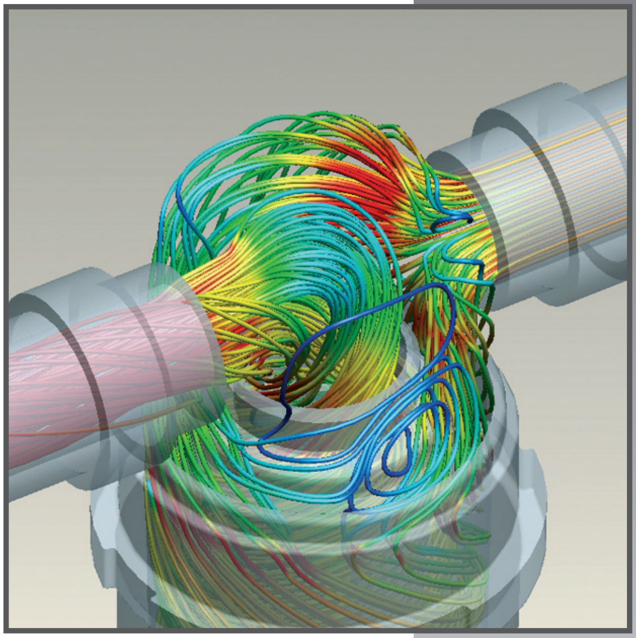
FloEFD™

Concurrent CFD for Creo Parametric



Mentor Graphics®

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

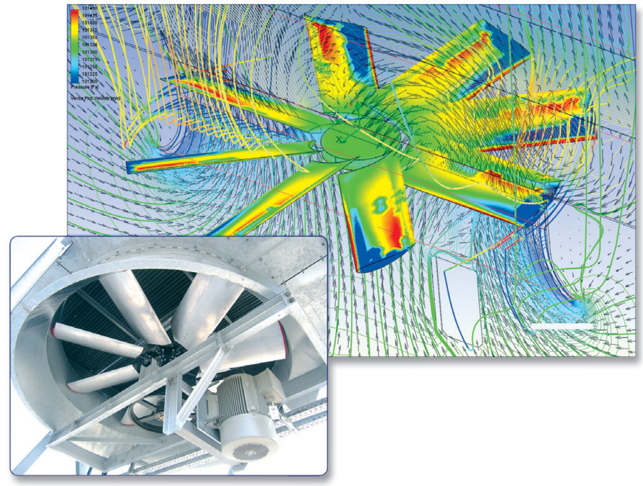
www.mentor.com

FloEFD for Creo Parametric is the only fully embedded Computational Fluid Dynamics (CFD) tool for Creo Parametric. As a Pro/TOOLKIT application, FloEFD enables you to analyze and optimize complex fluid flow and heat transfer effects on your designs directly inside Creo Parametric.

FloEFD has the same "look and feel" as Creo Parametric so you are not forced to learn a new interface simply to use the software. And unlike other CFD programs, FloEFD interacts directly with the native 3D CAD data defined by Creo Parametric - with no translation or copies - in order to keep pace with on-going design changes.

Concurrent CFD can reduce simulation time by as much as 65 to 75% in comparison to traditional CFD tools and enables users to optimize product performance and reliability while reducing physical prototyping and development costs without time or material penalties.

If you use Creo Parametric, you ought to take a closer look at FloEFD - the only fluid flow and heat transfer simulation tool that fits into your design process without requiring you to change the way you design products.



" FloEFD is a natural extension of traditional CFD that is easier to use and more intuitive for mechanical engineers. "

G. Bertels, Senior Engineer, Bronswerk Heat Transfer BV

ENGINEERS…

FloEFD for Creo Parametric was developed for engineers by engineers. So it works the way you do.

FloEFD is fully embedded into Creo Parametric and has the same look and feel as Creo Parametric. Since FloEFD interacts directly with native Creo Parametric data, you can create models and immediately analyze and improve them without having to deal with any file transfer or data loss.

FloEFD is also extremely easy to use – as a matter of fact, most users report that they can use FloEFD with less than 8 hours of training.

In short, FloEFD helps you get on with the business of improving product performance/functionality and reducing prototyping costs without requiring you to become a full-time fluid dynamics specialist.

CFD SPECIALISTS…

FloEFD for Creo Parametric co-exists quite easily alongside your traditional CFD programs and will increase your overall productivity.

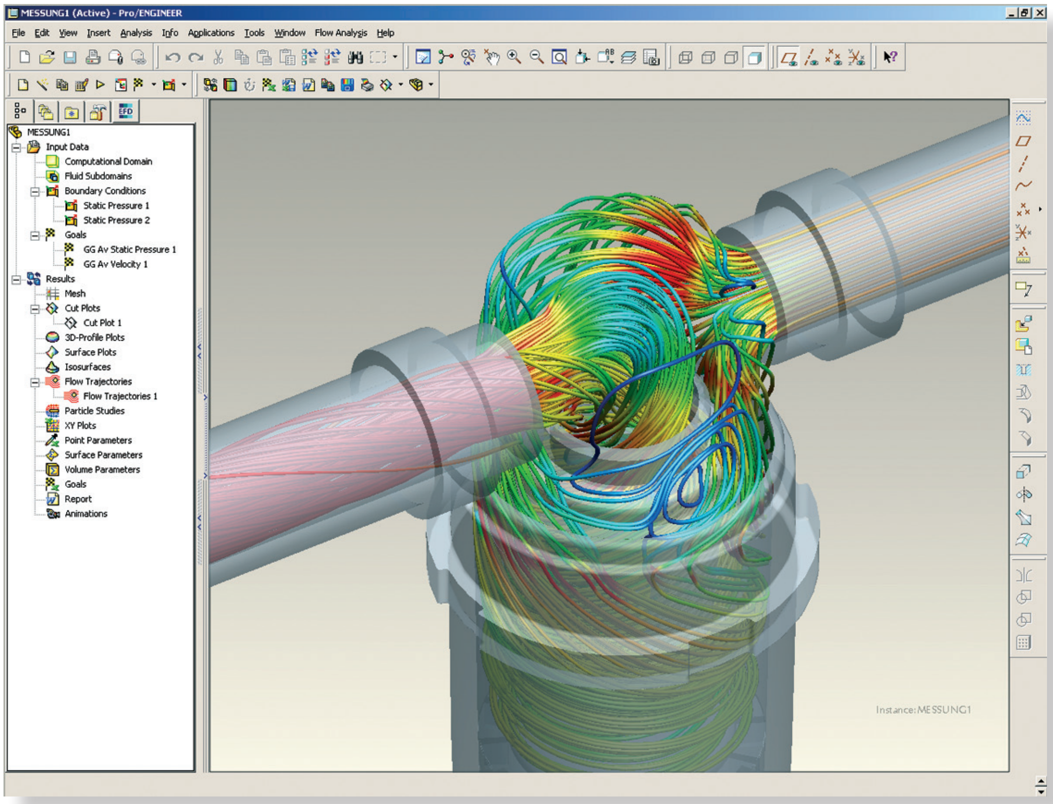
By enabling design engineers to conduct first-level CFD analyses directly from CAD models to remove unreasonable options with FloEFD, you'll be able to focus your time and energy on research and conceptual design. With FloEFD you can take advantage of advanced meshing technology which makes analysis of real-world designs even faster and more accurate.

Finally, as the resident analysis expert, you will be able to use your extensive knowledge to help guide the design engineering team at your organization.

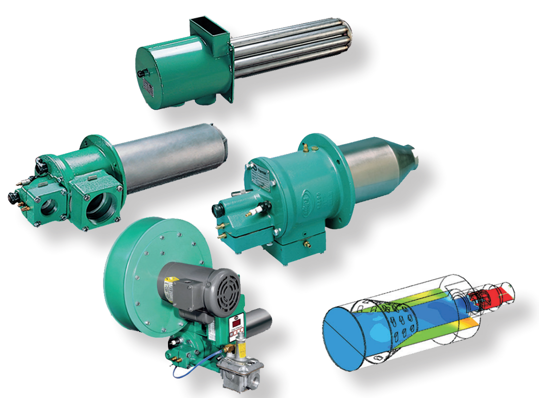
MANAGERS…

FloEFD for Creo Parametric supports your "lean engineering" efforts directly by reducing the burden on your budget and saving thousands of man-hours. FloEFD enables broad-based mechanical design engineers to perform fluid flow and heat transfer simulations directly from their 3D CAD models, in a fraction of the time taken by other CFD codes, and with very little training:

- Reduce prototyping costs drastically by replacing physical tests with virtual tests
- Increase product quality while reducing production costs by helping your team reduce errors and create better products
- Shorten the development cycle by enabling your team to conduct "what-if" tests quickly

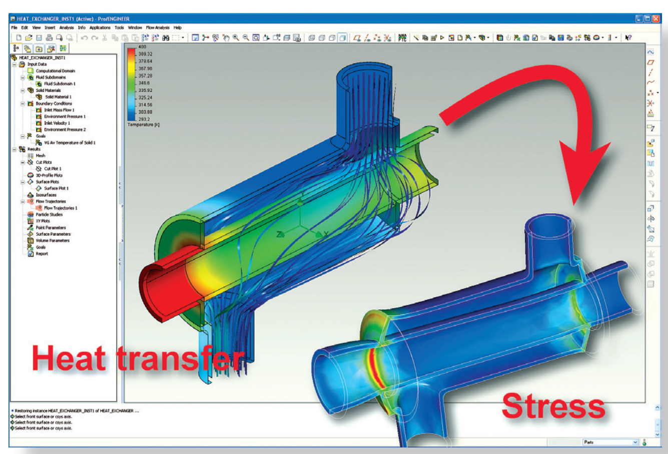


FloEFD for Creo Parametric is the only fully embedded CFD tool for Creo Parametric - effectively fluid flow and heat transfer analysis becomes an extension of Creo Parametric capabilities.



" FloEFD results are easy to understand and interpret by any engineer who is already familiar with the subject- matter being studied. "

A. Heijmans, Development Manager, Eclipse Combustion

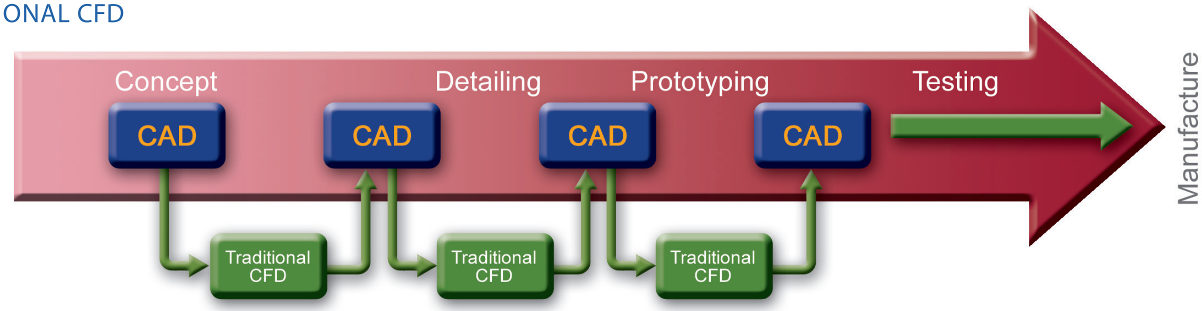


Interface to Pro/MECHANICA

FloEFD for Creo Parametric results can be applied as loads for structural analysis inside Pro/Mechanica with EFD2Mech. The automated interface significantly reduces the amount of time necessary for preparing analysis models and enables you to calculate structural damages due to thermal loads.

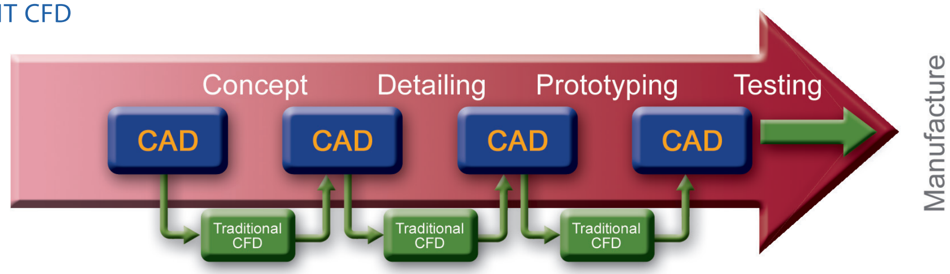
Concurrent Computational Fluid Dynamics (CFD) is a breakthrough technology that enables design engineers to conduct upfront, concurrent CFD analysis throughout the product's lifecycle, using the familiar MCAD interface, thus reducing design times by orders of magnitude compared to traditional methods 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 and reliability while reducing physical prototyping and development costs without time or material penalties.

TRADITIONAL CFD



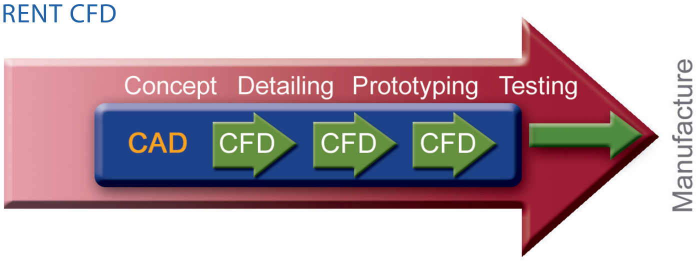
In traditional CFD, the model geometry is first exported from the CAD system. The geometry then needs to be re-imported into the user's CFD tool, meshed, solved, the results post-processed and then reported back to the design team. The work is usually done by a specialist analysis group, or outsourced,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

UPFRONT CFD



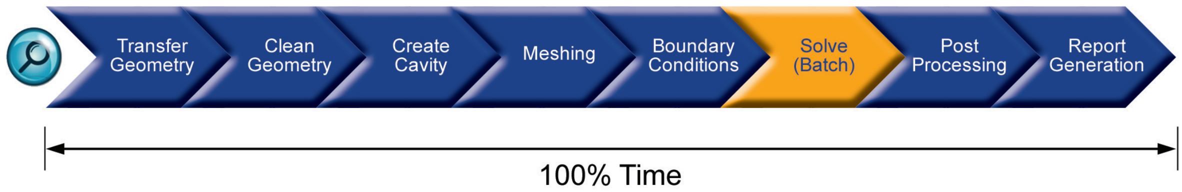
Upfront CFD attempts to improve this situation by streamlining the interface between the CAD and the CFD tool. Although this results in a much cleaner import of the geometry, the analysis is still performed outside of the CAD system. This frequent transfer from the CAD and CFD software can result in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

CONCURRENT CFD

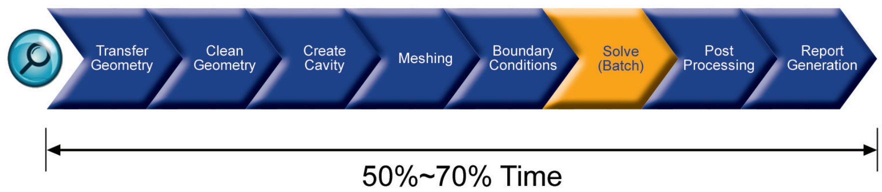


Concurrent CFD operates very differently. It is MCAD-embedded CFD so the work is done within the MCAD environment. Embedding CFD inside an MCAD tool is hard to accomplish, but delivers very significant benefits. Design changes necessary to achieve the desired product performance are made directly on the CAD model, so the design is always up-to-date with the analysis.

TRADITIONAL CFD



UPFRONT CFD

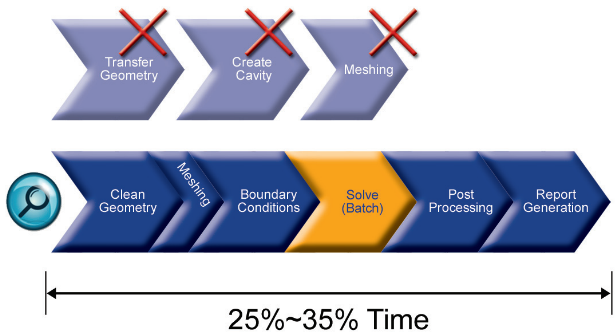


By looking at the CFD process in more detail, a number of steps are revealed. Within both conventional and upfront CFD it is necessary to transfer geometry from the CAD system and clean it before it is suitable for analysis. This process has to be repeated as design changes are made in the analysis suite and taken back into the CAD system to keep the two synchronized.

Typically this approach will require fluid spaces to be watertight for the analysis. In CAD terms this is referred to as 'healing' the geometry to make it 'manifold', whereas analysts often refer to it as 'cleaning the CAD model'. This is a generic requirement for CFD analysis, so it appears in all three approaches.

Also, these systems require creation of a 'cavity' to represent the flow space. Most conventional CFD meshing tools work by meshing a solid, so they require a solid object to mesh. For a CFD simulation the solid object is the flow space, which has to be created as a dummy part within the CAD system by Boolean subtraction of the entire model from an encapsulating solid. This is usually done in the CAD system and it is this inverted flow space that is transferred to the CFD system for meshing.

CONCURRENT CFD



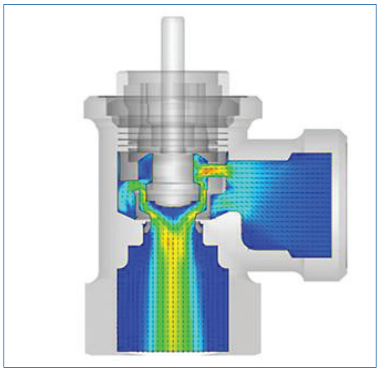
By comparison, Concurrent CFD works differently. The geometry used for the analysis is native to the CAD system. This means that geometry does not need to be transferred because the designer never has to leave the CAD system

Concurrent CFD therefore does not require the "transfer geometry" or "create cavity" steps. Meshing still takes place, but takes just minutes rather than hours of iterating back and forth.

Concurrent CFD provides another benefit that is not shown here. As mechanical designers undertake their own analyses, they quickly learn how to build analysis-friendly geometry within the CAD tool, eliminating the "clean geometry" step, so the time savings can be even greater than those indicated.

PRESSURE DROP

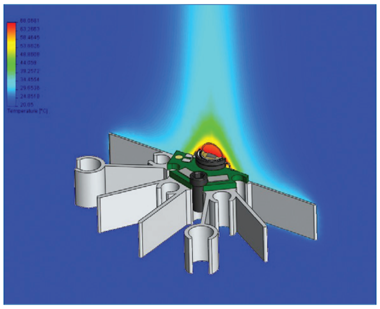
Investigate and optimize pressure and a range of pressure-related parameters in a wide variety of products such as valves, nozzles, filters and control devices in real-life operating scenarios.



A valve showing regions of high and low pressure due to restricted/changing fluid flow

HEAT TRANSFER

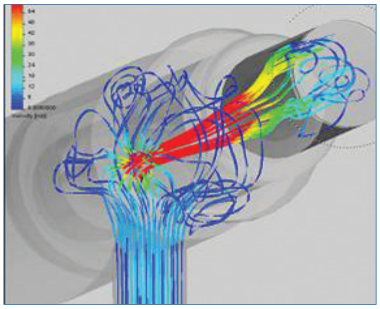
Visualize and understand temperature fields in and around practically anything including ovens, heat exchangers and drilling heads. Analyze the complex physical processes such as heat conduction, heat convection, conjugated heat transfer between fluids, surrounding solid materials as well as radiation among many others.



Analysis output showing rising thermal plume from a single high power LED unit

MIXING PROCESSES

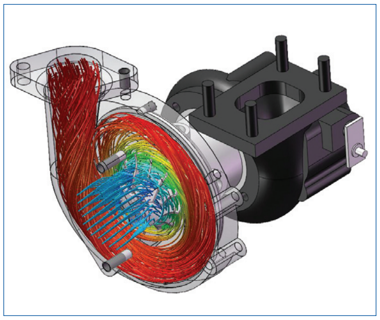
Explore and visualize mixing of fluids and gases to determine the optimum mix inside a wide range of products such as washing machines, dishwashers, kitchen and bathroom appliances and even fuel cells.



FloEFD is able to visualize how fluids or gases will react when mixed

FLOW FIELDS

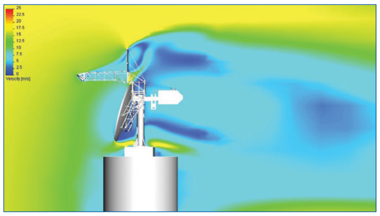
Inspect and optimize complex flows in and around objects. Optimize how gases and liquids interact with and inside smoke detectors, cyclones, cleanrooms and air-handling devices.



FloEFD can show the visual results of 'Flow Field' analysis in several formats; shown here as stream lines that can be tracked from point of origin to point of exit

FORCE PREDICTION

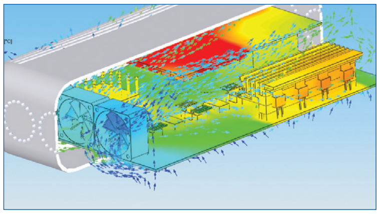
Examine operational forces for valves, flow-induced loads for stress and deformation analysis temperature distribution for thermal stress analysis.



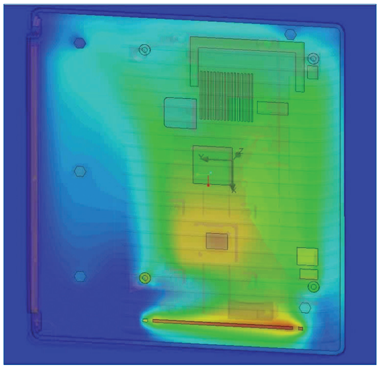
Wind force on a radar tower; an engineer can visualize wind being deflected by the dish and resulting turbulent areas behind the tower

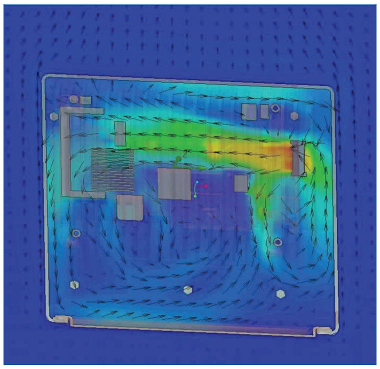
ELECTRONICS COOLING

Early analysis and optimization with FloEFD before a product leaves the 'drawing board' helps companies to design better, more reliable products faster by overcoming everyday design engineer challenges. Challenges such as PCB thermal design, heatsink design, package junction temperatures, case temperatures and airflow optimization enables these companies to get their products to market faster.



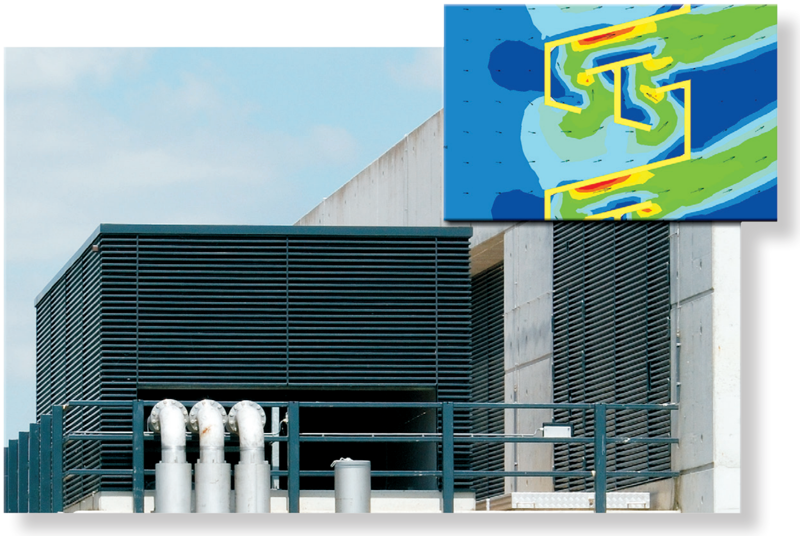
Vector plots showing air flow inside an electronics box





" FloEFD computational fluid dynamics (CFD) software enables design engineers without a fluid analysis background to perform thermal simulation. 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. "

James Young, Design Engineer, Azonix



" The combination of Creo Parametric for solid modeling and FloEFD for airflow analysis allows us to test our louvers for form, fit and function effortlessly. We can show the finished design to our customer complete with how it will look and work in just one day – that's a savings of 3 weeks and thousands of Euros for each model. "

H. Aaldering, Technical Director, JAZO Zevenaar B.V.

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

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 05-11 1028820-w