

FloEFD™

Engineering Fluid Dynamics

The Next Generation CFD

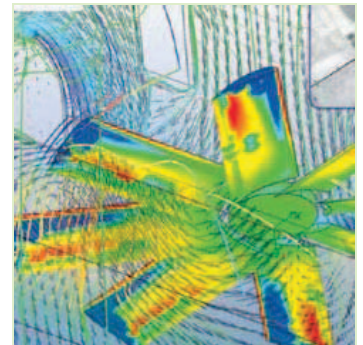
FloEFD is part of a new breed of CFD tools called Engineering Fluid Dynamics (EFD). FloEFD is a full-featured general purpose CFD tool serving the needs of Inventor, NX, Solid Edge, SolidWorks as well as other popular MCAD systems. FloEFD combines all simulation steps - starting with use of your 3D CAD data for model preparation, to mesh generation, solving and results visualization - in one easy-to-use package.

FloEFD is based on the same mathematical foundation as traditional CFD software, however, our 7 key technologies are what set FloEFD apart.

These technologies make FloEFD quicker and easier to use and more accurate.

Combined with your MCAD software, FloEFD provides you with a powerful simulation tool to:

- Improve product performance, functionality and reliability
- Reduce physical prototyping and production costs
- Minimize risk of making design mistakes



Engineers...

FloEFD was developed for engineers by engineers. Therefore, it is extremely easy to use – as a matter of fact, most users report that they can use FloEFD with less than 8 hours of training.

Since FloEFD lets you take advantage of your solid models for analysis, you'll be able to save a massive amount of time and effort. In short, FloEFD helps you to 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 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 our advanced meshing technology which makes analysis of real-world problems even faster and more accurate.

Also, 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 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

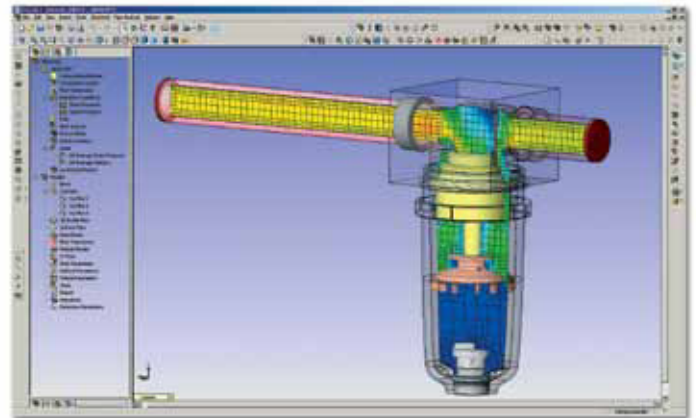
7 Key Technologies That Set FloEFD Apart

FloEFD takes away the complexity of mathematical schemes and methodologies by using an advanced set of knowledge-based automated processes. As a result, you can focus on your engineering tasks instead of worrying about identifying the most appropriate CFD algorithm or mesh. The following key technologies set FloEFD apart from all other CFD codes:

Direct CAD-to-CFD Concept

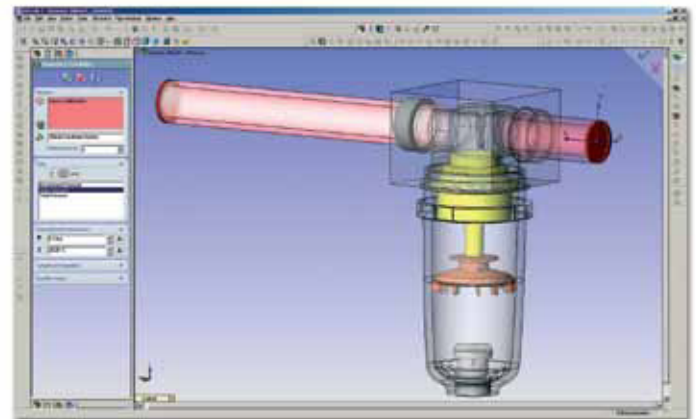
FloEFD is based on CAD. Offering a fully-integrated solid modeler, with FloEFD you can create your geometry or, use existing 3D CAD geometry and solid model information including features and parameters to help you simulate your designs in real-world conditions. Either way, you will find FloEFD extremely intuitive and easy-to-use as it offers functionality not available in other CFD programs.

For example, most CFD programs require you to create phantom objects to represent the fluid (empty) regions - an extremely time-consuming process since you need to identify each region manually and then create geometry to fill it. FloEFD automatically differentiates between the solid and fluid regions and automatically creates the fluid domain. As a result, FloEFD frees you from the hassle of creating unnecessary geometry just to prepare a solid model for analysis.



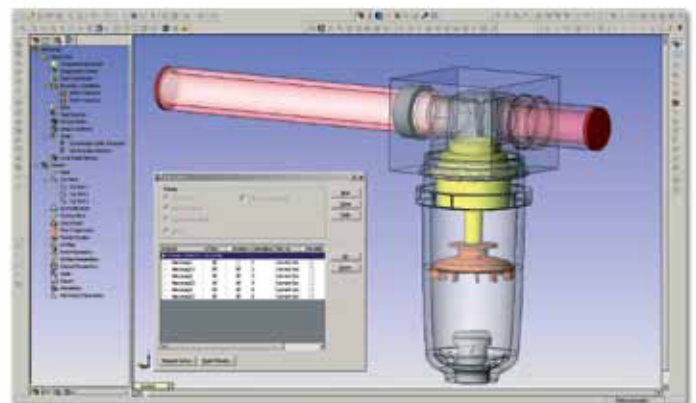
Engineering vs. Analysis User Interface

FloEFD features an extremely easy-to-use user interface. Instead of dealing with technical jargon such as y^+ value and Dirichlet Boundary Conditions, you deal with terms such as walls, inlets or outlets. In addition, FloEFD features a powerful analysis wizard that guides you through the set-up process. Therefore, you can focus on solving your problems as opposed to figuring how to use the software. As a matter of fact, FloEFD is so intuitive that most engineers report that they can use the software with less than eight hours of training.



Design Variant Analysis

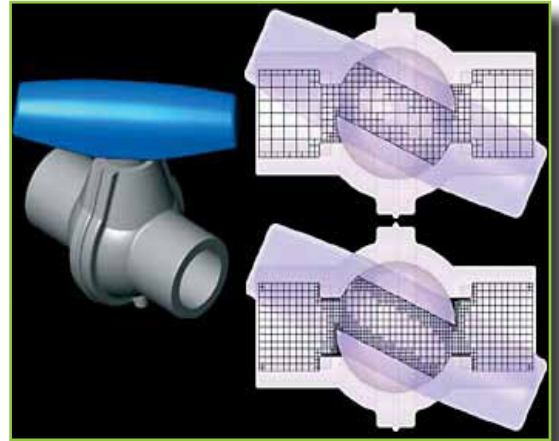
Perhaps the most powerful feature of FloEFD is the ease with which you can conduct "what-if" analyses. FloEFD makes it easy to modify your models and then immediately analyze them. The process is quite simple: FloEFD helps you create multiple variations of your designs by modifying your solid model and without having to reapply loads, boundary conditions, material properties etc., you can analyze them. Simply compare the results among the many design options to choose the best possible design. It really is that easy.



Intelligent Rectangular Adaptive Mesh

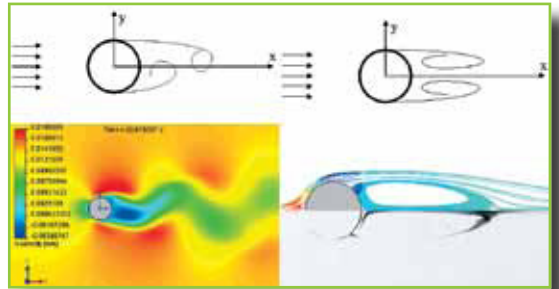
While some vendors offer only a black-box approach, the meshing process with FloEFD is completely transparent and you retain as much or as little control as you need.

As every CFD user knows, finding an optimum mesh can be extremely time-consuming. Mentor Graphics has developed a robust automatic mesher that can mesh real-world geometry – not just academic or theoretical ones. FloEFD features automatic meshing for fluid and solid regions including automatic mesh refinement/unrefinement due to geometrical and/or physical (solution adaptive) requirements. With FloEFD, your model will be meshed in the shortest time possible and with a high level of accuracy.



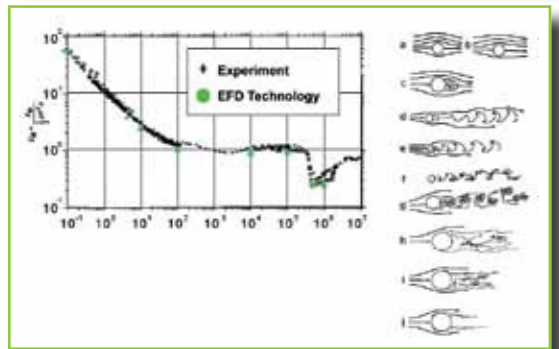
Modified Wall Functions

FloEFD features grid-independent near-wall modeling using Partial Cell technology. This technology enables you to simulate the boundary layer phenomena for fluid flow and heat transfer effects without needing a specifically designed mesh or a lot of time to solve the problem.



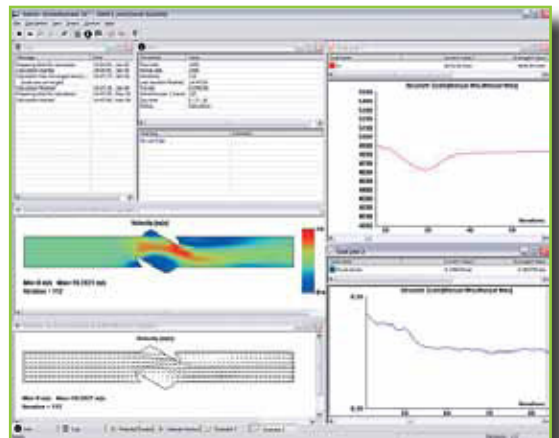
Laminar-Transitional-Turbulent Modeling

FloEFD does not require specific user input in order to simulate transitional flow characteristics since it supports laminar, transitional and turbulent flow regimes ... all automatically and within the same model. Therefore you don't need to worry about identifying when and where the flow characteristics change within your model. FloEFD will automatically do it for you.



Automatic Convergence Control

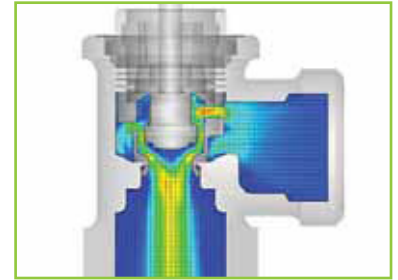
There is nothing more aggravating than setting up an analysis session and returning hours later to find that the software has failed to provide a solution due to numerical convergence problems. With FloEFD, you get real-time feedback about the solution progress such as the convergence of the solution, residual errors, monitor point values etc. Cutting-edge numerical methods with multi-grid schemes and robust convergence behavior in FloEFD ensure that you get engineering results in a timely manner.



Engineering Challenges

Pressure Drop

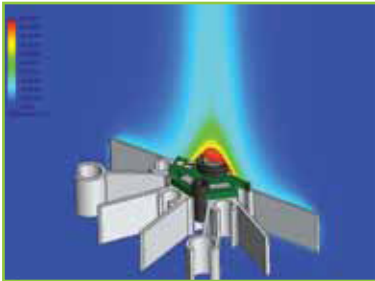
Investigate and optimize pressure and a range of pressure-related parameters in a wide variety of products such as valves, nozzles, filters and control devices in real-life operating scenarios.



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

Heat Transfer

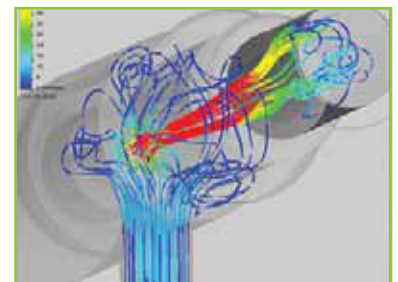
Visualize and understand temperature fields in and around practically anything including ovens, heat exchangers and drilling heads. Analyze the complex physical processes such as heat conduction, heat convection, conjugated heat transfer between fluids, surrounding solid materials as well as radiation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

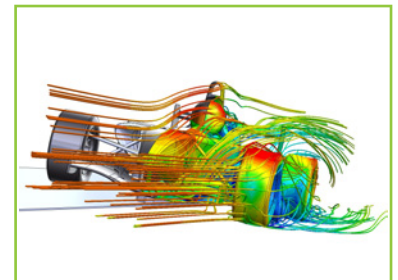
Inspect and optimize complex flows in objects. Optimize how gases and liquids interact with and inside smoke detectors, cyclones, cleanrooms and air-handling devices.



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

Flow Fields around objects

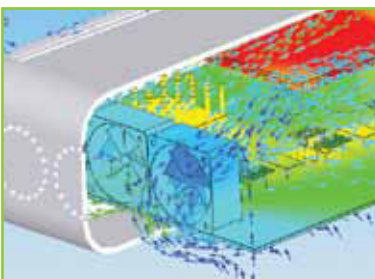
Examine and optimize complex flows around objects. Optimize how gases and liquids interact with vehicles, buildings, aeroplanes, et cetera.



Another 'Flow Field' analysis; shown here as stream lines around a Formula 1 car

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



Mechanical Analysis

Please visit our website at:
www.mentor.com/mechanical

Copyright © 2009 Mentor Graphics Corp
Mentor Graphics is a registered trademark of Mentor Graphics Corporation.
All other trademarks mentioned in this document are trademarks of their respective owners

For more information, please contact:

Holland Engineering Consultants bv

sales@hec bv.nl