVA) product line. (Image courtesy of Sturman Industries)

VENTREX AUTOMOTIVE: DRIVING FASTER TIME TO MARKET WITH FloEFD

Ventrex Automotive GmbH in Graz, Austria is a supplier of compressors and air conditioning valves to all major automobile manufacturers. One recent challenge was to develop valves suited for new CO_2 refrigerants that are supplanting those based on hydrofluorocarbons. These new fluids operate at pressures seven to ten times higher and require a redesign of many air-conditioning system components such as the valves used to evacuate and charge the system.

> "We were able to improve the flow rate of our new CO_2 valve by 15% while eliminating roughly 50 prototypes and reducing time to market by 4 months."
>
> Daniel Gaisbacher,
> Project Manager, Ventrex Automotive

Ventrex Automotive has long used CATIA MCAD tools, so they selected FloEFD, which integrates directly into CATIA V5. Without leaving the mechanical CAD environment, Ventrex Automotive engineers can simply execute a menu selection that invokes the FloEFD software to simulate the design. A key advantage of FloEFD is that it can simulate the pressure drop in a new design without the need to build a prototype. It also provides diagnostic information such as the flow velocity and direction at every point in the flow field so engineers can determine the optimal design

Because FloEFD automatically identifies the voids within the valve where fluids could flow, the engineers simply had to specify the boundary conditions, in this case the inlet and outlet pressures. "Within a few hours we had a complete simulation of the initial concept design and were able to turn our attention to improving it," says the firm's project manager. The result? "FloEFD makes it possible to determine simulation results nearly as fast as we can change the design. We were able to improve the flow rate of our new CO_2 valve by 15% while eliminating roughly 50 prototypes and reducing time to market by four months."

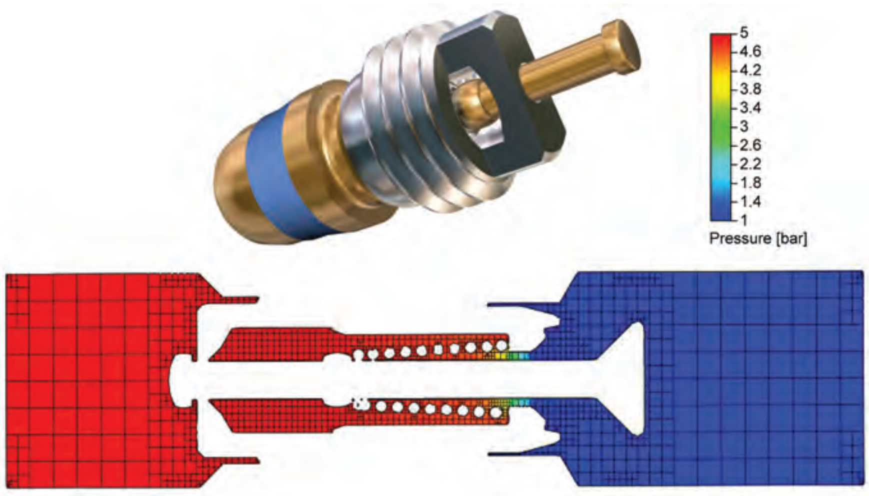


Figure 7: Ventrex Automotive used FloEFD to improve the flow rate of this CO_2 valve by 15%.

BOLL & KIRCH FILTERBAU MAKES THE MEASURE WITH FloEFD

Boll & Kirch Filterbau GmbH, Kerpen, Germany, a leading filter manufacturer, uses FloEFD to simulate pressure losses so small that they would be very difficult to measure with a prototype. "There is an American standard that says an oil filter in a clean state may generate a pressure loss no greater than 0.35 bar," said Karsten Cartarius, Research and Development Team Manager for Boll & Kirch Filterbau GmbH. In the gas sector maximum pressure loss may be as low as 0.05 bar, an extremely low number.

> "We used FloEFD to simulate the pressure drop with astonishing speed and accuracy."
>
> Karsten Cartarius, Research and Development Team Manager, Boll & Kirch Filterbau GmbH

These values have been reduced considerably in recent years because smaller pumps are being used on many types of equipment to save energy. "We need to be able to prove that the pressure drop in a filter really is 0.05 bar. It is not practical to measure pressure drops this low. But we can use FloEFD to simulate the pressure drop with astonishing speed and accuracy. The simulations can be performed at a very early stage in the design process to ensure that the proposed design is capable of meeting the standard."

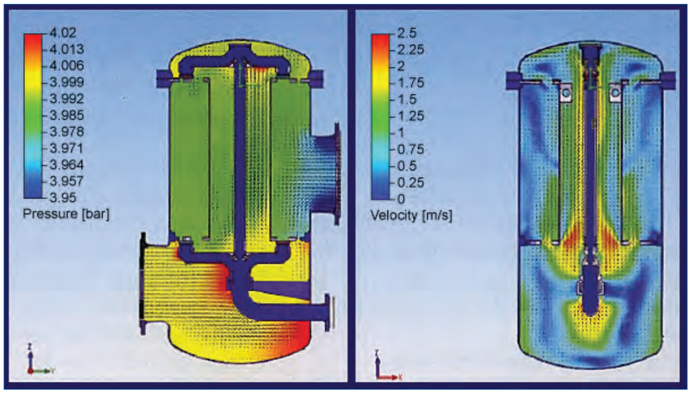


Figure 8: Boll & Kirch Filterbau GmbH uses FloEFD to simulate pressure losses so small that they would be very difficult to measure with a physical prototype. (Image courtesy of Boll & Kirch Filterbau GmbH)

SHAW AERO DEVICES ENGINEERS BOOST FUEL VALVE PERFORMANCE

Shaw Aero Devices, Inc. (Naples, Florida, now a division of Parker Hannifin Corp) manufactures valves for aerospace applications. A Shaw customer was poised to order a large quantity of a solenoid valve very similar to one of Shaw's off-the-shelf products. But first Shaw had to meet some performance conditions stipulated by the customer.

The order was for fuel control valves for an unmanned aerial vehicle (UAV). Six of these valves operate in a manifold to route fuel to distributed tanks and keep the aircraft in balance as fuel is consumed. The customer wanted to increase the aircraft's payload by using a smaller fuel pump—a pump too light to overcome the pressure drop of the standard Shaw valves. To earn the order, Shaw Aero Devices would need to reduce the pressure drop by almost 88% in the valve. And the whole project had to meet a very aggressive timeline. That is why engineering team decided to rely on Mentor Graphics FloEFD.

"In the past we would have had to make an educated guess on what was raising the pressure drop in the valve" said Shaw project engineer Rob Preble. "For each guess we would build a prototype to see if we were right or not. Each prototype would cost about $3000 and would take about a month to machine, assemble and test. One of the weaknesses of this build-and-test approach is that physical tests to determine the pressure drop in each new design iteration provide very little additional information to help us diagnose problems such as determining which areas of the valve are constricting fluid flow."

In contrast, Shaw Aero designers were able to complete the first software prototype in just one day, without any prior FloEFD experience. The simulation results revealed areas in which the old design could be improved and by repeating a series of simulations at various flow rates, the engineers located and corrected the problem areas in the valve Figure 9 depicts one of these iterations. FloEFD reduced the time required to simulate flow by analyzing each evolving model and automatically identifying fluid and solid regions without user interaction.

For example, after viewing the cut plot of the pressure gradient of the first design run, they noticed that the valve's angled wing seat was a key source of the pressure drop. So they modeled a new valve with a larger opening and without the angled valve seat. After several prototypes and revisions on a similar scale, the final design emerged.

> "The entire redesign was completed without building a single hardware prototype of the valve."
>
> Rob Preble, Project Engineer, Shaw Aero Devices

The entire redesign was completed without building a single hardware prototype of the valve. Ultimately the new design reduced the pressure drop from 6.09 psi (at a flow rate of 4.45 gal/min) to 0.71 psi, meeting the customer's exacting specifications. Equally important, the project met the urgent schedule requirements. Instead of a month-long development cycle for each of several hardware prototypes, Shaw engineers were able to complete virtual prototypes in as little as one day. Preble states, "We had complete confidence in the results because FloEFD provided such an accurate simulation of the original design. We took our simulation results to the customer and were awarded the contract to build the valve."

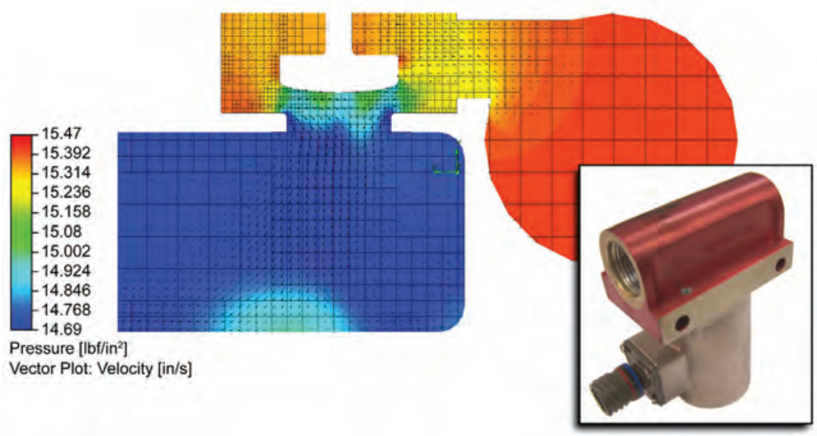


Figure 9: The original Shaw Aero Devices solenoid valve shown here exhibited a pressure drop of more than 6 psi. The Cut Plot shows the results of two design revisions that reduced the drop to about 1 psi. and subsequent revisions reduced the drop all the way to .71 psi, well within the customer's specifications.
(Image courtesy of Shaw Aero Devices)

Corporate Headquarters
Mentor Graphics Corporation
8005 SW Boeckman Road
Wilsonville, OR 97070-7777
Phone: 503.685.7000
Fax: 503.685.1204

Sales and Product Information
Phone: 800.547.3000
sales_info@mentor.com

Visit www.mentor.com/company/office_locations/ for the list of Mechanical Analysis Division Offices

MGC 08-11 TECH10190-w