

WHAT THEY DIDN'T TEACH YOU IN ENGINEERING SCHOOL ABOUT HEAT TRANSFER

MENTOR GRAPHICS



M E C H A N I C A L A N A L Y S I S

W H I T E P A P E R

www.mentor.com

Using Computational Fluid Dynamics (CFD) is no longer relegated to the realm of the specialist. A new class of CFD analysis software, 'Concurrent CFD', is proving to be highly effective at performing heat transfer analysis, enabling mechanical engineers to accelerate key decisions at their workstations and 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.

Until recently, the commercial software available for CFD typically has 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 heat transfer 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 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 thermal behavior.

Fortunately, new tools have emerged that embed a complete range of flow analyses including heat transfer simulation within mainstream MCAD toolsets such as CATIA® V5, Creo™ Elements/Pro™, 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 heat transfer analyses, all within the familiar MCAD environment of their choice. 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. However, 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 roughly 25% to 35% of the time it would take for using traditional tools. This offers designers a fundamental breakthrough in design efficiency.

HEAT TRANSFER ANALYSIS IN THE CAD ENVIRONMENT

Mentor Graphics FloEFD CFD simulation software provides a complete environment for performing heat transfer evaluation by combining all the phases of analysis in one single package: from solid modelling, to problem set up, solving, results visualization, optimizing the design and reporting.

With FloEFD designers can focus on analyzing, in detail, temperature distribution in the fluid and solid areas of their product. You can analyze complex physical processes such as heat conduction, heat convection, conjugated heat transfer between fluids, surrounding solid materials, radiation, joule heating and many more using 'what-if' scenarios, you then quickly modify and optimize the design's geometry within the MCAD tool, as shown in Figure 1. which illustrates the visualization of an oil cooler after simulation in FloEFD.

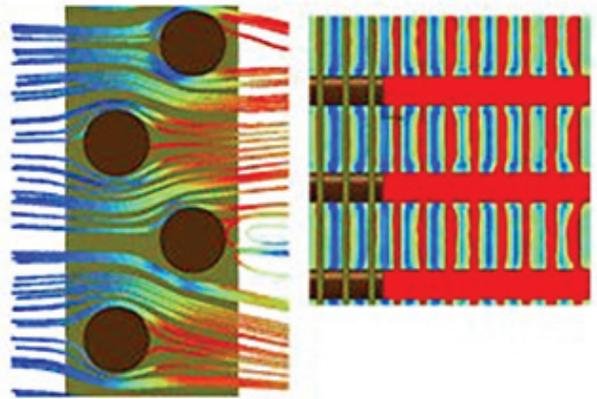


Figure 1: Visualization of heat transfer in an oil cooler after simulation in FloEFD.

FloEFD solves for all the three modes of heat transfer in 3D: conduction, convection and radiation which is why it can analyse a wide spectrum of applications. Typical temperature analysis applications include heat exchangers, injection molding device cooling, sterilization in food processing, solar towers, laser systems, brake design, and many others. In the case of a heat exchanger not only can you look at the efficiency on the thermal side but you will also be able to predict the pressure drop through the heat exchanger. Combining these parameters in one single model helps design a better product up front.

To use FloEFD software, 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 report that they can use FloEFD with less than 8 hours of training.

The starting point of any heat transfer analysis is to define the overall boundary conditions of the problem. FloEFD offers a wizard to direct the setup, including the selection of material properties. FloEFD allows a designer to take advantage of existing MCAD models for analysis, without having to export or import additional data, saving a significant amount 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.

Once a model is created, the model needs to be meshed. Developing a mesh is one of those skills that previously separated CFD specialists from mechanical engineers. With FloEFD, meshes are created automatically and in a matter of minutes as opposed to many hours of tedious proportioning of regions and cells. FloEFD actually creates an adaptive mesh that reduces the cell size, increasing the resolution of the analysis, to ensure more accurate simulation results in complex areas of the model, as shown in Figure 2.

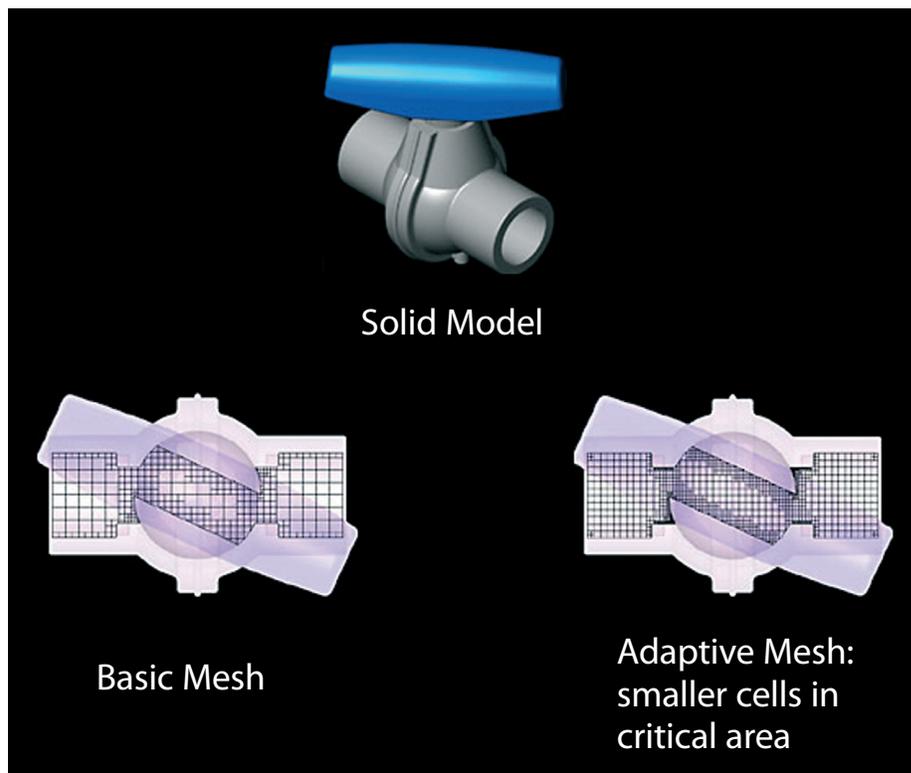


Figure 2: Using a rectangular adaptive mesh, Concurrent CFD tool FloEFD can automatically adjust cell size to deliver better resolution anywhere it is needed.

SOLVING ADVANCED HEAT TRANSFER CHALLENGES

When analyzing heat transfer, it is important to build a computational grid, or mesh, to capture the complex geometry of the system or device. The mesh is simple in concept, yet it is the heart of complex CFD calculations. The surface of the device is mapped into tiny rectangular cells, each of which is split into solid and fluid volumes which are analyzed discretely. The process then develops a composite result that incorporates all of the cells.

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

One way to examine the temperature field is to use a cut plot, which depicts the heat distribution on a plane through the model, as shown in Figure 3. A cut plot of results can be displayed with any results parameter and the representation can be created as a contour plot, isolines, or as vectors. It can also be created in any combination such as velocity magnitude, and velocity vectors. In addition to cut plots, a surface plot can be easily displayed for any particular face as well as automatically for the entire model.

Solving heat distribution problems is an iterative process. After seeing the initial analysis results, most designers want to modify their models to explore different scenarios. 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. This allows the design engineer to quickly and easily determine which designs have promise, and which designs are unlikely to be successful.

To examine alternatives, multiple clones of the solid model can be created in FloEFD that automatically retain all analysis data such as heat sources and other boundary conditions. When the engineer modifies a solid model, it can be analyzed immediately without having to re-apply boundary conditions and material properties.

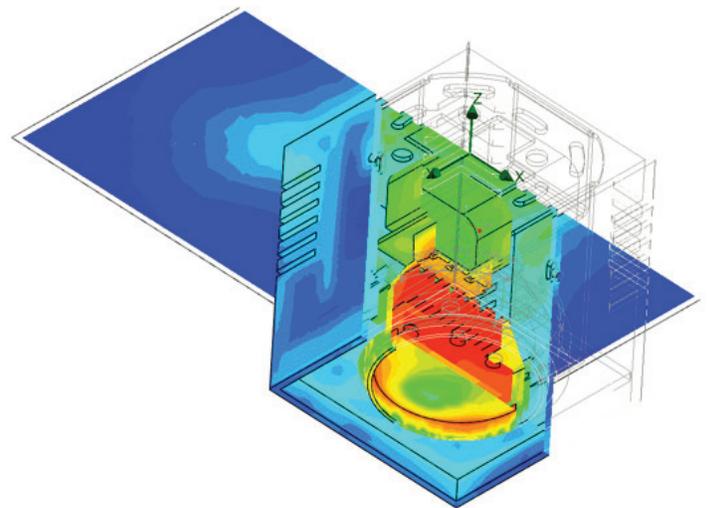


Figure 3: Temperature inside an LED lamp shown with a 50% slice.

With traditional CFD software, after each geometrical change it is necessary to recreate the mesh which usually involves time consuming manual intervention. In contrast, FloEFD software operates immediately on the changed 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 analysis—for example, running an analysis multiple times with various vent sizes to determine the optimal design for heat distribution. 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 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.

Sharing results and findings is simple. 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; thus making the last step in any analysis – that of creating reports -- effortless.

REAL WORLD DESIGNERS AND FloEFD HEAT TRANSFER ANALYSIS

With FloEFD, designers can focus on improving product performance and functionality without resorting to full time fluid dynamics specialists. The following are some real world examples that demonstrate the effectiveness of FloEFD in helping designers meet tight deadlines, achieve higher quality results or keep costs to a minimum.

THERMAL SIMULATION SIMPLIFIES LED LUMINAIRE DEVELOPMENT

Every form of electric lighting produces an unwanted by-product: heat. In the case of incandescent and fluorescent lighting, generations of engineers have developed ways to minimize and/or divert heat from luminaires and fixtures. However LED lights appearing today in growing quantity and variations, poses new and different challenges. Heat buildup can reduce an LED's light output and cause a color shift and at the same time, shorten the component's useful life. It has been said that thermal management is by far the most critical aspect of LED system design.

The key to successful LED system design is to transfer the active device's heat efficiently from its own PN junction to the ambient. The path involves both the printed circuit board, that mounts the LED, and the enclosure. The designer must confirm that housings and shrouds participate efficiently in carrying heat away from the LED.

The baseline for design validation, whether the product is a new luminaire or a replacement for an existing design, is of course a detailed understanding of the LED device's thermal behavior. This is also important when developing an LED lamp intended to retrofit into an existing fixture, as it is essential to match the thermal and cooling characteristics of the original equipment.

Thermal information should be available from the LED vendor, though in the absence of industry standards for such information, printed specifications are sometimes less than complete. For example, thermal performance data is normally provided, but may not span the operating temperature range at which practical end-user systems actually work. Nevertheless, designers must proceed with their luminaire evaluation based on data or compact models provided by the LED maker, in-house measurements, or experience with earlier luminaire designs.

The process starts with the mechanical design of the luminaire. Figure 4. illustrates the outcome of this initial design step. The depicted system includes an integral connector (shown in yellow) attached to a lamp housing whose fins act as a heat sink. The connector plugs into a socket which may itself be designed to further disperse heat, acting as part of the cooling system for the lamp. However in this particular system the socket is simply a means of supporting and connecting the lamp. The light source is a power LED mounted on a metal-core PCB. In Figure 4. the lamp's lens is omitted to expose a more detailed perspective on the LED itself.

FloEFD automatically models any internal cavities as fluid regions, a requirement for analysis. While the need for this step is more obvious for a pipe carrying a liquid, it is also necessary to predict the air flow through and around the luminaire.

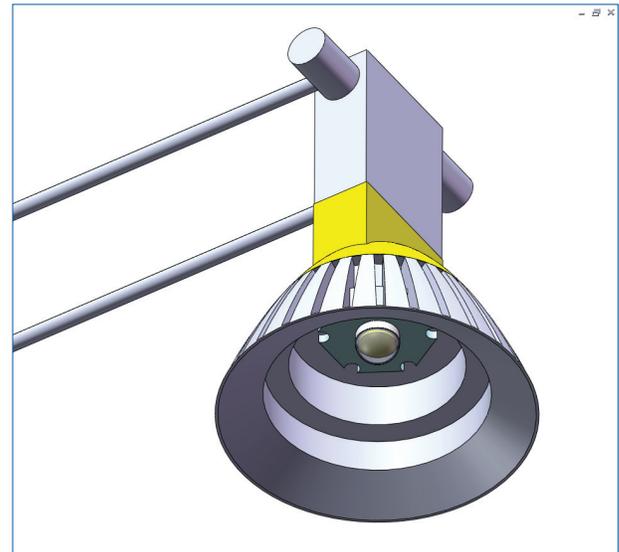


Figure 4: The MCAD representation of the LED luminaire

The next step is to create the computational grid, or mesh, which FloEFD does automatically. The mesh for the luminaire is shown in Figure 5.

Note that the cells in Figure 5. are not uniform in size. Those clustered around the LED are much smaller than those on the periphery of the housing. This feature found in FloEFD provides higher resolution where it is most needed.

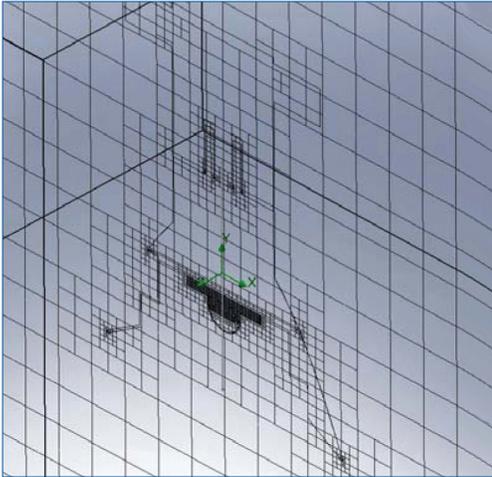


Figure 5: Computational mesh shown on a central slice through the lamp assembly.

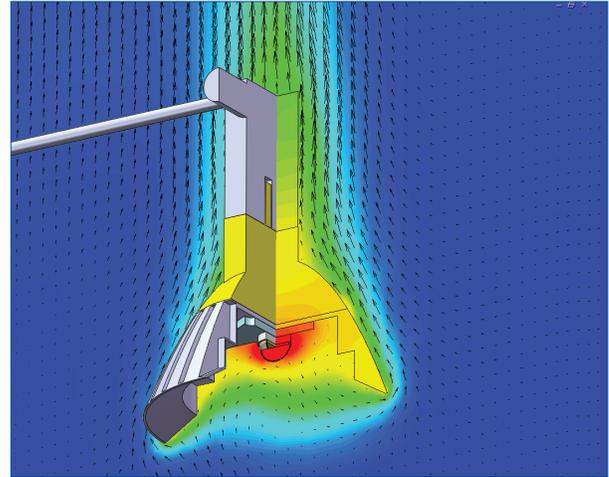


Figure 6: Temperatures on a 2D slice through the center of the lamp with vectors showing flow caused by natural convection over the outside of the lamp. The vectors in this and subsequent images were calculated by Mentor Graphics FloEFD.

Next the engineer must define boundary conditions, that is the operating parameters and limits that will be used in the calculations. Values for the external air temperature and the LED device heating power must be specified, and can be incremented over multiple iterations of the CFD analysis.

Figure 6. displays the outcome of the CFD operation on the cross-sectional slice. It shows not only the heat distribution across the physical elements of the lamp, but also the vectors for air flow due to convection over the exterior of the lamp. In this example the 3D perspective has been restored for visual reference but the flow vectors pertain to the slice. In this view, the color spectrum goes from red (hottest) to blue (coolest), with gradations of orange and green between the two extremes.

Of course, the object of this exercise is to ensure that a proposed physical design will transport heat away from the LED source and conduct it safely toward the ambient environment. Figure 7. (overleaf), another Concurrent CFD view, provides an answer to this important question. In this view, weightless particles trace the air flow path, almost like infinitesimal dust motes. Here again the color spectrum reveals the heat distribution and in this image a color legend quantifies the values. Note the flow pattern: blue (cool) air comes up from below and is warmed to blue-green as it passes over the luminaire. Convection carries the warmed air up and away from the lamp. Is this sufficient dispersion for the lamp itself and any other housings that will be part of the final design? It is a question only the engineer can answer, but the Concurrent CFD analysis has provided the data necessary to support an informed judgment.

Figure 8. provides a little more detail. It is the “After” view of the LED and enclosure showing the heat gradients applied to the “Before” image shown earlier. Here again the legend offers quantitative details about the temperatures within the housing. This makes it easy to tell if, for example, touch temperatures are within safety limits.

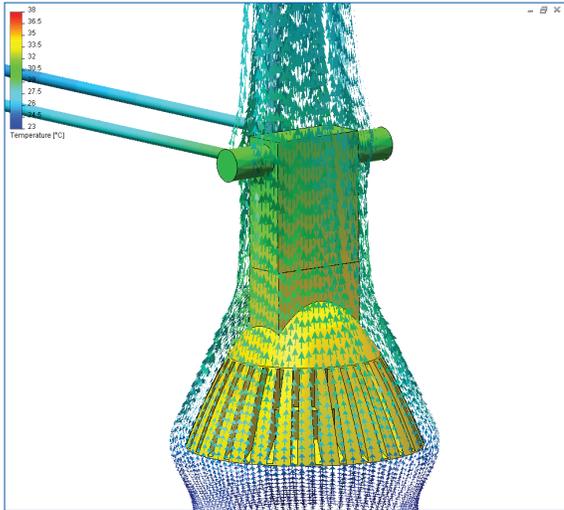


Figure 7: Particle tracks over the lamp, mounted horizontally and facing downwards, showing how the natural convection causes the air to flow smoothly over the exterior of the lamp housing. The thermal plume contracts as the flow accelerates above the lamp. (Note: Legend has been enlarged for legibility in this view.)

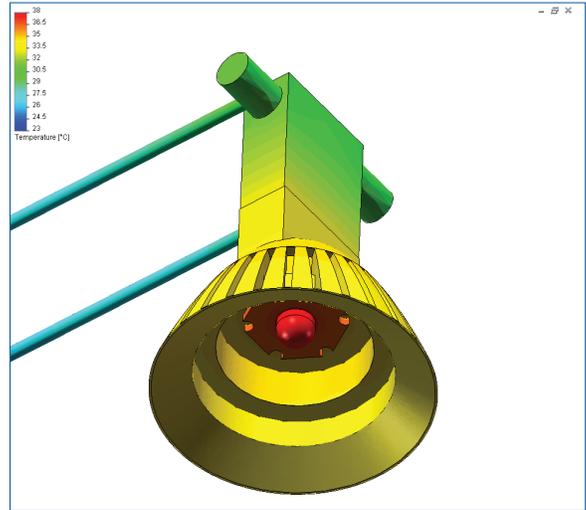


Figure 8: Predicted surface temperatures shown on the MCAD model

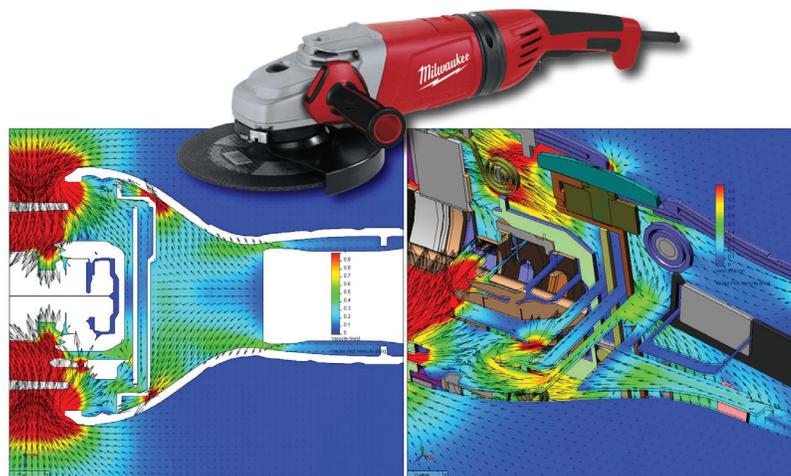
Using a Concurrent CFD tool such as FloEFD flow simulation and analysis to refine design proposals can prove indispensable. It is far less costly than building and testing a succession of physical prototypes, and the automation built into Concurrent CFD means that preparation for the first evaluation cycle is brief and for every subsequent attempt, even faster. It is an environment that encourages experimentation until the design is truly optimized. For example, Concurrent CFD can be used to quickly determine the optimum number slots in the bell-shaped housing and the thickness of the metal between them to maximize the heat loss to the surrounding air.

FLOW SIMULATION HELPS AEG DESIGNS COOL POWER TOOLS FASTER

According to the Freedonia Group, the world power tools market is estimated at about \$23.4 billion. The market is estimated to grow 4.1% annually through 2011. Professional users account for the majority of the world power tool market and AEG is well placed to continue capturing its lions' share of the market sector by being one of the suppliers of choice.

AEG optimizes their new tools from the start of the project and use computer-aided design and engineering software, including PTC's Pro/ENGINEER Wildfire for design alongside FloEFD for Pro/ENGINEER from Mentor Graphics for airflow and thermal analysis. Motor cooling problems are investigated in order to better understand the effectiveness of the cooling method used. Previously past experience or results obtained from physical prototype testing were used to draw conclusions. Using FloEFD allows AEG engineers to have a much better idea of how the products perform well in advance of the physical prototyping stage. On a recent project, they were able to reduce engine temperature by 15% more than previously thought possible, and it was done much faster than previous attempts.

AEG chose FloEFD because it was easy to use. Design engineer Markus Wörner said, "I found meshing and the addition of boundary conditions extremely easy to do." The FloEFD mesher can handle complex geometries with tight crevices and sharp angles easily, due to the adaptive shape of its cells. Also, the FloEFD user interface is geared towards the needs of an engineer as opposed to analysis specialists. Therefore, instead of technical jargon, FloEFD deals with terms such as walls, inlets or outlets. "In addition, with FloEFD we don't need to create the fluid area so the program is quite intuitive to use" added Wörner. Most traditional fluid flow simulation programs require users to create "phantom" solid parts to represent the (empty) fluid regions – an extremely time-consuming process since users need to identify each region manually and then create geometry to fill it. FloEFD saves users time and effort by automatically differentiating between solid and fluid regions for internal and external flows to create the fluid domain.



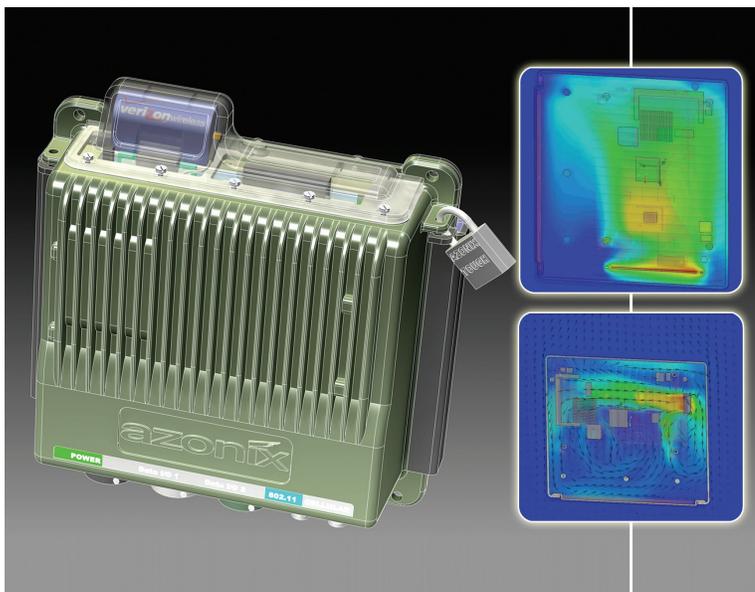
AEG Electric Tools uses FloEFD to optimize airflow and cooling effects in power tools

AEG uses FloEFD to optimize thermal issues. "With FloEFD we can investigate thermal tasks regardless of whether we are interested in the thermal behavior of a single component or the entire tool. Aside from dealing with engineering design issues, the AEG design team faces another challenge. AEG Electric Tools in Winnenden develops tools for Milwaukee Electric Tools and AEG Power Tools. Using a modular design for the machines and the platform strategy makes it possible to achieve the different needs of different users in a fast and efficient way without losing quality. Even small changes in components affect the airflow and temperature. Testing each configuration would take a lot of time. But with simulation, they can identify the effects caused by all the different design options and ensure proper performance for all machines.

AZONIX REDUCES THERMAL PROTOTYPES FROM TWELVE TO ONE

Azonix used FloEFD software to reduce the number of thermal prototypes required from up to 12 to 1 on its new Terra embedded computer. "FloEFD enables design engineers without a fluid analysis background to perform thermal simulation," said James Young, Design Engineer for Azonix. "The result is that we got the design right the first time, only had to make one prototype, and avoided expensive design changes that typically occur in the late stages of the development process."

Azonix is a division of Crane Co. and a leading provider of highly engineered computers and displays designed for extremely harsh environments. The Terra is a new embedded computer designed for use in the transportation industry that is, like other Azonix products, completely sealed from the elements and designed for use in very hot environments. "As with most of our products, we were limited to conduction and natural convection cooling," Young said. "This presents a difficult challenge for modern electronics equipment."



"We opened the SolidWorks model in FloEFD and defined the heat dissipation sources, material properties, and the ambient temperature outside the enclosure at the product's design limit of 60°C," said Young. "Then we defined the goals and performed a thermal simulation. The software analyzed the CAD model, automatically identified fluid and solid regions and allowed the entire flow space to be defined and gridded without user interaction and without adding extra objects to the CAD model. The software took about five hours to generate simulation results. The results showed, as expected, that temperatures on the surfaces of key components exceeded the allowable limit of 90°C."

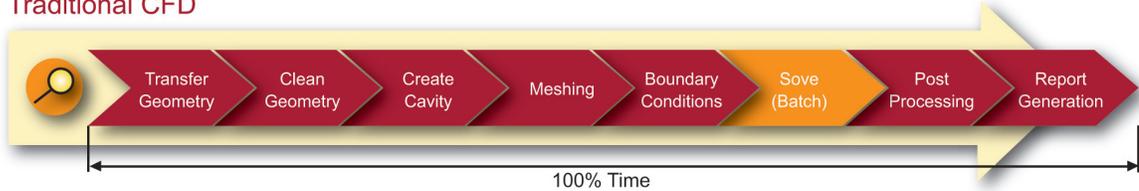
The conduction path from the heat dissipating components to the heat sink and heat sink geometry were the primary design parameters that were available to improve thermal performance. The cross-section of the heat spreader was increased and changed from aluminum to copper. Gap type thermal interface material was inserted at the interfaces between the components and the heat spreader. The thermal interface material was modeled as a contact resistance, reducing the number of cells, rather than conduction through material. "These changes substantially reduced the surface temperatures on the dissipating components but not enough to meet the thermal requirements," Young said. "Then we optimized the design of the heat sink." After about a half dozen different iterations, in each case changing the spacing and height of the fins, the heat sink was optimized and the internal component temperatures were minimized.

"The changes to the heat sink reduced the surface temperatures below the maximum allowable levels," Young said. "The result was that we were able to complete the thermal design prior to building the first prototype. When the prototype was built and tested, the measurements were within 5% of the simulation predictions. As a result, this was the only thermal prototype that needed to be built. This is a good example of how the new generation of embedded CAD tools can save money and time by enabling design engineers to optimize the design from a thermal standpoint early in the design process."

BENEFITS OF CONCURRENT CFD (CONCURRENT COMPUTATIONAL FLUID DYNAMICS)

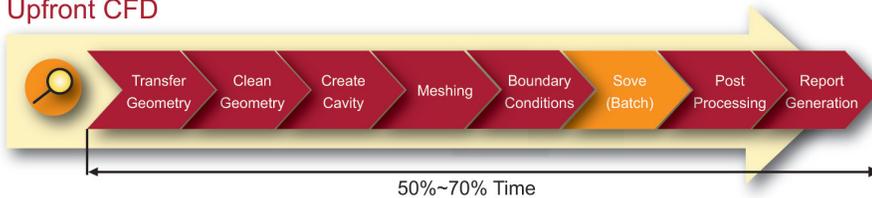
Concurrent Computational Fluid Dynamics (CFD) is a breakthrough technology that enables design engineers to conduct up-front, concurrent CFD analysis throughout the product's lifecycle, using the familiar MCAD interface, thus reducing design times by orders of magnitude compared to traditional methods 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.

Traditional CFD



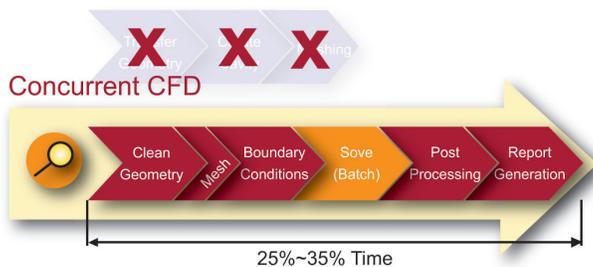
In **traditional CFD**, the model geometry is first exported from the CAD system. The geometry then needs to be re-imported into the user's CFD tool, meshed, solved, the results post-processed and finally reported back to the design team. The work is usually done by a specialist analysis group, or outsourced, 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

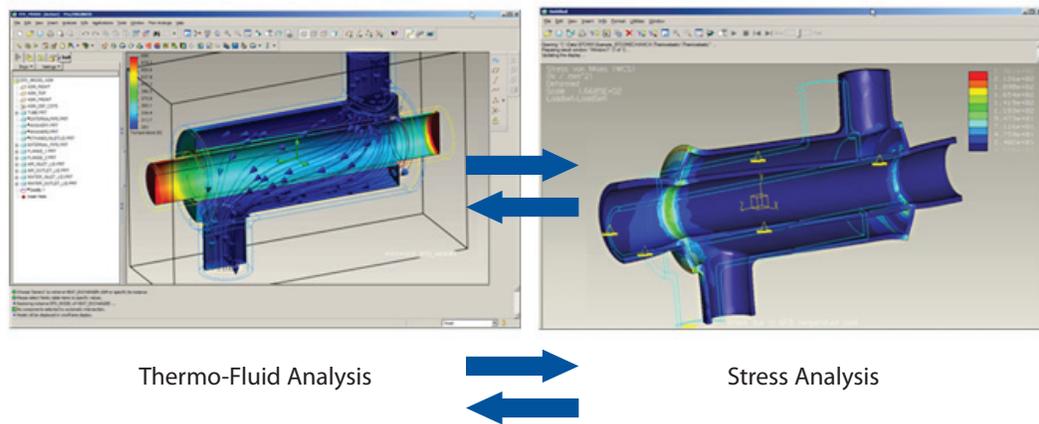


Upfront CFD attempts to improve this situation by streamlining the interface between the CAD and the CFD tool. Although this results in a much cleaner import of the geometry, the analysis is still performed outside of the CAD system. The frequent transfer from the CAD and CFD software can result in a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



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.



Interface with Pro/ENGINEER Mechanical -- use thermal and flow analysis results for structural analysis

For the latest product information, call us or visit: www.mentor.com

©2012 Mentor Graphics Corporation, all rights reserved. This document contains information that is proprietary to Mentor Graphics Corporation and may be duplicated in whole or in part by the original recipient for internal business purposes only, provided that this entire notice appears in all copies. In accepting this document, the recipient agrees to make every reasonable effort to prevent unauthorized use of this information. All trademarks mentioned in this document are the trademarks of their respective owners.

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

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



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