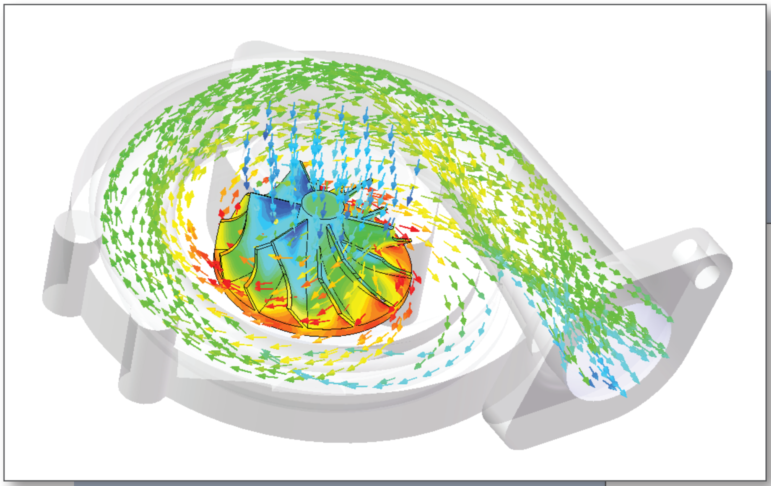
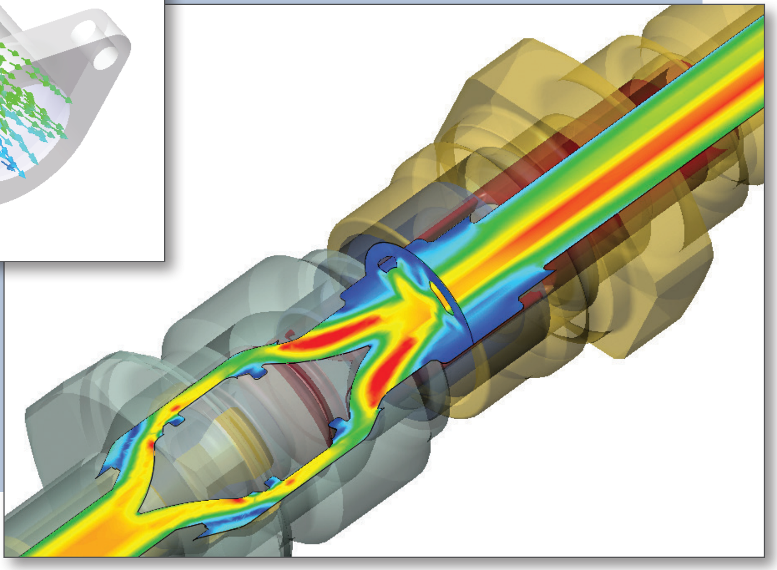
CAD-INTEGRATED COMPUTATIONAL FLUID DYNAMICS IS ESSENTIAL TO PRODUCT ENGINEERING SUCCESS

White Paper

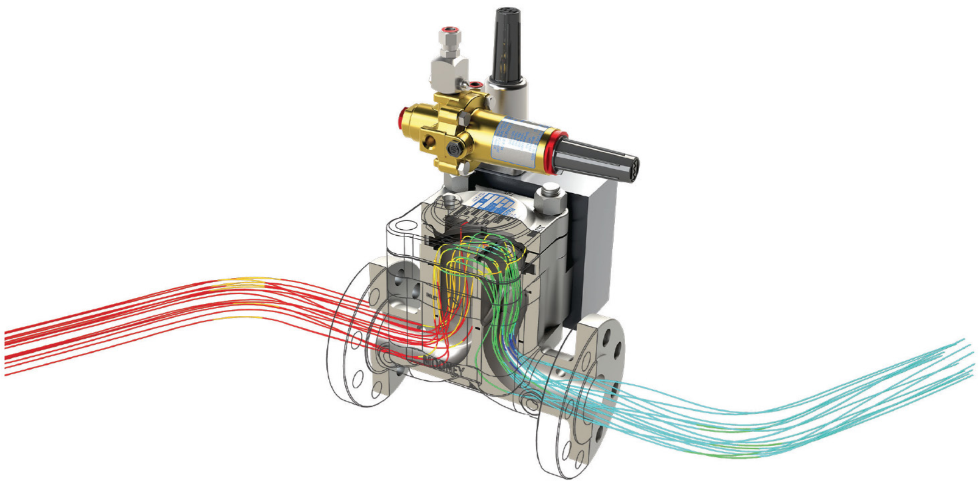




SUMMARY

Engineering Fluid Dynamics (EFD) is a new breed of Computational Fluid Dynamics (CFD) soft ware that enables mechanical engineers to simulate fluid flow and heat transfer applications with powerful, intuitive, and accessible 3D tools. Engineering fluid dynamics is driven by engineering criteria and goals so every product engineer can get the technical insights necessary to answer the questions faced in the product development process.

SOLIDWORKS® Flow Simulation is based on the same mathematical foundation as traditional computational fluid dynamics (CFD) soft ware, but key benefits set SOLIDWORKS Flow Simulation apart, making it quicker and easier to use, while still delivering a robust and highly accurate solution.



Examine Fluid Flow in and around your design to detect turbulence and recirculation issues and determine flow conditions

REASON 1: USING EXISTING GEOMETRY

With most traditional CFD programs, you need to modify settings substantially and then transfer existing CAD models to a different program in order to create a model for analysis. The main reason is that traditional CFD programs require a considerable amount of manual intervention. The translation process might work for 80 percent of the geometry but the rest has to be re-created or simplified by hand. Some users have reported wasting days waiting for their model to be transferred—if it succeeds at all! Therefore, many users have found it more reliable to start from scratch by creating the geometry in the CFD program (although this activity also involves a considerable expenditure of time).

Recently a company testing a traditional tool popular in the aerospace industry reported a difference of two weeks compared to two days with SOLIDWORKS Flow Simulation technology. They reported that they spent a majority of their two weeks trying to get their geometry into their current traditional CFD tool. But the same complex geometry was transferred into SOLIDWORKS Flow Simulation and analyzed in under two days—a significant savings.

The main difference is that SOLIDWORKS Flow Simulation uses native SOLIDWORKS 3D CAD data directly for fluid flow simulation. The fluid domain is automatically created based on the geometry and then is automatically updated for any design changes. Flow conditions are defined directly on the SOLIDWORKS CAD model and organized similarly to other design data in the feature tree. As a result, the original SOLIDWORKS CAD model is used natively for the analysis with SOLIDWORKS Flow Simulation, saving preparation time and making sure the design updates are taken into account for the CFD analysis.

REASON 2: HANDLING COMPLEX GEOMETRY

To understand how your design behaves in real-world conditions, you need to simulate its performance in its operating environment. All analysis vendors recommend that you defeature your models. But then how do you know how far to take it and whether your analysis results will actually reflect their field conditions?

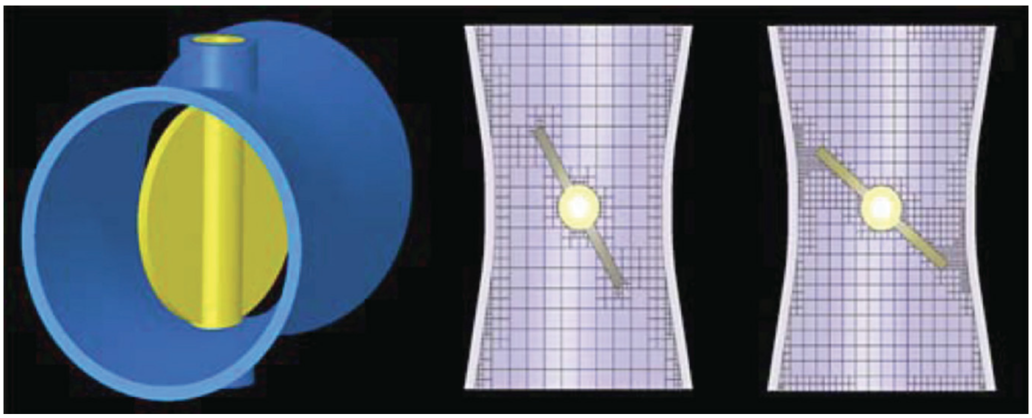
SOLIDWORKS Flow Simulation is extremely robust and can handle very complex geometries. It can easily handle CAD geometries containing tight crevices and sharp angles without needing to defeature the model. However, if you decide to simplify your model, you have access to powerful simplification geometrical features for preparing your model for the CFD analysis.

REASON 3: EFFORTLESS MESHING

In a standard CFD package, obtaining an optimum mesh is not easy. However, this is perhaps one of the most important steps in the analysis process. After all, meshing directly affects the accuracy of the results. One airplane engine manufacturer spends a minimum of three months on finding the best mesh for its designs.

While automatic meshers have long been available, traditional CFD tools still require a considerable amount of manual intervention in order to maintain the quality of the mesh by eliminating gaps and overlaps, and maintaining the required skewness, aspect ratio, warpage, and volume of individual cells. This manual process has to be repeated for every design change.

SOLIDWORKS Flow Simulation offers you an extremely robust automatic mesher for fluid and solid regions with automatic mesh refinement/unrefinement due to geometrical or physical requirements. SOLIDWORKS Flow Simulation also features grid-independent near-wall modeling by using Partial Cells technology. This technology enables the software to correctly simulate the boundary layer phenomena for fluid flow and heat transfer effects. The result is that new parts and design changes can be meshed in a matter of minutes, dramatically reducing the time required for analysis.



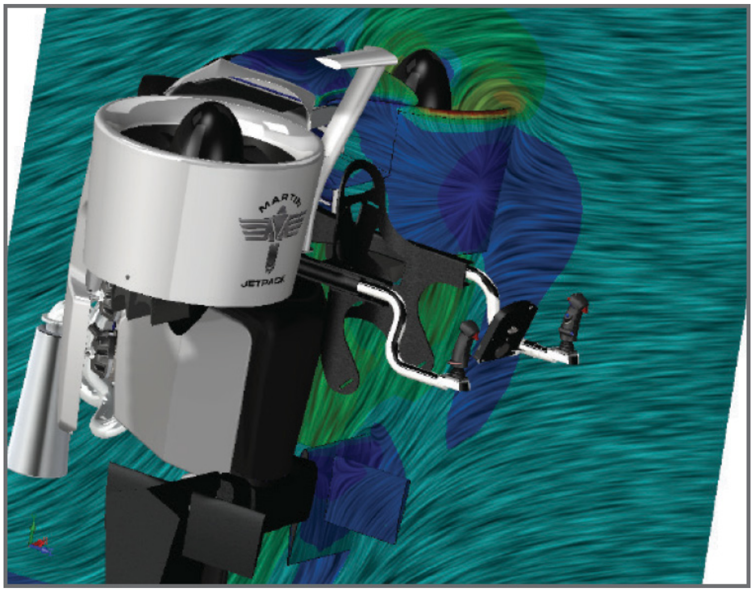
The Cartesian mesh generation enables SOLIDWORKS Flow Simulation to offer a best-in-class CFD solution centered upon simplicity, speed, and robustness.

REASON 4: CREATING ADDITIONAL GEOMETRY IS UNNECESSARY

When conducting fluid flow or thermal analysis, you are interested in understanding what is happening in the empty region (the fluid) and how it affects the solids which it comes into contact with. However, the space which is filled with liquid or gas is not normally modeled as a separate solid in the design.

Other analysis programs require you to create additional geometry in your solid modeler to represent this region. While some programs can create solids for internal flow volumes automatically, unfortunately, they do it indiscriminately. Therefore, they create solids even for unnecessary isolated volumes that you're not interested in analyzing.

SOLIDWORKS Flow Simulation automatically creates the fluid domain and can identify all "empty" spaces—the enclosed internal flow space and the outer flow area, as well as the solid areas of different materials involved in heat transfer. In addition, it excludes cavities without flow conditions to avoid unnecessary mesh creation.

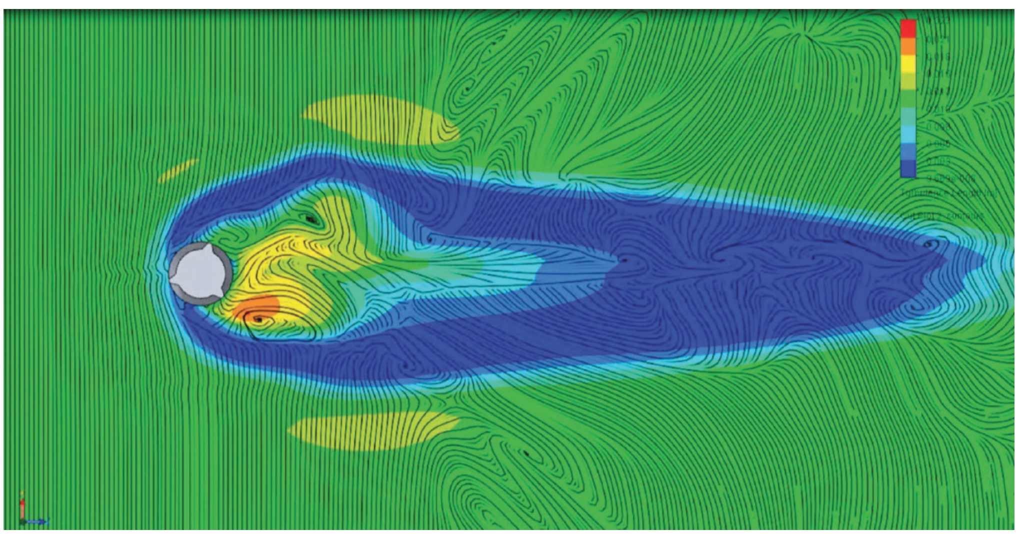


Complex or large geometry are handled for fluid flow and heat transfer simulation in SOLIDWORKS Flow Simulation

REASON 5: CRYSTAL BALL... NOT NEEDED!

With SOLIDWORKS Flow Simulation you do not need to choose between turbulent or laminar flow as its modified wall function supports a laminar-turbulent transition model.

In addition, SOLIDWORKS Flow Simulation will account for compressibility automatically.

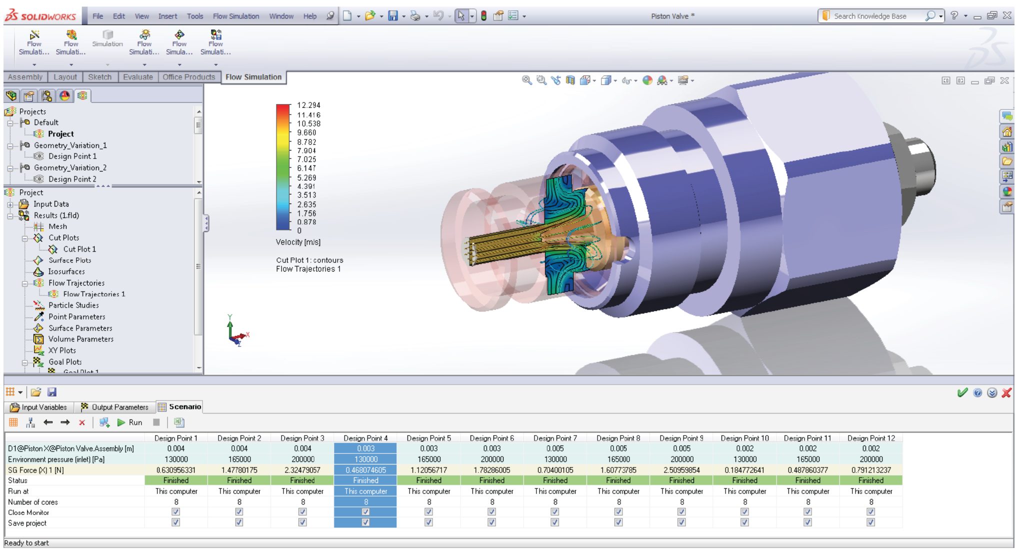


The modified k-ε turbulence model describes laminar, turbulent, and transitional flows automatically.

REASON 6: POWERFUL "WHAT IF" ANALYSIS

Solving flow and heat transfer problems is an iterative process: after seeing the initial analysis results, most users modify their models repeatedly in order to solve problems. When your 3D design and analysis platform is integrated, as in the case of SOLIDWORKS Flow Simulation with SOLIDWORKS, after your initial run, you simply create multiple clones of your model. The cloned models retain ALL analysis data such as loads and boundary conditions. So when you modify your solid model, you can immediately analyze it without having to re-prepare it. It really is that simple.

With other programs you may need to go back to the original CAD model. And while you may be able to use the CAD cloning feature, after the translation process, you still need to reapply all your loads and boundary conditions. This becomes a real hassle if you want to analyze several versions of your model.



Product Engineers tackle a vast number of fluid flow problems using the parametric study which makes the evaluation of various scenarios of the design intuitive and powerful.

REASON 7: SOLIDWORKS FLOW SIMULATION IS AFFORDABLE

Another factor that sets SOLIDWORKS Flow Simulation apart is cost. Traditional CFD codes cost in the region of $25,000 to lease for one year. An even greater cost for most companies is the need to hire experts to use the software. These analysts need to spend a considerable amount of time training in order to keep up with the latest changes in the code.

SOLIDWORKS Flow Simulation substantially reduces the cost of performing fluid flow and heat transfer analysis. The software is reasonably priced for a perpetual license. SOLIDWORKS Flow Simulation also can be used by mechanical design engineers with minimal training. Finally, SOLIDWORKS Flow Simulation runs on personal computers and even laptops which cost just a few thousand dollars.

SOLIDWORKS Flow Simulation is truly the right choice for best-in-class manufacturers.

REASON 8: LARGE COMMUNITY OF USERS

The SOLIDWORKS Community is a large product engineers community developing innovative, best-in-class products worldwide. Thousands of SOLIDWORKS companies in high tech, life science, and industrial equipment benefit from SOLIDWORKS Flow Simulation to develop their product with CFD insight.

Moreover, SOLIDWORKS Flow Simulation is included in the SOLIDWORKS educational offer so students—future engineers—learn virtual simulation and CFD with this unique engineering approach.

YOUR DECISION CHECKLIST

Think about these eight key points for efficient and concurrent CFD analysis:

1. Can I use my existing 3D geometry? If yes, how robust is the transfer process?
2. How are design changes taken into account?
3. Do I need to simplify my models? How would I do it?
4. How is the fluid domain created?
5. What is the level of automated meshing available to me?
6. Do I need to designate the task to be laminar or turbulent flow?
7. How can I test design scenarios with "what if" or parametric analysis?
8. Is the CFD software intuitive enough for nonexperts? Can I read customers' references?

To learn more about SOLIDWORKS Flow Simulation, visit
http://www.solidworks.com/sw/products/simulation/flow-simulation.htm
or contact your local authorized SOLIDWORKS reseller.

SIMTEK
Real Time. Real Results

SIM Technologies Pvt. Ltd.

3rd Floor, "Mamanjee Centre" ,
S7-A, Thiru-Vi-Ka Industrial Estate,
Guindy, Chennai – 600 032

+91-8754447021 marketing@simtek.in