



— Mechanical Analysis Division

Web Seminar: - CFD Analysis for MCAD Designers - Better Products, Faster

Event Questions & Answers...

Q: If I run a set of design variations through the analysis, how do I manage the design data and results to determine the best one?

A: FloEFD supports the design parameter and configuration management system of the CAD system. All EFD data are managed the same way as the CAD data. FloEFD is also compatible with major Product Data Management (PDM) systems.

Q: Our products are very complicated. Can FloEFD mesh them and provide accurate answers?

A: Yes. FloEFD includes meshing technology that is able to process complex real-life designs. It uses hexahedral mesh cells in the computational domain. This cell shape is the most efficient one regarding computational efforts necessary to obtain required accuracy. In addition, it is very robust and enables fast solution times.

Q: What MCAD systems besides Pro Engineer does FloEFD work with?

A: We offer CAD-embedded solutions for Pro/ENGINEER Wildfire, CATIA V5, and SolidWorks (exclusively sold by SolidWorks Corporation). Additionally, FloEFD reads native CAD data directly from major CAD systems such as Siemens NX, SolidEdge, Autodesk Inventor, etc. without the need for prior data conversion.

Q: What has changed in the industry over the past few years? We have always had design time pressures.

A: Time, cost and quality continue to be pressures. However, the evolution to a global market and competition has intensified the need to get better products to market faster. With more competition, companies must respond faster in order to maintain their market share and brand loyalty. Compressing the development process is also a factor in reduction of the overall cost of a product.

Q: I am what you referred to as the CFD Specialist for my company. Do you believe a designer is capable of running CFD software?

A: Yes. Over the past 10 years we have had many companies successfully integrate FloEFD directly into their early design process. FloEFD was specifically developed with the needs of the CAD designer in mind and enables design engineers who are subject-matter experts to conduct analysis on everyday design tasks, this in turn releases the CFD specialist to concentrate on their custom code projects.

Q: My company's current CFD product is a nightmare to get to converge. What's so different about FloEFD?

A: FloEFD comes with a unique technology to ensure extremely robust solver convergence without compromising on results quality by introducing numerical diffusion or similar effects. This is a fully automatic technology which does not require specific numerical expertise to tweak solver parameters.

Q: Do you have any references from users who are using FloEFD?

A: Please check out a selection of success stories on our web site.
<http://www.mentor.com/products/mechanical/success/>

Q: Do you have any research papers that discuss what trends are happening related to the PLM industry?

A: CIMdata has several papers on its website that discuss PLM industry trends. We have just released a paper on CFD that is available on both our site and on Mentor Graphic's site.
<http://www.mentor.com/products/mechanical/techpubs/> Other papers look at ECAD-MCAD integration, the evolution to unified architectures, PLM and Automation (shop floor integration), and more.

Q: How long has FloEFD been a product?

A: FloEFD is well established and has been used by many companies during the past 10 years.

Q: My company has never done any CFD. How do I start?

A: Mentor Graphics has a wealth of experience in helping new users to implement FloEFD as an efficient tool integrated in the product development process. Besides software training, and excellent technical support, our staff will advise you how to introduce FloEFD into your development process to maximize the benefits of using this design evaluation tool on a day-to-day basis. As a first step, we recommend that you download the 60-day trial version of FloEFD from our website so you can test the technology first-hand.

Q: Can FloEFD identify regions of low or zero velocity within a valve?

A: Yes, please join us at the next web seminar 'Simulating and Optimizing Pressure Drop' to see a more detailed presentation on this subject.
<http://www.mentor.com/products/mechanical/events/measuring-optimizing-pressure-drop-webseminar>

Q: What parameters are used to describe the fluid?

A: We support gases, Newtonian and non-Newtonian liquids and steam (two-phase). Depending on the fluid type, different parameters may be required to characterize the fluid, e.g. density, viscosity, thermal properties, etc.

Q: What is the cost of single license (used with Pro/E WF4.0)?

A: Please contact your local sales representative.

Q: Are batch analyses applied through a family table environment or can batch analysis be run on similar but separate assemblies?

A: Yes, batch analyses are applied through the family table concept.

Q: What technology does this use? Finite Element or Finite Difference? Is it 3D or 2D with simulated 3D?

A: FloEFD uses fully 3D Finite Volume Method.

Q: Can FloEFD deal with phase transfer? What about supersonic flow - shock waves and other complex flow fields?

A: FloEFD can handle cavitation in liquids using a homogeneous equilibrium model, and has specific models for water and steam, including volume condensation.

A: Supersonic flows: Yes, including transonic and hypersonic flows with related physical effects

Q: What versions of the CAD tools do current EFD products support?

A: FloEFD is fully embedded into major MCAD systems including Pro/ENGINEER, SolidWorks, CATIA V5, and tightly integrated with Inventor, Siemens NX and Solid Edge.

Q: When is it possible to calculate a mix of water and air?

A: We are continuously working to enhance our products. However, we are unable to talk about detailed development plans at this point.

Q: There was a couple of references to combustion. What capabilities do you have?

A: The current release doesn't feature a combustion model. We are continuously working to enhance our products. However, we are unable to talk about detailed development plans at this point.

Q: Is the analysis data saved in the model?

A: All project setup data is saved within the model file and the results data is saved as separate files.

Q: Can FloEFD mix two fluids?

A: Yes, FloEFD can simulate mixing of up to 10 fluids of the same phase.

Q: Does FloEFD plan to include absorption media in a surface-to-surface radiation model. This will open a new area of application: Ultraviolet water treatment

A: We are continuously working to enhance our products. However, we are unable to talk about detailed development plans at this point.

Q: Since the tool has automatic meshing and the meshing is taken out of the specialist, how can the accuracy be assured? Does the software perform grid study analysis?

A: The meshing technology bases on an innovative automatic meshing algorithm involving local refinements (adaptation) in multiple steps based on both, critical geometry and flow field parameters (solution). This ensures high result quality for a given setup. However, all meshing parameters are transparent to the user and customizable, so that also manual mesh studies can be performed.

Q: Comment and Q: CFD has grown today to CFD Design Optimization. That means automatically updating the geometry (or the mesh) in meeting performance goals. This requires that CAD-CFD be able to interact in batch process until the optimal geometry is found by multi-objective optimization procedure. It also requires clusters or thousands CPUs. Many companies in the EU have access to such clusters but unfortunately, very few in the USA can afford it.

Q: Can Flow Simulation of Solid Works be run in batch procedure by reading different geometries and running them for CFD? - If not, how about the FloEFD. We are performing a benchmark with Canaero in Brussels on CFD Design Optimization of pumps and need a CFD software which can be run in batch mode as described above.

A: Flow Simulation and FloEFD come with a full-featured API to be driven from external applications such as optimization software. Additionally, both products also feature an easy-to-use GUI for batch runs of preconfigured projects based on geometry and/or flow variations.

Q: One of the main issues of Flow Simulation is that the mesh cannot be read in some cases of cloning the model required to run the same geometry at different flow rates.

A: Please contact your local support engineer to discuss this issue in more detail.

Q: Can you share the cost of the software?

A: Please contact your local sales representative.

Q: Is Flow Simulation the same CFD software as FloEFD or FloEFD has more capabilities in terms of physical models, turbulent models and numeric options?

A: Flow Simulation is an analysis product exclusively designed for SolidWorks. FloEFD is a stand-alone CFD product designed to read native CAD model data from all major CAD systems. The physical modeling capabilities of both products are different. Please contact your local sales representative for details.

Q: Do you have a student version?

A: Not at this point.

More Information:-

EFD Solutions Area: <http://www.mentor.com/products/mechanical/fluid-dynamics/>

FloEFD Overview Animation: <http://www.mentor.com/go/floefd>

FloEFD Product Page: <http://www.mentor.com/products/mechanical/products/floefd/>