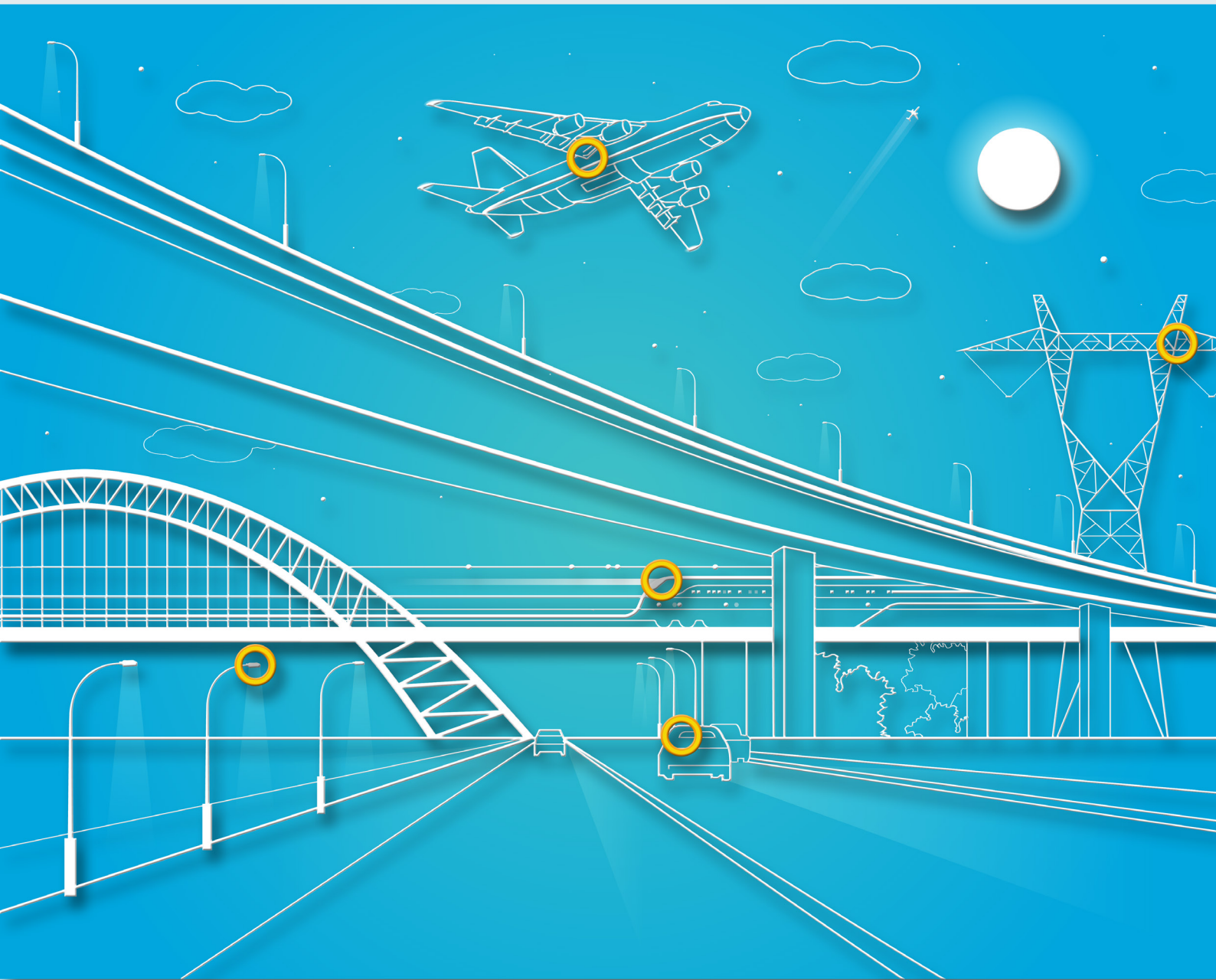


# FloEFD™

FloEFD™ for Creo® | FloEFD™ for CATIA V5 | FloEFD™ for NX® | FloEFD™ for Solid Edge®

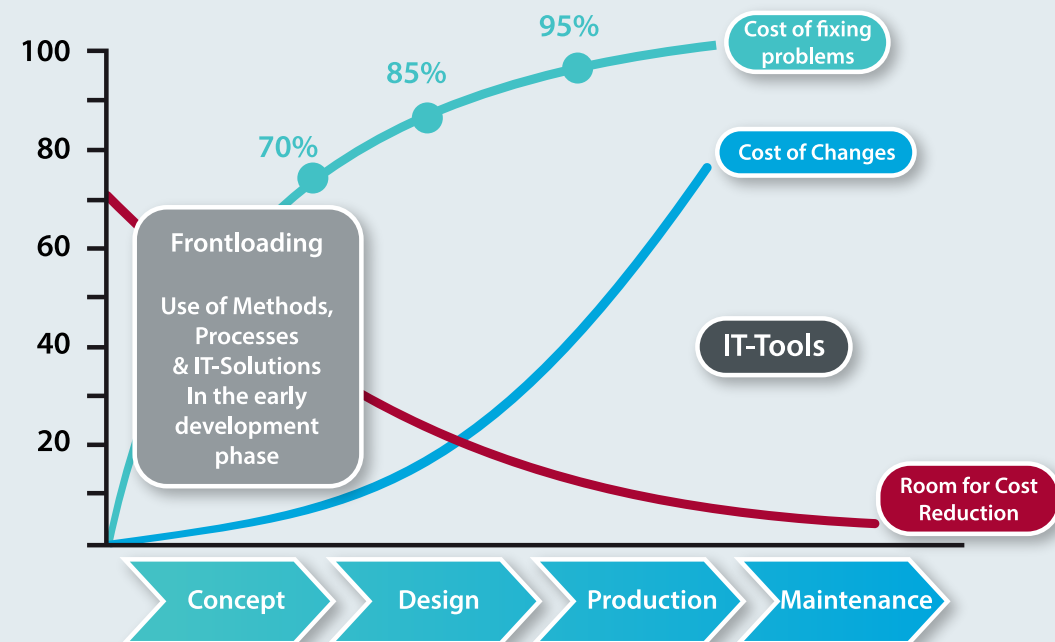


***Design it. Simulate it. Build it.***  
***Frontloading CFD with confidence using FloEFD.***



**Mentor®**  
A Siemens Business





FloEFD is an award-winning frontloading computational fluid dynamics (CFD) simulation solution for the design engineer targeting complex design challenges. Frontloading CFD refers to the practice of moving CFD simulation early into the design process, enabling design engineers to evaluate design options and optimize product performance as well as reliability.

FloEFD is a paradigm shift in CFD. Through the use of best in class and intelligent technology, FloEFD opens the world of fluid dynamics analysis to a wider range of users including design engineers. It helps engineers move simulation into the design process where CFD can greatly benefit the user to understand the behavior of the design earlier and discount the less attractive options.

In a world where many vendors purport to offer you the same functionality, let's take a look at what makes FloEFD the best option for you:

## FloEFD is a Design Simulation Tool

With a 25 year history of development expertise and unprecedented knowhow in embedding CFD into MCAD, Mentor, a Siemens Business, offers a design-centric CFD solution to the engineering community. As an embedded application, FloEFD is completely integrated into the most popular CAD programs available on the market. FloEFD plugs effortlessly into familiar PLM software, delivering compressed timescales for product development workflows in the creation and testing of the digital twin.

In addition, FloEFD is easy-to-use and offers an intuitive user experience for MCAD users. FloEFD uses engineering terms instead of technical jargon so you can focus on solving flow problems as opposed to figuring out how to use the software. And its powerful analysis wizard guides you through problem set-up, analysis and results visualization. Most users have reported that they can start using the software with only 8 hours of training. Lastly, it offers the widest range of local language implementations to make analysis as accessible as possible.

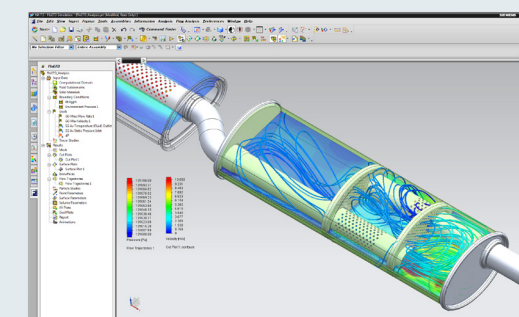


## FloEFD Frontloading CFD Award

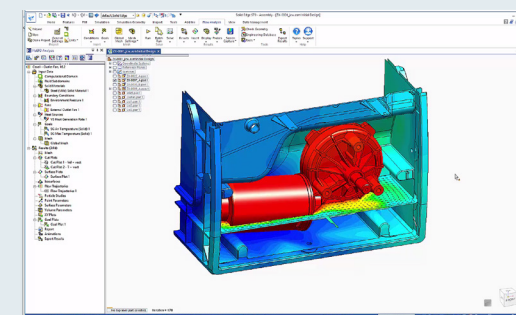
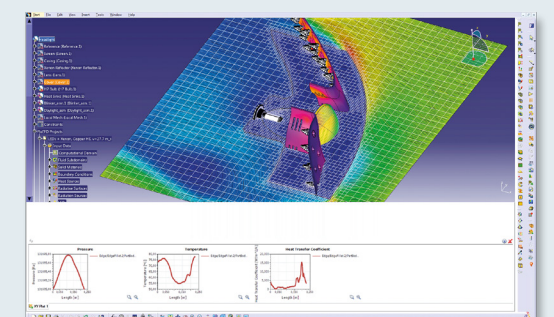
Frontloading CFD refers to the practice of moving CFD simulation early into the design process where it can help design engineers examine trends and dismiss less desirable design options. The Mentor Frontloading CFD Award recognizes excellence in implementing frontloading CFD through award-winning FloEFD.

<http://go.mentor.com/4QZWL>

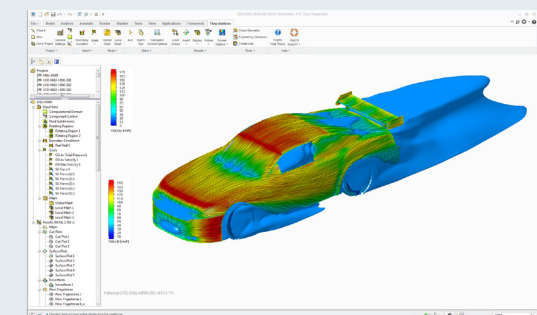
FloEFD for Siemens NX®



FloEFD for CATIA V5



FloEFD for Solid Edge®



FloEFD for PTC Creo®



FloEFD is the only CFD software with user interfaces in Japanese, Chinese, Korean, French, German and Russian.



## FloEFD has “Intelligent Automation” at its Core

The physics behind fluid flow are perhaps some of the most complicated disciplines in the field. As a result, CFD software is notoriously difficult to use. These programs require intimate knowledge of turbulence models and their appropriateness for specific situations. They also require users to manually tweak the mesh to get the most accurate analysis results – some specialists have been known to spend weeks if not months on optimizing the mesh. This is why a great majority of CFD software programs are unsuitable for use by design engineers who need access to physics-based simulation software.

Based on Navier-Stokes equations, FloEFD adds intelligent automation and technology to help make analysis easier, faster and accurate. Its unique SmartCells™ technology allows you to use a coarse mesh without sacrificing accuracy. The meshing technology is also robust and can easily capture arbitrary and complex geometry. As a result, the meshing process can be completely automated and requires less manual user input. Lastly, with FloEFD you can use your CAD geometry directly – without any translation or modification. In short, the intelligent technology helps you harness the power of fluid flow simulation.

## FloEFD is Accurate

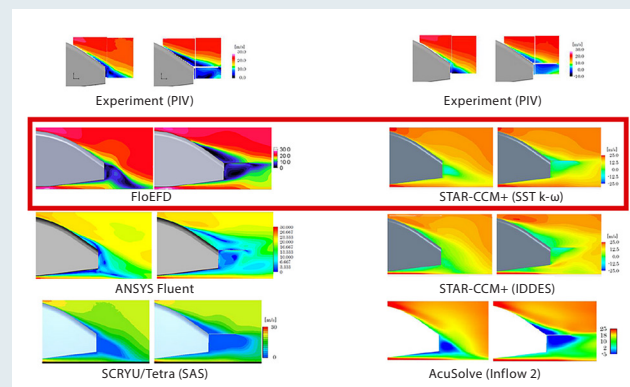
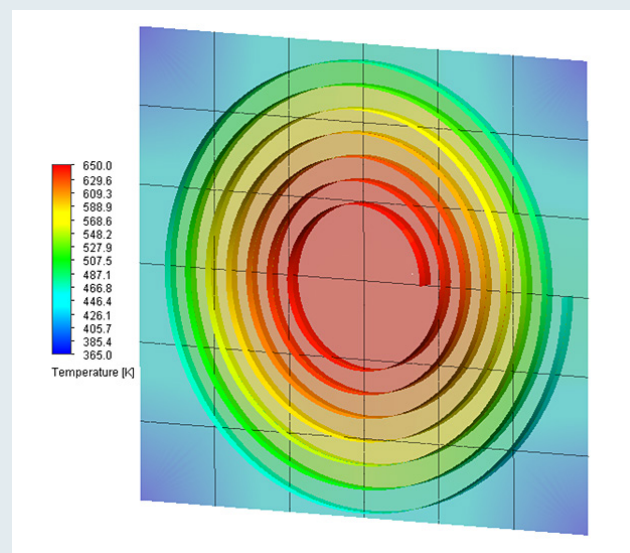
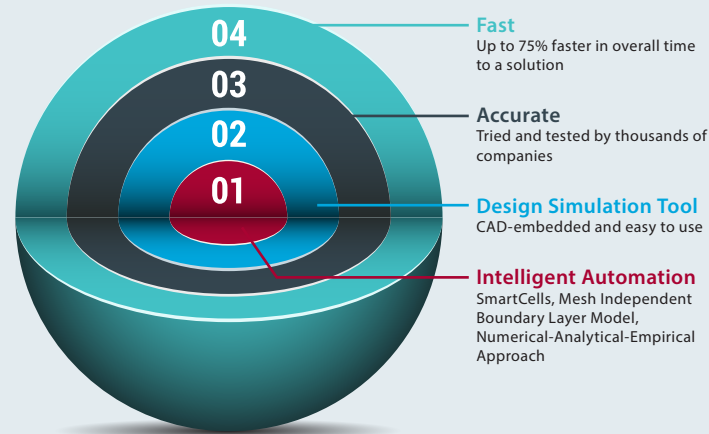
Just because FloEFD is easy to use it doesn't mean it isn't accurate. In fact, the accuracy of FloEFD results has been verified by thousands of companies that use the software to solve complex design problems. The intelligent technology at its core is what makes FloEFD just as if not more accurate than other CFD software and without the burden required by the other systems.

There are many testimonials on the accuracy of FloEFD. For example, the Society of Automotive Engineers of Japan (JSAE) conducted a blind benchmark for commercial CFD software to demonstrate their accuracy against test validation data. The JSAE blind benchmark proved that FloEFD is just as accurate if not more accurate than other commercial CFD software for challenging automotive external aerodynamic study. For a detailed version of the benchmark please read <http://go.mentor.com/4Phzl>.



“Meshing of FloEFD is very robust... FloEFD eats everything raw and meshes it.”

- Danfoss Silicon Power GmbH



Experimental test results from particle imaging velocimetry (PIV) measurements contour plots show that FloEFD and STAR-CCM+ (SST k-ω) can be seen most closely match the wind-tunnel results best.

## FloEFD is Fast

With FloEFD you don't sacrifice speed for accuracy. FloEFD is fast. It's so fast that it can reduce the overall simulation time by as much as 75%.

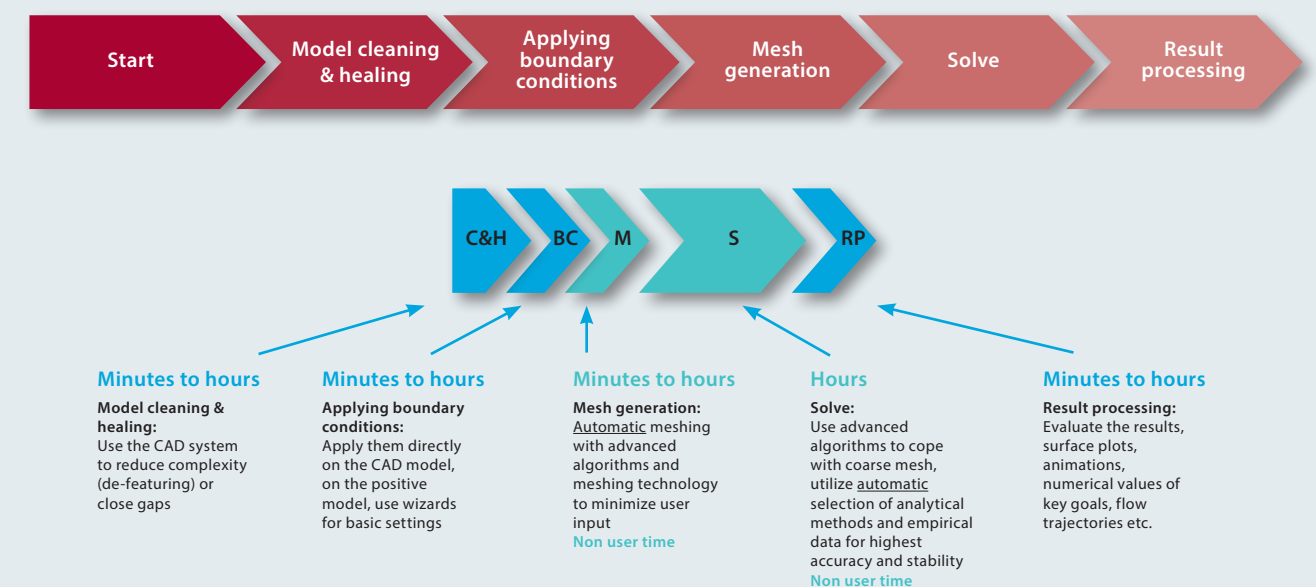
How? Let's take a look at the process.

Gone are the days when your analysis model and your design model bifurcate and move down the design process at different rates. As FloEFD is embedded into CAD it uses native geometry. Therefore, you don't lose any time on transferring a model, modifying it, cleaning it or generating extra geometry to represent the fluid domain. You can immediately prepare your model for analysis – in fact your material properties and

boundary conditions remain with the model so when you modify your model for another round of analyses, you don't have to repeat this step again! In addition, you can mesh highly complex models quickly. And since the design process is iterative in nature, you can simply create different variants and analyze them quickly. In short, you can analyze your problem quickly and within engineering timeframes.

After all, design engineers are responsible for design – all other tasks are secondary. In order for analysis to become a part of the design process, it needs to be easy to use, agile, fast and accurate.

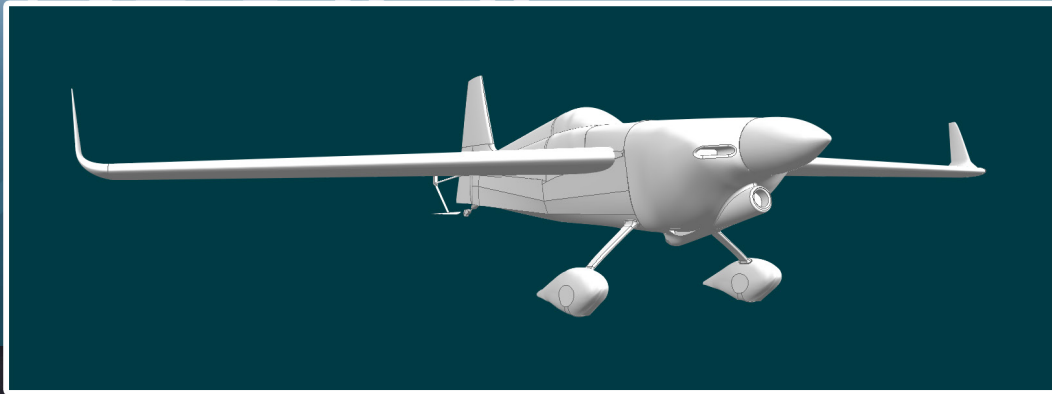
How does your CFD solution meet these requirements?



“We have 8 designers in our group and three use FloEFD. You can use it once every three months because you won't forget how to use it! The special thing about FloEFD is that you are closer to reality in this software.”

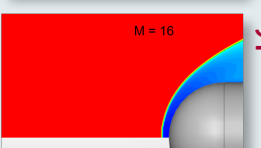
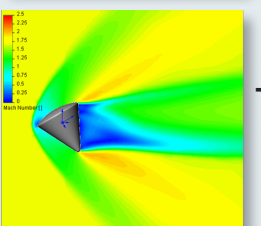
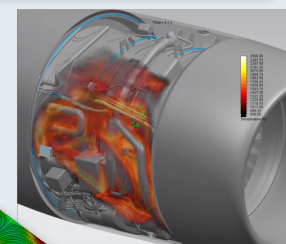
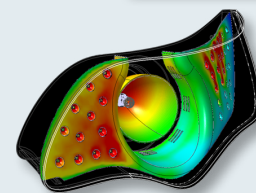
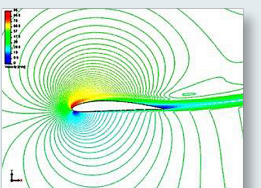
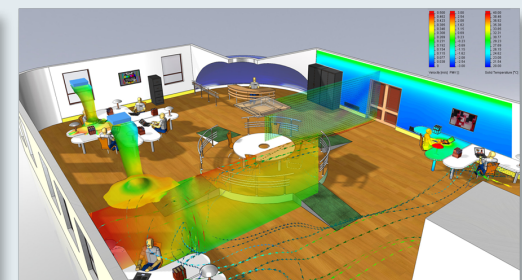
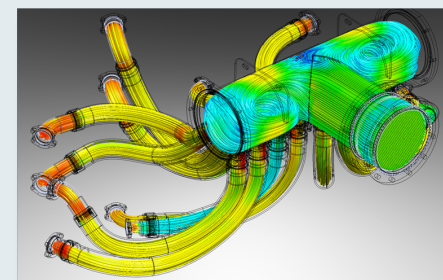
- Orbotech





## FloEFD Capabilities

- Heat conduction in fluid, solid and porous media with/without conjugate heat transfer and/or contact heat resistance between solids.
- Subsonic, transonic, and supersonic gas flows, hypersonic air flows with equilibrium dissociation and ionization effects.
- Radiation heat transfer between opaque solids, absorption in semi-transparent solids and refraction in semi- and transparent solids.
- Volume (or surface) heat sources, e.g. due to Joule heating, Peltier effect, etc.
- Various types of thermal and electrical conductivity in solid medium, i.e. isotropic, unidirectional, biaxial/axisymmetrical, and orthotropic.
- Equilibrium volume condensation of water from steam and its influence on fluid flow and heat transfer.
- Water film evolution (surface condensation/evaporation, melting/freezing, film motion).
- Joule heating from direct electric current in electrically conducting solids.
- Fluid flows with boundary layers, including wall roughness effects.
- Fluid flows in models with moving/rotating surfaces and/or parts.
- Compressible gas and incompressible fluid flows.
- Relative humidity in gases and mixtures of gases.
- Multi-species fluids and multi-component solids.
- Fluid flows and heat transfer in porous media.
- Steady-state and time-dependent fluid flows.
- Cavitation in water and other liquids.
- Free, forced, and mixed convection.
- Combustion in gas-phase mixtures.
- Two-phase (fluid + particles) flows.
- Laminar and turbulent fluid flows.
- Flows of non-Newtonian liquids.
- External and internal fluid flows.
- Real gases with phase change.
- Flows of compressible liquids.
- FEA Bridges including Mechanica, Patran, MpCCI and SolidWorks Structural Analysis
- Optional Add-ons:
  - Electronics cooling module: Besides extended databases of fan curves and packaging material or 2-Resistor components, it also contains specialized features and physics such as a PCB model and joule heating due to current flow.
  - LED Module: The LED Module is the ideal module for the lighting industry and especially for LEDs, it includes a new unique LED model that provides the junction temperature and hot lumen.
  - Advanced Module: provides capabilities to consider hypersonic flow for up to Mach 30 and combustion.
  - HVAC Module: Special simulation capabilities for comfort and safety for occupied spaces such as buildings and transportation vehicles.
  - EDA Bridge: Import data from EDA software such as Mentor Expedition, Cadence, Zuken and Altium.



Subsonic

Supersonic

Hypersonic



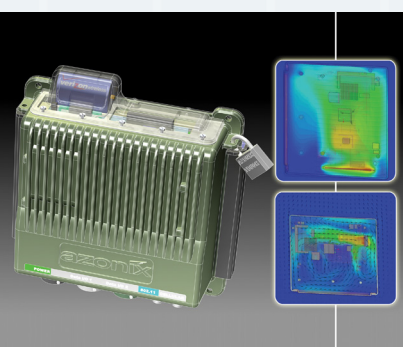


“We like FloEFD because it is fast in calculation for steady analysis. Since we have no specialist CFD experts, our designers take care of simulation analysis. FloEFD is the best for CFD because of its simplified auto-meshing setting inside our preferred CAD package, PTC Creo. We found the cut cell CFD function to be very valuable.”

- Mitsubishi Materials Corporation

“Anthony Kumpen won by a margin of just nine points, and we believe using FloEFD was valuable in his win by improving the speed and performance of his race car. We value the FloEFD software because it optimizes product performance and reliability, while eliminating physical prototyping and reducing product development costs.”

- Voxdale

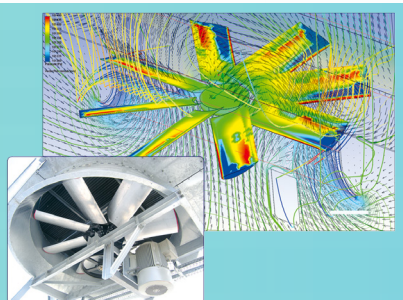
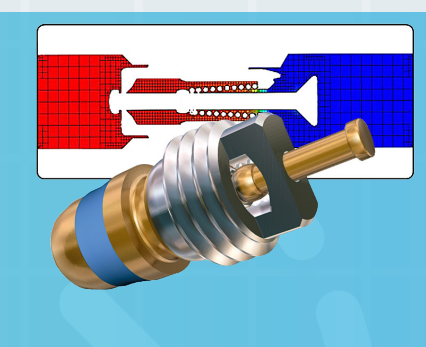


“FloEFD computational fluid dynamics software enables design engineers without a fluid analysis background to perform thermal simulation. The result is that we got the design right the first time, only had to make one prototype and avoided expensive design changes that typically occur in the late stages of the development process.”

- Azonix

“CAD-embedded CFD makes it possible to determine simulation results nearly as fast as we can change the design. The result is that we were able to improve the flow rate of our new CO2 valve by 15% while eliminating about 50 prototypes and reducing time to market by 4 months.”

- Ventrex

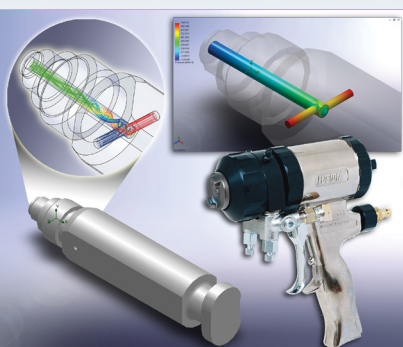


“Physical measurements were essential to the project’s success, but couldn’t produce the needed data in every case. With simulation we were able to look at static pressure distributions through a flow field and get information on the total pressure, which is a direct measure of the entropy in the system. A loss in total pressure is energy loss, and CFD delivers a color picture of where the losses are. We could never have hoped to measure that with physical measurements alone.”

- Bronswerk

“We can show the finished design to our customer complete with how it looks and works in just one day – that’s a savings of 3 weeks and thousands of euros for each model.”

- JAZO

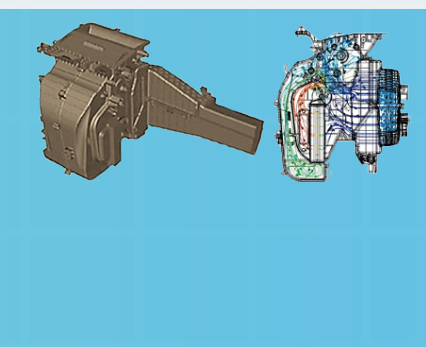


“The complete design process of all sizes took only about 3-4 months or about one-third as long as would have been required using the trial and error method.”

- Graco

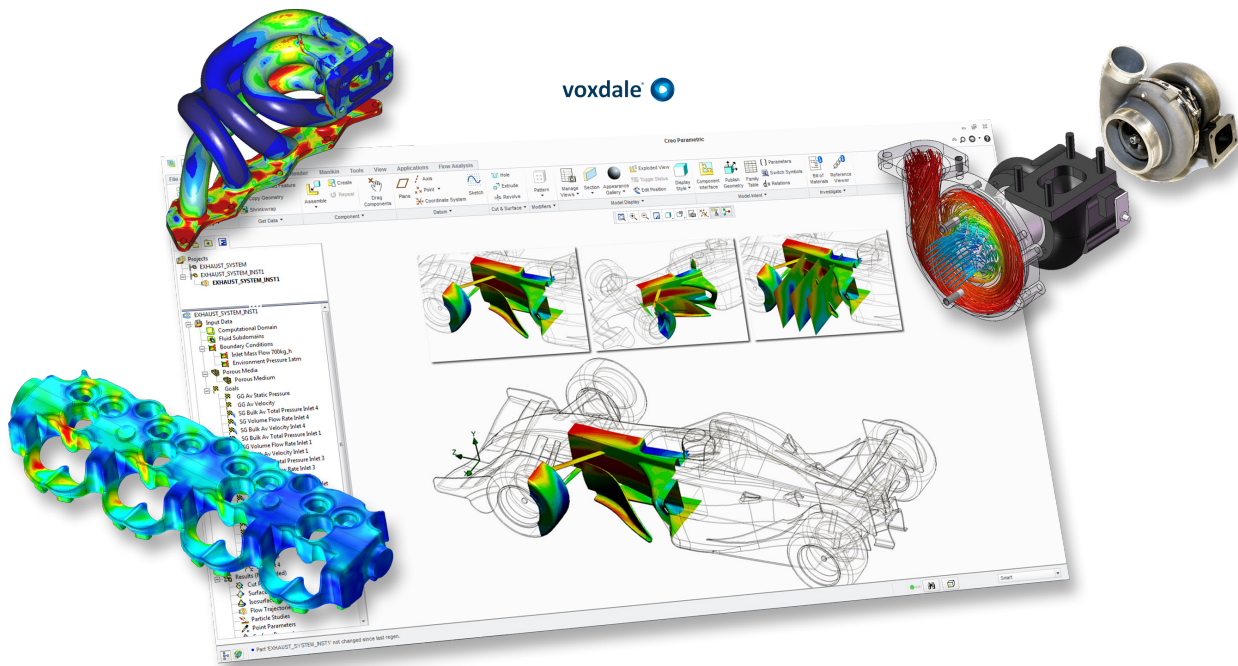
“We found that FloEFD gives more accurate and more efficient CFD simulation results. Since it works within the mechanical CAD environment, it is a highly engineered universal fluid flow and heat transfer analysis software. It showed great design performance improvements in terms of achieving an optimized design while at the same time reducing our overall cost of development.”

- Pan Asia Technical Automotive Center





# FloEFD for Creo



Analyze and optimize complex fluid flow and heat transfer effects on your designs directly inside Creo Parametric with FloEFD for Creo - the only fully embedded frontloading CFD tool for Creo.

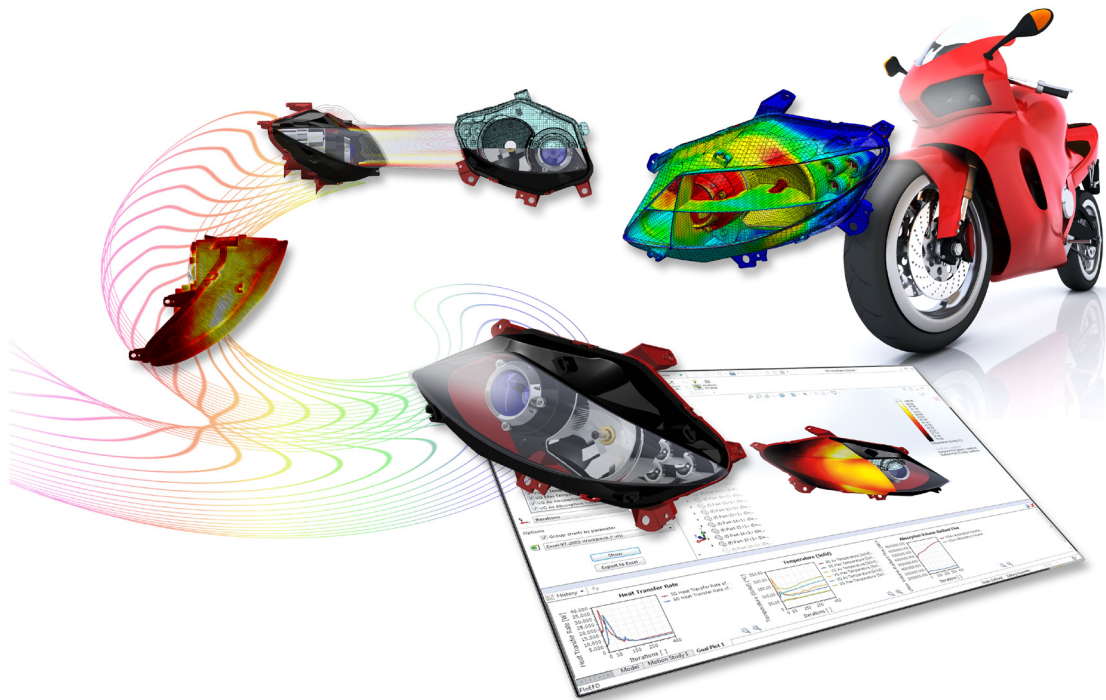
When you're ready to test your design, you simply go to the "Flow Analysis" menu in Creo and start preparing your model. It is that simple.

FloEFD for Creo features an intuitive user interface which uses engineering terms instead of technical jargon; therefore, you can get started quickly and focus on solving flow problems as opposed to figuring out how to use the software. And

because FloEFD interacts directly with native Creo CAD data – without requiring any translation or copies – your model keeps pace with on-going design changes. In addition, its powerful mesher and robust convergence criteria make light work of even the most complex geometries. And the compare configuration and parametric study capability enables you to understand the influence of changes in the geometry or boundary conditions on the results. You can even export the results to Creo Simulate for structural simulations.

In short, use FloEFD for Creo and find out how easy it is to optimize your designs quickly and effortlessly.

# FloEFD for CATIA V5



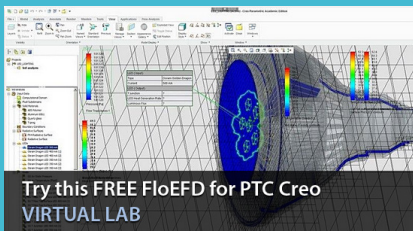
Gain valuable insight into complex fluid flow and heat transfer effects on your CATIA V5 designs quickly and effortlessly with FloEFD for CATIA V5.

Unlike other CFD programs, FloEFD is fully embedded into your CAD platform of choice. It interacts directly with the native 3D CAD data. Therefore, you don't need to translate or transfer your model or learn how to use a new interface – you simply prepare your model for analysis directly inside CATIA V5.

Through a mix of intelligent automation, powerful engine and

intuitive user interface, FloEFD can make light work of analyzing even the most complex geometries. Use the compare configuration and parametric study capability inside FloEFD to understand how changes in the geometry or boundary conditions influence your designs and only advance the designs that best meet specifications.

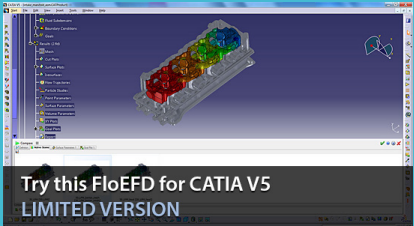
In short, if you use CATIA V5 for design, you need FloEFD for CATIA V5 - the only affordable and intuitive CFD simulation tool that fits into your design process without requiring you to change the way you design products.



## Take FloEFD for Creo for a Free Spin on the Cloud

Go to **FloEFD for PTC Creo Virtual Lab** and get started immediately

<http://go.mentor.com/4TNVx>



## Take FloEFD for CATIA V5 for a Free Spin

Go to **FloEFD for CATIA V5 Limited Version** and get started immediately

<http://go.mentor.com/4TNVz>



“The most important consideration in selecting an analysis software tool was that all team members could use it regardless of their level of ability...The people who don't have much experience of analysis can use it easily...It was important that the tool integrated with Pro/ENGINEER. We didn't want to have to create another model for analysis and being CAD-embedded we could validate various analysis models repeatedly. We also wouldn't have any difficulty in switching between processes (from design to analysis).”

- Seiko Epson

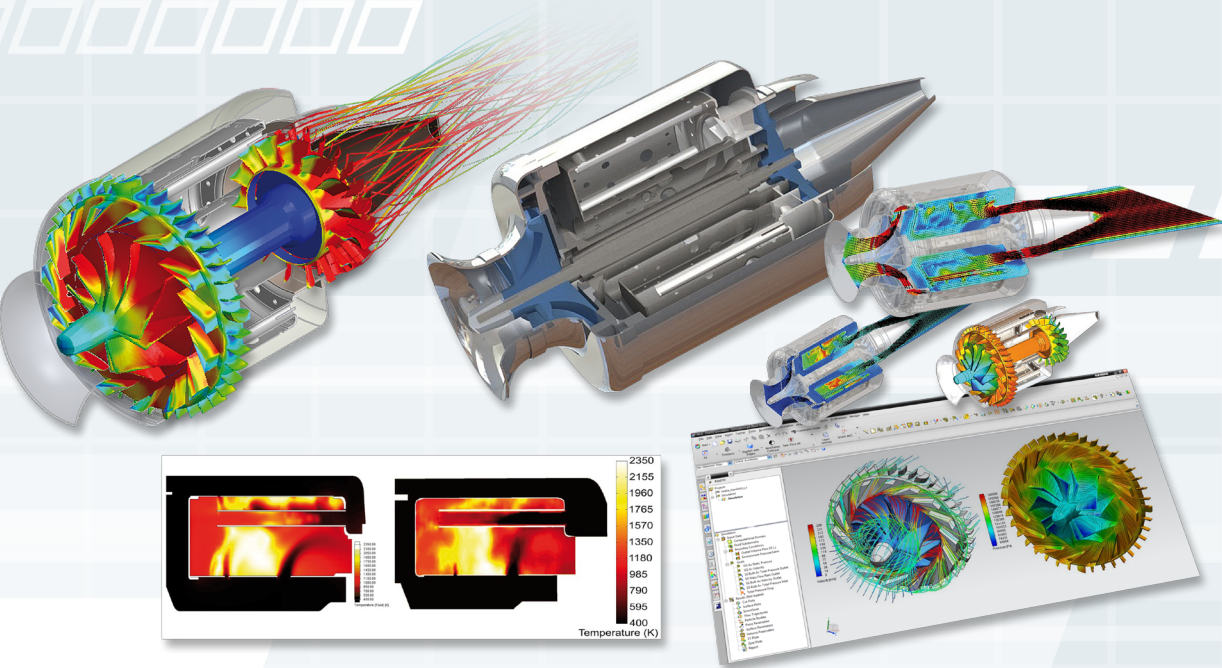


“FloEFD from Mentor helps us to understand and optimize headlamps. Even very complex geometries and test conditions can be investigated with a minimum of effort. New features such as Monte-Carlo radiation and the LED module are especially helpful in speeding the development of very complex products.”

- Automotive Lighting



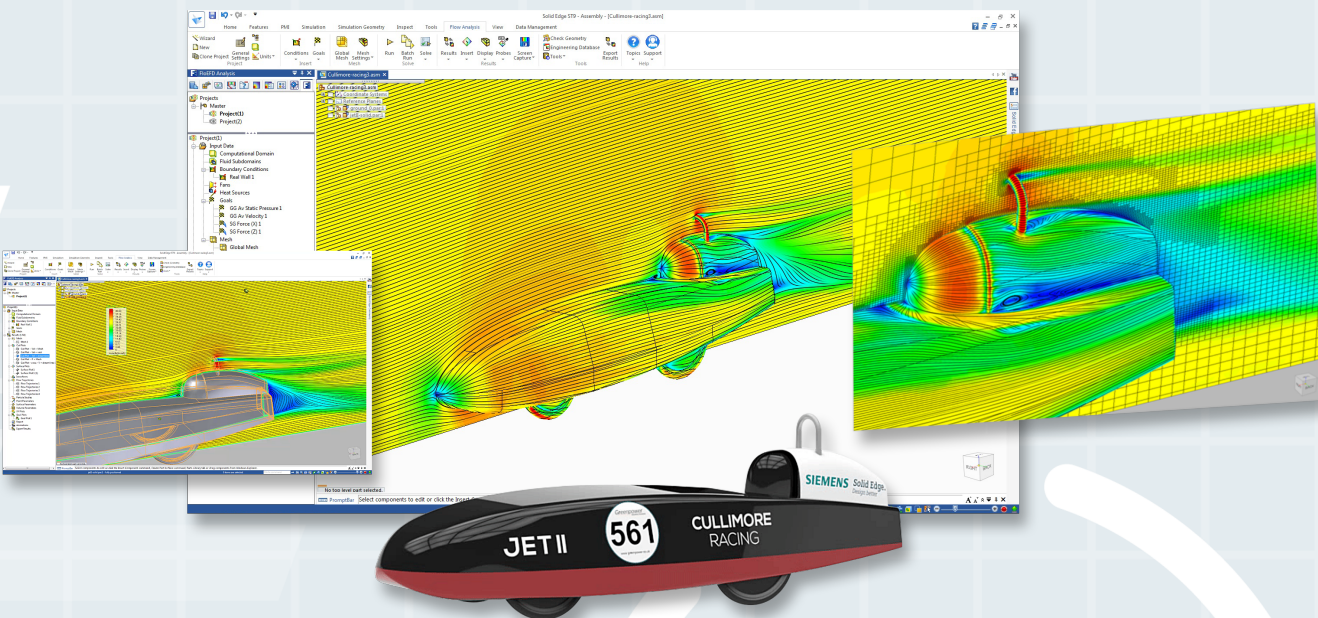
# FloEFD for NX



FloEFD is a fully embedded frontloading CFD solution for Siemens NX. FloEFD for NX enables you to gain insight into and optimize complex fluid flow and heat transfer effects on your designs directly inside NX. It “looks and feels” exactly as NX. In other words, you use the same user interface for design and analysis thereby rendering CFD simply as an extension of your design process. Unlike 3rd party CFD programs, FloEFD interacts directly with the native 3D CAD data without any transfer or translation so

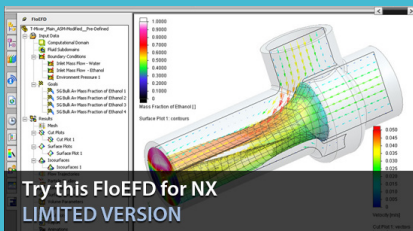
you can keep pace with on-going design changes. It features a powerful mesher which can automatically mesh highly complex geometries. And its compare configuration and parametric study capability helps you understand how changes in the geometry or boundary conditions affect the analysis results. Lastly, you can export the results to NX NASTRAN for even more realistic structural simulation. If your organization is interested in CFD analysis, then simply add FloEFD for NX without any disruption to your current design process and start creating the digital twin.

# FloEFD for Solid Edge

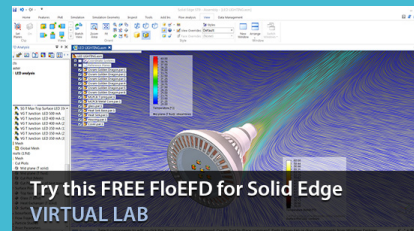


CFD is no longer out of the reach of the SMB. Use the powerful combination of Solid Edge and FloEFD for Solid Edge for a complete design and CFD analysis suite. Simply create your model in Solid Edge and immediately prepare and analyze it – without any translation or fluid body creation. Since FloEFD runs directly inside Solid Edge you don’t need to learn how to use a second interface in order to use analysis. Use the powerful wizard to start the process.

Take advantage of its powerful technology including an automatic mesher and very robust solver to get real insight into how your designs will perform in real life. And understanding the impact of changes on your design is easy – simply let its compare configuration and parametric study tool to do the heavy lifting for you. With FloEFD you can create better designs, faster.



**Try FloEFD for NX for Free**  
Go to **FloEFD For NX Limited Version** and get started!  
<http://go.mentor.com/4TNVB>



**Take FloEFD for Solid Edge for a Free Spin on the Cloud**  
Go to **FloEFD for Solid Edge Virtual Lab** and get started immediately  
<http://go.mentor.com/4TNVw>



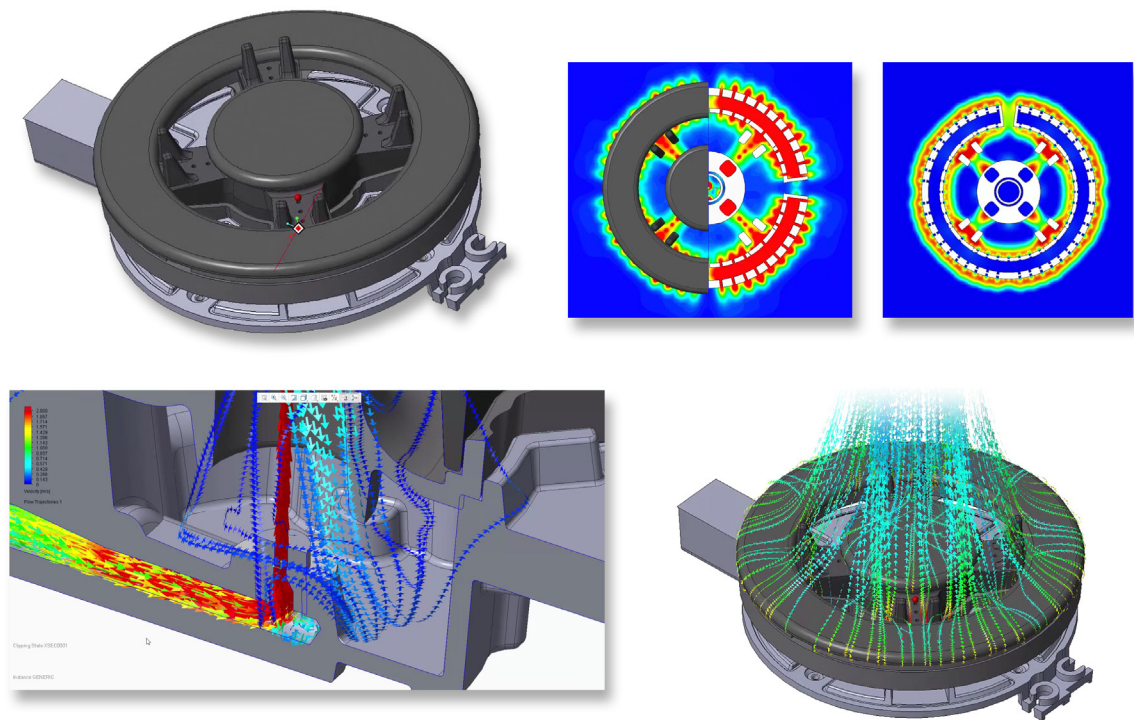
“We anticipate that because FloEFD is embedded in the NX CAD-interface it will benefit our design group by minimizing design-to-analysis time. This product has the potential to utilize design enhancements made during the analysis to save us three days or more in our current process, making us more productive while improving overall design quality.”  
- Hutchinson



“FloEFD helped me to work on contracts that involved very complex geometries, such as a stator coil end turn support system, which I wouldn’t have been able to do with other CFD software.”  
- E-Cooling



# FloEFD General Purpose Frontloading CFD



FloEFD is a standalone efficient frontloading CFD analysis solution for the design engineer. FloEFD offers a powerful and easy-to-use CAD-like user interface. Transfer geometry from most popular CAD programs and start preparing your models. Its powerful

mesher and robust convergence criteria make light work of even the most complex geometries. Evaluate the design envelope by assessing results by numerical values, graphs, visual images and animations. With FloEFD gaining insight into your designs is easy.

## Take FloEFD for a Free Spin on the Cloud

Go to **FloEFD Standalone Virtual Lab** and get started immediately

<http://go.mentor.com/4OkSk>

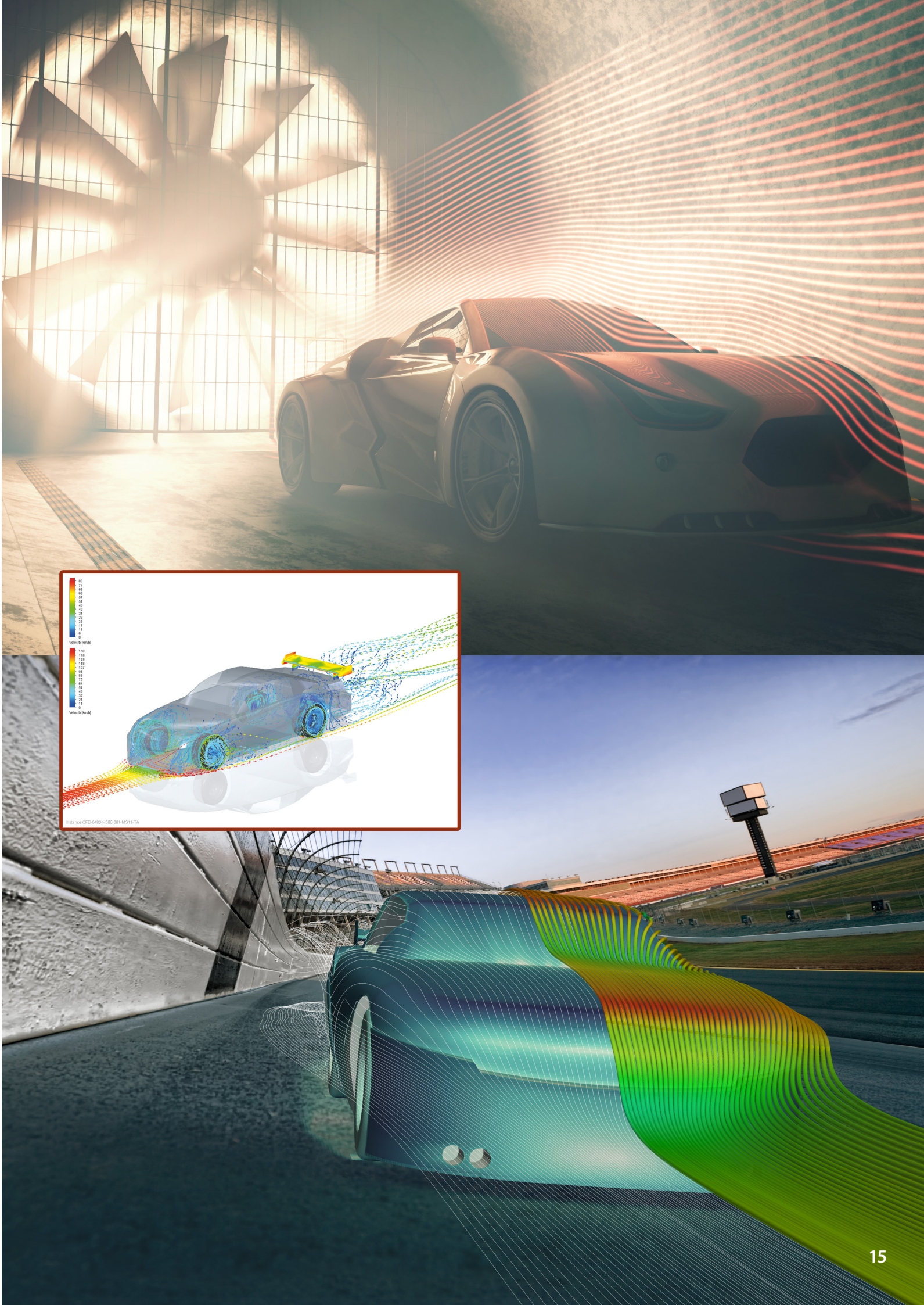


Try this **FREE FloEFD Standalone VIRTUAL LAB**



“The entire design, simulation and physical testing process took half the length of time it would have taken using traditional design processes.”

- Marengo







# Mentor®

A Siemens Business

**Mentor Graphics Corporation**, a Siemens business, is a world leader in electronic hardware and software design solutions, providing products, consulting services, and award-winning support for the world's most successful electronic, semiconductor, and systems companies.

Corporate headquarters are located at  
8005 S.W. Boeckman Road,  
Wilsonville, Oregon 97070-7777.

Web site: <http://www.mentor.com>.