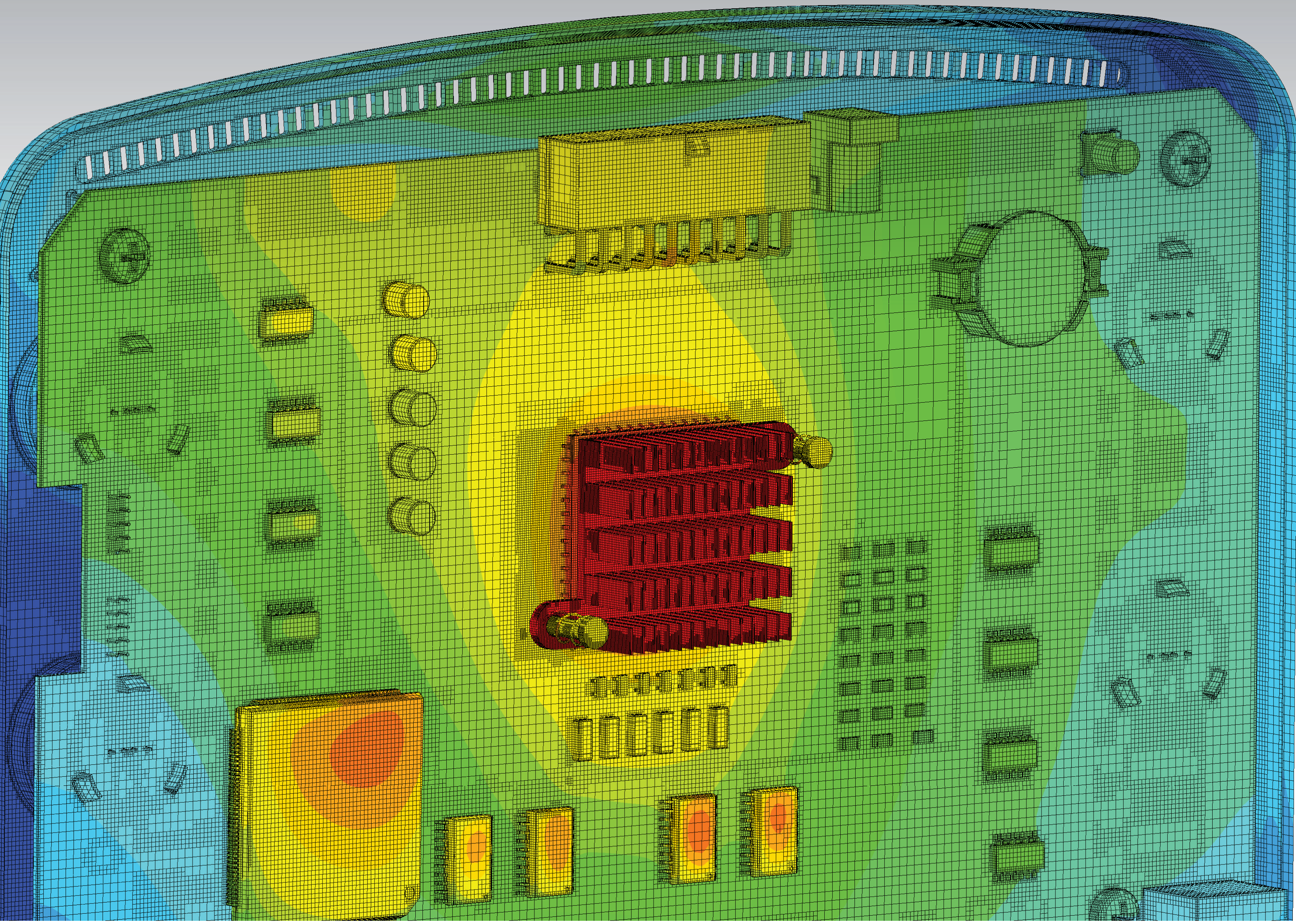
SMARTCELLS®
ENABLING FAST & ACCURATE CFD

White Paper



This white paper introduces the concept of SmartCells® in the context of Computational Fluid Dynamics (CFD) simulation software. They exist in SOLIDWORKS Flow Simulation™.This document will explain what they are, how they differ from other meshing approaches employed in the CFD industry, and their benefits to CFD users:

- SmartCells are Cartesian based meshes that are typically 10 times smaller in size than traditional CFD meshes while providing the same level of flow field resolution and attaining high levels of simulation accuracy.
- They employ numerical approaches and engineering models embedded within SOLIDWORKS Flow Simulation that typically reduce the manual time spent by traditional CFD tool users in meshing their geometries by an order of magnitude on average thus dramatically reducing this historical bottleneck to CFD user productivity.
- They ultimately allow for automated mesh generation because of their built-in artificial intelligence based on decades of industrial CFD simulation experience using engineering model data.
- SmartCells deal with both the ubiquitous need for turbulent boundary layer simulation accuracy in all fluid flow processes while handling complex geometries which may have many fluid-solid control volume zones in one mesh SmartCell.
- They lead to design engineer and CFD analyst workflow gains of typical analyst workflow gains of between twofold and tenfold through the associated built-in pre-processing and user experience benefits inside SOLIDWORKS Flow Simulation.

INTRODUCTION

Computational Fluid Dynamics is a well-established computer-aided engineering simulation software industry of over 40 years standing with the commercial sector accounting for over $1Bn/yr in revenues worldwide today (Ref 2). The bulk of CFD simulation (over 90%) carried out globally is based on the finite volume (FV) methodology using a Reynolds-averaged Navier-Stokes (RANS) approach because of its robust nature and computational efficiency (Ref 2). CFD is based on well-accepted numerical methods for solving the fundamental Navier-Stokes equations that govern fluid flow, heat and mass transfer (Ref 3). But the technology enablers for the traditional CFD industry invariably are a synergy between numerical and engineering techniques and analytical methods that are between 30-40 years old (Ref 4). Indeed, the vast majority of CFD carried out in the world today is based on variants of the tried-and-proven k-E turbulence model, the accepted workhorse of the industry that is now over 40 years old (Ref 5).

First-time users of traditional CFD simulation tools can find them very difficult to use because the user must master very complicated pre-processing (geometry and grid generation) approaches. Frequently the codes themselves demand a deep understanding of the physics and numerical algorithms underlying them because of their inherent mathematical nature. Invariably the quality of a CFD prediction is very much affected by the pre-processing approach employed. By applying new and more modern analytical methods to numerical CFD tasks to resolve phenomena describing fluid flow, heat and mass transfer the required user skills for high accuracy near-wall mesh building and the manual time spent on this task can be reduced. The use of what we call SmartCells in CFD simulations will lead to a coarser mesh being applied to a given application to capture the physical phenomena being resolved (e.g., turbulence vortices, thin channels etc.) due to the implementation of more modern engineering data approaches (Ref 1). As a result, the SmartCell approach enables a reduced cell count for a CFD simulation compared to traditional CFD mesh approaches that are based on a fine resolution of boundary layers. Instead, it enables the automation of the meshing process completely with very low numerical skills or time requirements for the engineer or the CFD analyst using them.

The SmartCells approach to CFD has proven successful over the last 20 years for a wide range of industrial benchmarks and applications, and is regularly employed by OEMs and Tier 1 suppliers in the automotive, aerospace and other industries. Ultimately, the benefit from these synergies of numerical and engineering techniques must be seen in comparison with traditional CFD approaches. With the speed of manufacturing design cycles ever increasing, and always in the context of ever-present Product Lifecycle Management (PLM) software that all engineers use in order to improve their designs, engineers need CFD simulation results ever faster but without loss of accuracy (Ref 6).

The approach described by this white paper enables the use of CFD in design processes by non-experts and experts alike, through the automation of the pivotal meshing task without compromising accuracy of final results, even with coarse meshes compared to typical CFD meshes. Applicability of the SmartCells approach will be illustrated by the simulation of external aerodynamics characteristics of an ASMO generic car model (Ref 7). SmartCells work for engineers who want to dip in and out of CFD usage in their jobs by reducing the numerical skills required for employing and deploying CFD. It is remarkably robust and can be employed by CFD analysts as well as designers, allowing them to "frontload" their CFD simulations (Ref 8) in order to yield maximum simulation productivity at least time cost in a product manufacturing workflow.

NUMERICAL FOUNDATION OF SMARTCELLS

To understand SmartCells, one must first understand the other meshing types typically used in traditional finite volume CFD simulation codes. These tend to include unstructured triangular, structured triangular, structured curvilinear, immersed boundary Cartesian meshes as well as SmartCells (see Table 1 - Ref 6). Table 1 shows a mathematical formulation of CFD simulation accuracy, ||LTE||L1, related to the various CFD meshing approaches used (the lower the LTE number the more accurate the CFD prediction is). The table clearly shows that to offset accuracy issues in unstructured triangular, structured triangular, and structured curvilinear meshes versus cut-cell Cartesian and SmartCells meshes, traditional CFD codes require more and more cell counts to be added to the simulation. This obviously has both a memory and a CPU overhead associated with it.

Mesh	Unstructured triangular		Structured triangular		Structured curvilinear		Cartesian [Aftosmis]		SOLIDWORKS Flow Simulation SmartCells	
	Cells	$\|\|LTE\|\|_{L1}$	Cells	$\|\|LTE\|\|_{L1}$	Cells	$\|\|LTE\|\|_{L1}$	Cells	$\|\|LTE\|\|_{L1}$	Cells	$\|\|LTE\|\|_{L1}$
Results	128	0.52552	144	0.37926	144	0.30998	138	0.03065	140	0.03014
	505	0.22529	525	0.07571	525	0.09223	507	0.00930	516	0.00916
	1918	0.11936	2001	0.01565	2001	0.02422	1928	0.00246	1944	0.00235
	7490	0.05940	7809	0.00347	7809	0.00629	7549	0.00059	7526	0.00058

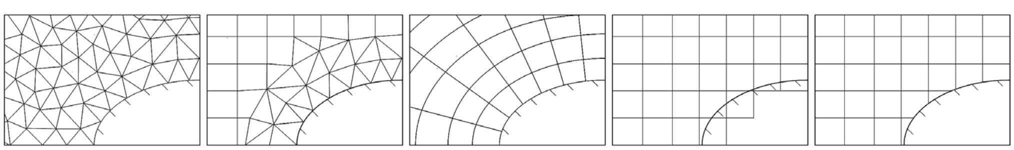


Table 1: Comparing CFD mesh types with the numbers of cells required to achieve a given level of numerical solution accuracy.

It is clear from Table 1 that SOLIDWORKS Flow Simulation can generate accurate results with low Cartesian cell counts when compared to multi millions of cells typically necessary for the same level of accuracy in traditional CFD codes. This is due in part to the numerical methods inherent in SmartCells and in part due to Cartesian cells not suffering from skewness accuracy issues typically associated with tetrahedral, hybrid and polygonal meshes used most frequently in traditional CFD approaches.

Conventional wisdom with the application of CFD is that one needs to add more and more computational grid cells in any given real-world simulation to get higher and higher accuracy by resolving finer and finer details at crucial wall boundary layers in particular. With geometrically complicated applications that include complex narrow passageways, for instance, this may involve hundreds of millions of computational cells with the incumbent memory, CPU and post-processing overheads that come with these large models. These are always necessary to get an accurate traditional CFD solution. However, this approach based on 1980s thinking is insatiable with regard to CPU demands, and invariably sucks up all the available computational resources. Indeed, it could be argued that this bottleneck has been the single biggest barrier to the democratization of CFD usage in the last 25 years (Ref 1). This paper contends that there is another approach to industrial-level RANS CFD that is smarter, computationally more efficient, just as effective, and well validated, but uses orders of magnitude fewer cells, and therefore uses fewer computational resources for the same level of accuracy as traditional CFD approaches. And it is also embedded within CAD and Product Lifecycle Management (PLM) workflows, which is intuitively the most optimal place for CFD simulation to be, thus enhancing user productivity in one familiar CAD/PLM interface.

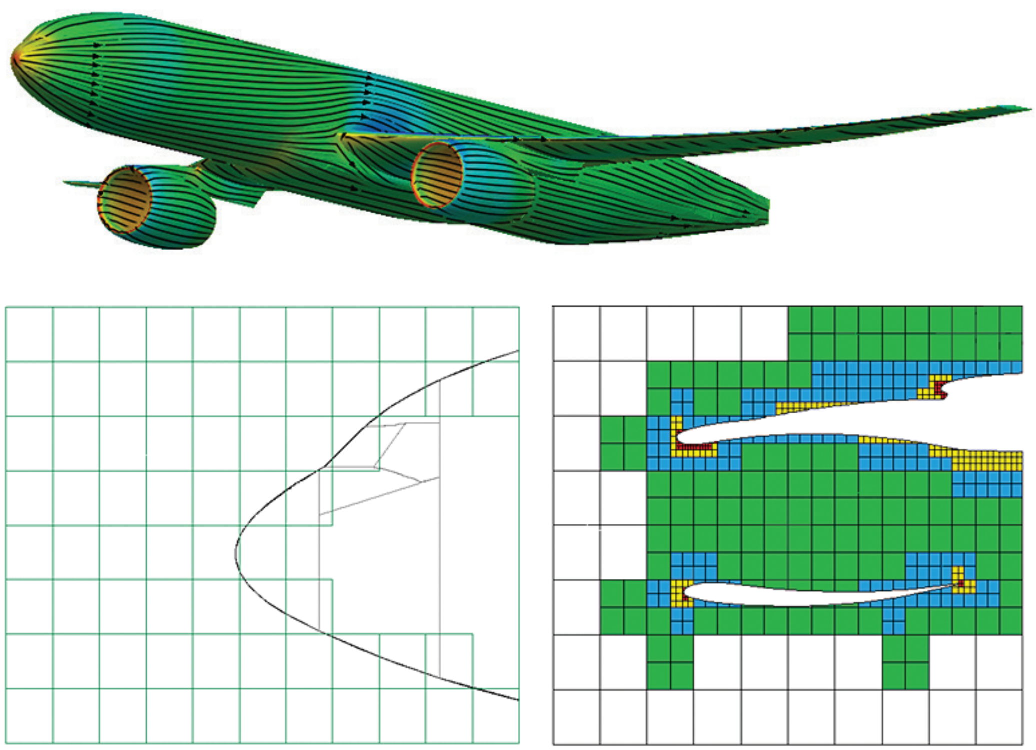
In engineering design simulation practice today, whatever the industry, PLM concepts are widely deployed by engineers as the means by which 3D manufactured product data is used and maintained consistently during an entire product's lifecycle and across all its design changes (Ref 1). The basis of the PLM concept is the availability of complex 3D product model data within a mechanical Computer-aided Design (CAD) system as its central element. 3D product model data is therefore both the foundation and starting point for all virtual prototyping and physical engineering simulations today. The performing of fluid flow simulations using CFD in such a CAD-embedded context is obviously very attractive, as it can not only accelerate the design process, but make these processes more predictable and reliable, against a background of increasing design complexity and dependence on external development partners.

It is essential to note that all major CAD systems were created 20-30 years ago and were optimized as design tools and only later was the necessity for embedding CAE (and in particular CFD) recognized. Therefore it was logical that for some period in the 1980s and 1990s that CFD continued on an independent development trajectory. Nevertheless, from the standpoint of using CFD during engineering design, and as a requirement of all PLM roadmaps, the need to fully embed CFD within CAD becomes more and more pressing (Ref 9). The biggest obstacle to achieving this is high resource requirements for performing CFD calculations as applied to typical real-world complex 3D CAD geometries. In particular, such CFD analyses based on solving the Navier-Stokes equations have specific requirements for detailed grid resolution of flows near the fluid/solid boundaries. Such obstacles first appear during the grid generation stage of CFD followed by more problems at the numerical solution stage. In addition, highly qualified CFD experts are usually required to do such traditional simulations, but such skillsets are rarely available in design engineers. In order to resolve this issue and make CFD calculations less resource consuming and available for design engineers, the "Engineering Fluid Dynamics" (EFD) approach (Ref 1) was developed in the 1990s and this has extended into the product SOLIDWORKS Flow Simulation. This CFD approach inside SOLIDWORKS Flow Simulation is based on 2 main principles:

- Direct use of native CAD as the source of geometry information;
- Synergy of full 3D CFD modeling with simpler engineering methods in the cases where grid resolution is insufficient for full 3D simulation.

THE SMARTCELLS TECHNIQUE

This synergy of CAD and 3D numerical models is a critical element because it allows SOLIDWORKS Flow Simulation to reduce resource requirements on grid generation and numerical solution stages by an order of magnitude, compared to traditional CFD approaches. It simplifies obtaining CFD results and enables usage of complex CAD models as a source of geometry information. Surface and volume mesh grid generators in traditional CFD tools are also usually based on body-fitted algorithms. An alternative approach is to use an immersed-body grid (Ref 6). In this approach, the creation of the mesh starts independently from the geometry itself and the cells can arbitrarily intersect the boundary between a given solid and fluid (see Figure 1 of an aircraft wing-fuselage geometry).



Such an immersed-boundary grid can be defined as a set of cuboids (rectangular cells) which are adjacent to each other and to the external boundary of the computational domain, orientated along the Cartesian coordinates. Cuboids intersected by the surface can be treated in a special way, described later in this paper, according to the boundary conditions defined on the surface. Each cuboid can be refined to 8 smaller cuboids (Figure 1) for better resolution of geometry or fluid flow singularities. It should be pointed out that the immersed-boundary grid approach can be implemented for tetrahedral and other types of elements but, in terms of numerical approximation accuracy and ease of implementation, Cartesian grids are the most preferable as they are inherently the most accurate cell type available for CFD.

As a result of using Cartesian-based grids for a given geometry, there will always be cells which are located fully in a solid body (solid cells), in fluid zones (fluid cells), and finally cells which will intersect the immersed boundary. In the simplest case, a Cartesian cell on the fluid/solid boundary consists of 2 control volumes (CV): a fluid CV and a solid CV. Within one single cell it is possible to have an arbitrary number of control volumes: 3 in case of one thin wall (fluid CV - solid

CV - fluid CV) or more in case of several layers of materials with different properties inside of a thin wall (Figure 2). This underlying philosophy of accommodating multiple CVs inside one mesh cell is what we call SmartCells. It can typically cope with 20 CVs inside one SmartCells.

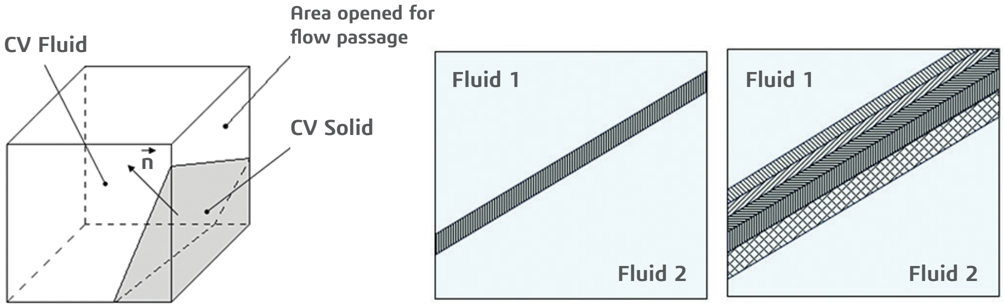


Figure 2: A "SmartCell" in the simplest case having 2 control volumes (CV) (left), with 3 control volumes (fluid-solid-fluid) in case of a thin solid wall (middle) and with 7 control volumes in the case of a thin solid wall having 5 layers with different material properties (right).

In addition to resolving two or more CVs inside one SmartCell, we have devised some unique engineering techniques over the years (eg. boundary layer treatments, thin wall treatments, thin channel treatments) that can be applied to these control volumes in order to calculate shear stresses or heat fluxes in a correct way if there is not enough grid resolution to resolve such phenomena by direct numerical modeling. These techniques will be described below. This approach of resolving the cells at the fluid/solid boundary we have called the "SmartCells" technique. Our unique approach involves a combination of fluid and solid control volumes inside one SmartCell where in order to achieve industrial levels of results accuracy, engineering methods have to be applied in addition to 3D full scale numerical modeling of continuous media in both solid and fluid zones. For each control volume all necessary geometrical parameters are calculated by extracting the corresponding data from the native CAD model. This allows us to specify all aspects of the geometry and to take the PLM data of an MCAD package into parametric CFD simulations very easily.

The SOLIDWORKS Flow Simulation SmartCells technique also includes "CAD/CFD bridge technology" which allows for good resolution of geometry features even in the case of relatively coarse meshes. Multilayer control volumes are increasingly essential for fluid flow modeling, and for heat transfer phenomena, including contact resistances and Joule heating calculations within a solid body (this being a fully-coupled multi-physics application). The solid and fluid control volumes can be alternated many times within each SmartCell (see for instance Figure 3).

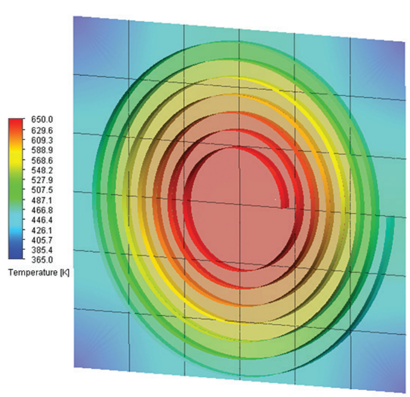


Figure 3: Multiple control volumes (solid-fluid-solid-fluid-.. etc.) for an array of SmartCells simulating a Joule Heating Coil.

SYNERGISTIC 3D MODELING & ENGINEERING TECHNIQUES IN SMARTCELLS

Within fluid regions of a SmartCell, fluid flow phenomena can be described by a system of 3-D differential equations of mass conservation of the fluid media, its momentum and energy, and turbulence characteristics. SOLIDWORKS Flow Simulation software, which is uniquely based on SmartCells techniques, is even able to consider both laminar and turbulent flows (Ref 11) in the same domain. Laminar flows occur at low values of Reynolds number. When the Reynolds number in a domain exceeds a certain critical value the flow naturally transits smoothly to turbulent flow. To simulate turbulent flows, the Favre-averaged Navier-Stokes equations are used by SOLIDWORKS Flow Simulation, where time-averaged effects of the flow turbulence on the flow parameters are considered, whereas the large-scale, time-dependent phenomena are taken into account directly. Through this procedure, extra terms known as the Reynolds stresses appear in the equations for which additional information must be provided. To close this system of equations, SOLIDWORKS Flow Simulation employs transport equations for the turbulent kinetic energy and its dissipation rate, using the modified k-E turbulence model with damping functions proposed by Lam and Bremhorst (Ref 12).

Within solid regions of a SmartCell, SOLIDWORKS Flow Simulation calculates two kind of physical phenomena: heat conduction and direct electrical current, with the resulting Joule heating being a source of heat in the energy equation. Each of these phenomena is described by an appropriate 3-D differential equation in partial differences. If a solid consists of several solid materials attached to each other in one cell, then the thermal contact resistances between them can be taken into account when calculating the heat conduction. As a result, a solid temperature step appears on the contact surfaces. The energy exchange between the fluid and solid media is calculated via the heat flux in the direction normal to the solid/fluid interface taking into account the solid surface temperature and the fluid boundary layer characteristics, and radiation heat exchange if necessary. For radiation heat exchange a set of approaches are available in SOLIDWORKS Flow Simulation ranging from Ray Tracing, also known as DTRM (Discrete Transfer Radiation Model), through Discrete Ordinates (or DO) models, to Monte-Carlo Models where the spectral properties can be taken into account. As a result of radiation calculations, the appropriate heat fluxes are taking into account in SmartCells for immersed fluid-solid boundaries or in solid cells within semi-transparent solid bodies.

The biggest issue for Cartesian immersed-body grids in CFD today is the resolution of boundary layers on coarse meshes. In most practical cases, such grids can be too coarse for the accurate solution of Navier-Stokes equations, especially within a high-gradient boundary layer. Therefore, in order to calculate skin friction and heat flux at the wall, the Prandtl approach for boundary layers is used (Ref 13). The key idea behind this approach is similar to the wall function approach used in traditional CFD codes. However, the wall treatment that forms part of the SOLIDWORKS Flow Simulation SmartCells technology uses a novel and original Two-Scale Wall Function (2SWF) approach that consists of two methods for coupling the boundary layer calculation with the solution of bulk flows and an automated hybrid approach:

- A thin boundary layer treatment that is used when the number of cells across the boundary layer is not enough for direct, or even simplified determination of the flow and thermal profiles (Figure 4);
- A thick boundary layer approach when the number of cells across the boundary layer exceeds that requirement to accurately resolve the boundary layer (Figure 4);
- In intermediate cases, the SOLIDWORKS Flow Simulation code automatically employs a compilation of the two above-mentioned approaches, ensuring a smooth transition between the two models (Figure 4).

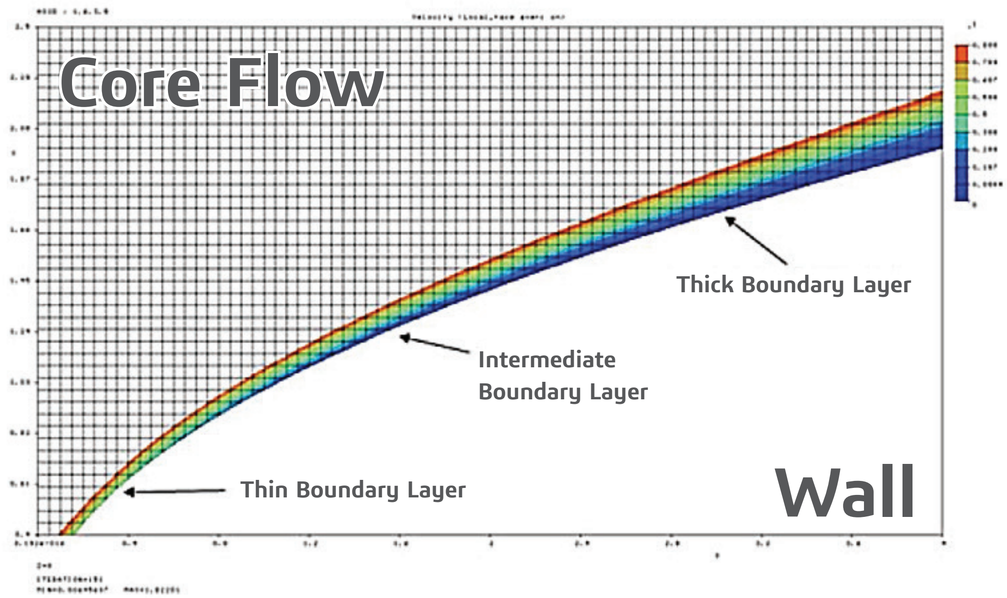


Figure 4: "Thin," "Intermediate" and "Thick" boundary layers.

Essentially such a turbulence modeling approach can be applied for dynamic as well as for temperature and concentrations boundary layers. In the thin-boundary-layer approach within SOLIDWORKS Flow Simulation, Prandtl boundary layer equations are used along fluid streamlines covering the walls (Ref 12). For their solution an integral boundary layer technology is applied (Ref 12). In the case of turbulent flows, for the determination of turbulent viscosity, the Van Driest hypothesis on the mixing length in turbulent boundary layers is used (Ref 13). The influence of wall roughness, considered as the equivalent sand grain roughness, compressibility and the external flow's turbulence on the boundary layer are modeled through semi-empirical coefficients correcting the wall shear stress and the heat flux from the fluid to the wall. From a thin-boundary-layer calculation the boundary layer thickness, wall shear stress, and the heat flux from the fluid to the wall are calculated, and are used as boundary conditions for the Navier-Stokes equations. When the number of cells across a boundary layer is sufficient, a boundary layer modification of the well-known wall functions approach is used. However, instead of the classical approach where the logarithmic velocity profile is used, SOLIDWORKS Flow Simulation uses the full profile proposed by Van Driest (Ref 13). All other assumptions are similar to the classical wall functions approach in traditional CFD software.

The incorporation of a thin-boundary-layer approach is a key element of the SOLIDWORKS Flow Simulation SmartCells technique. Another similar engineering approach is used in the modeling of fluid flow phenomena in planar thin slots or cylindrical thin channels. Use of this technology in combination with a CAD/CFD bridge brings additional benefits for resolution of flows in dedicated elements of complex models where the number of mesh cells is not enough for full 3D modeling. Having direct access to the native CAD data, the SOLIDWORKS Flow Simulation technology platform can recognize that some geometry can form flow passages as pipes or thin channels, because this information exists in the CAD system. In such cases, analytical or empirical data is used to replace the 3D Navier-Stokes equations typically needed to model within such dedicated flow passages with minimal loss of accuracy. In addition to this resolution of fluid flow phenomena via effective simplified engineering approaches in SmartCells, the approach has also been applied successfully to heat transfer phenomena in solid thin walls and even over thin multilayer structures within one cell. Usage of other engineering methods also extends SmartCell models to various electronics devices such as PCBs, 2-Resistor Models, Heat Pipes, etc. with minimal grid cell counts. Extensive validations and verifications of SOLIDWORKS Flow Simulation's underlying technologies have been done by Ivanov, et. al. (Ref 10).

INDUSTRIAL VALIDATIONS OF THE SMARTCELLS TECHNOLOGY IN SOLIDWORKS FLOW SIMULATION

Plane Fin Heat Sink Design

Plane fin heat sink elements are widely used in various electronics devices today. Usage of traditional CFD approaches requires sufficient mesh cells across each channel in the heat sink in order to get accurate simulation results. Using SOLIDWORKS Flow Simulation's SmartCells technique on a rectangular grid together with the synergy of its numerical and engineering methods and the CAD/CFD bridge, an appropriate CFD accuracy can be achieved on a relatively coarse grid. In the example below (Figure 5), the above-mentioned thin channel technology is used, where the number of cells across the channel is one to two. The SOLIDWORKS Flow Simulation calculation results for a coarse mesh (3,900 cells in total), and for a relatively fine mesh (180,000 cells in total) are compared against experimental data (from Ref 14) in Table 2. In this example SOLIDWORKS Flow Simulation's boundary layer technique, thin-wall and thin-channel engineering models work together with 3D numerical methods for Navier-Stokes equations in fluid zones and heat transfer equation in solid regions. The coarse computation grid section (Figure 5 top right) corresponds with the grid with 3,900 cells in total for this example.

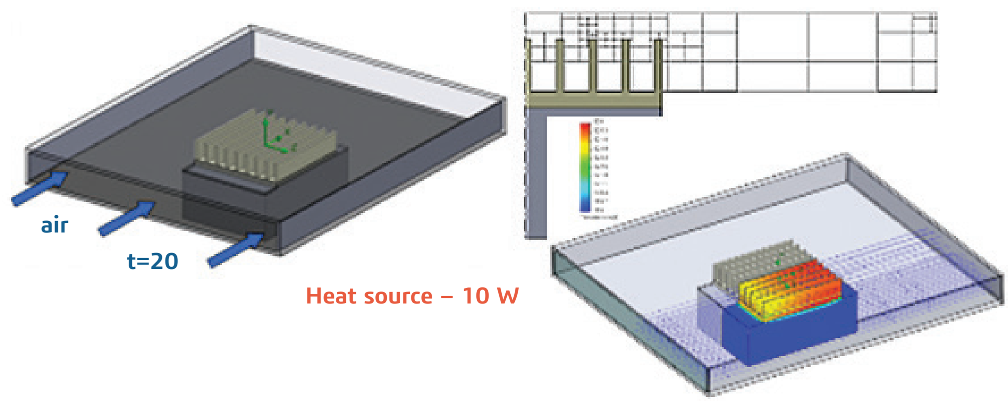


Figure 5: SOLIDWORKS Flow Simulation calculation using "Thin channel" technology for a Pin Fin Heat Sink.

Flow velocity	0.9 m/s		1.3 m/s	
$R_{t\ exp,}$ K/W	3.72		3.20	
Cells number	3,900	180,000	3,900	180,000
$R_{t\ calc,}$ K/W	3.714	3.77	3.213	3.22
δ, %	0.2	1.3	0.4	0.6

Table 2. SOLIDWORKS Flow Simulation calculation results for the Pin Fin Heat Sink example for coarse and fine mesh sizes versus experimental measurements.

It is clear from this example that SOLIDWORKS Flow Simulation can generate accurate results with thousands, tens of thousands and low millions of Cartesian cell counts when compared to multi millions of cells typically necessary for the same level of accuracy in traditional CFD codes because of the embedded technologies inherent in SmartCells.

ASMO Automotive External Aerodynamics Benchmark

The ASMO (Aerodynamisches Studien Modell) car body calculation with SOLIDWORKS Flow Simulation is shown below and its SOLIDWORKS Flow Simulation simulation prediction is compared with experimental data (Ref 7). This wind tunnel model was created many years ago by Daimler-Benz for the investigation of car configurations with very low drag coefficient and for testing different CFD tools against it. The 3D ASMO model is shown in Figure 6.

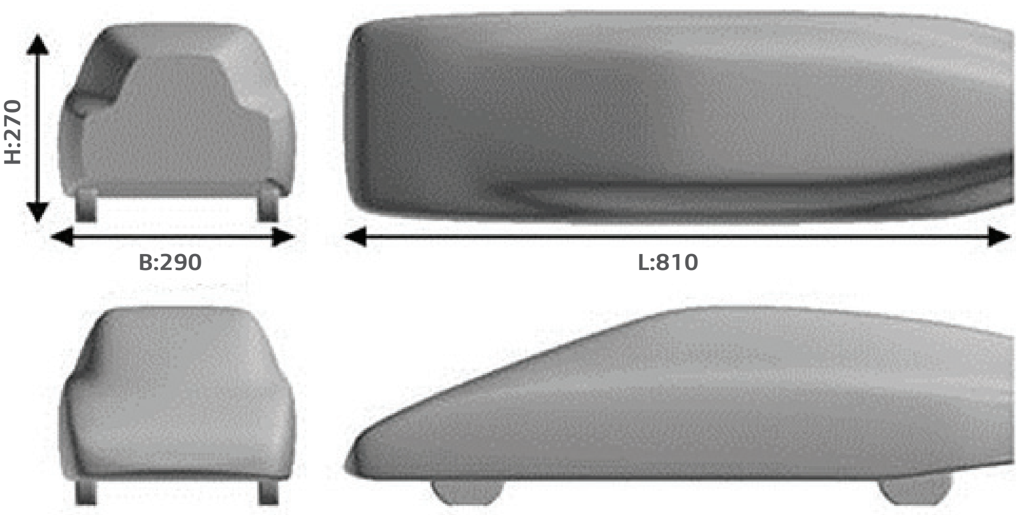


Figure 6: ASMO model geometry (dimensions are in mm).

The SOLIDWORKS Flow Simulation CFD calculations were done for an oncoming air flow speed of 50 m/s. During the simulation, another automated SOLIDWORKS Flow Simulation technology for "adaptive grid refinement" for flow singularities was used (Figure 7). The initial SmartCells computational grid consisted of 200,000 Cartesian cells with the final adapted mesh being about 2 million grid cells when the technology of adaptive grid refinement were applied for high flow gradient regions.

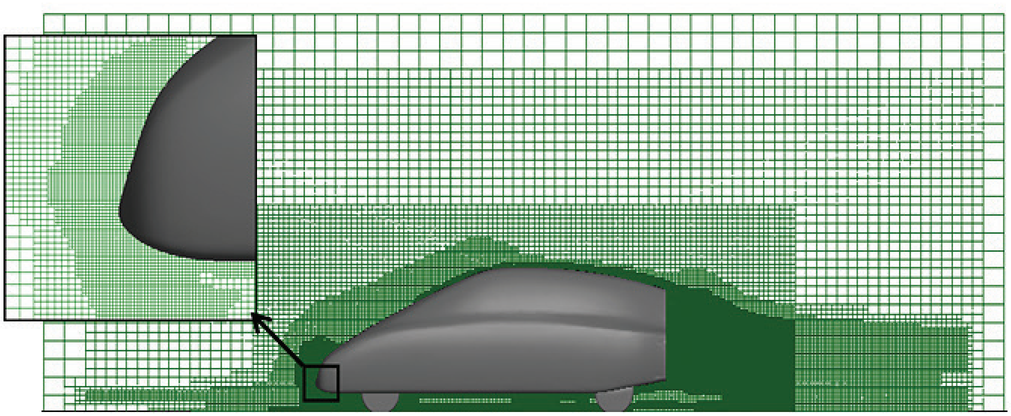


Figure 7: Final SOLIDWORKS Flow Simulation SmartCell grid for the ASMO validation model.

It can therefore be seen that the synergy of 3D numerical methods within the engineering technique of boundary layer resolution in SmartCells can achieve good accuracy without the detailed resolution of the boundary layer by large numbers of computational grid cells (Figure 7). In addition, significant computational resources are saved that would have to be deployed by traditional CFD methodologies. Pressure coefficient distributions on the ASMO model surface are shown in Figure 8 together with experimental data (Ref 7). The value of drag coefficient obtained by SOLIDWORKS Flow Simulation was 0.158, whereas the experimental data from Volvo experiments was 0.158 and from Daimler Benz was 0.153, highlighting the good agreement between predictions and wind tunnel data of this approach.

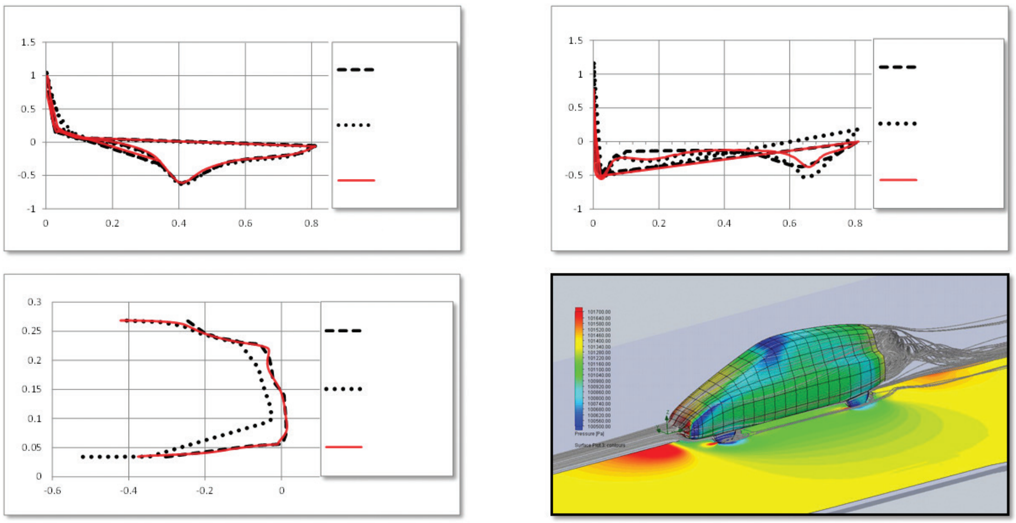


Figure 8: Comparisons of Cp calculated by SOLIDWORKS Flow Simulation with experimental data and depiction of the overall flow field.

The above SmartCells approach employed uniquely by SOLIDWORKS Flow Simulation has been used for solution of various tasks in the automotive industry like LED lighting applications (Ref 6, 15), internal combustion engine applications (Ref 11), and others. Such technologies can be extended to much more sophisticated physical models like cavitation (Ref 16) as well.

Finally, it should be noted that the comparison of the SmartCells technique realized in SOLIDWORKS Flow Simulation with many well-known traditional CFD tools in a 2013 Japanese Society of Automotive Engineers (JSAE) automotive external aerodynamic blind benchmark (Ref 17) demonstrated SOLIDWORKS Flow Simulation's out-of-the-box robustness and accurate turbulence model with significantly lower cell count versus traditional CFD approaches with similar or better results.

CONCLUSIONS

The biggest barrier to user productivity with industrial CFD tools today is dealing quickly and effectively with complex CAD geometries by generating usable meshes within realistic engineering timescales. Efficient usage of CFD tools embedded into CAD systems (like SOLIDWORKS Flow Simulation) requires the development of special engineering models that allow for rapid, robust and accurate solutions. Such approaches can be realized by the usage of a technology based on the synergy of numerical and engineering methods applied to solutions of fluid dynamic and heat transfer tasks on rectangular adaptive grids - the SmartCells technique. SOLIDWORKS Flow Simulation software based on this technology is showing high levels of accuracy in an efficient, practical tool for both CFD experts and design engineers to solve various tasks for many industries. SmartCells can mesh complex geometries in seconds and minutes versus hours, days and even weeks for traditional CFD meshing approaches. The approach can be applied to different stages of manufacturing design cycles and it allows for the optimal situation of frontloading of simulation technologies in order to keep up with global product manufacturing competition. It also solves the age-old "Achilles heel" of CFD, the time-consuming and specialized nature of mesh generation, thus paving the way for the democratization of CFD usage.

REFERENCES

1. Sobachkin A.A., Dumnov G.E. Numerical Basis of CAD-Embedded CFD. Proceedings of NAFEMS World Congress NAFEMS World Congress, Austria, Salzburg, June, 2013.
2. Hanna, R. K., Parry, J., "Back to the Future: Trends in Commercial CFD", NAFEMS World Congress, Boston, USA, June, 2011
3. Boysan, H.F., Choudhury, D. & Engelman, M.S., "Commercial CFD in the Service of Industry: The First 25 Years" in 'Notes on Numerical Fluid Mechanics and Multidisciplinary Design', Volume 100, 2009, pp. 451-461
4. Spalding, D. B., "CFD Past, Present and the Future", Lecture at the Sixteenth Leont'ev School-Seminar Saint Petersburg, Russia, May 21-25, 2007, http://www.cham.co.uk/Docs/CFDeng3.ppt
5. Launder, B.E.; Spalding, D.B., "The numerical computation of turbulent flows". Computer Methods in Applied Mechanics and Engineering. 3 (2): 269–289, March 1974. doi:10.1016/0045-7825(74)90029-2
6. Marovic Boris. "The New Role of CFD in the Ever Faster Development Cycle and the Struggle with Complex Geometries of the Lighting Industry." ISAL 2013 (Darmstadt, Germany), Sept. 2013.
7. Dumnov, G., Kharitonovich, A., Marovic, B.; and Sobachkin A., "Simulation Time Saving Approach Based on the Synergy of Numerical and Engineering Methods for Experts and Design Engineers", FISITA Automotive Congress, September 2016, Busan, South Korea.
8. Sabeur, M., "Frontloading CFD in the Automotive Development Process", NAFEMS European Conference, December 2015, Munich, Germany.
9. Weinhold, I., Parry, J., "The Third Wave of CFD", NAFEMS World Congress, Austria, Salzburg, June, 2013.
10. Ivanov A.V., Trebunskikh T.V., Platonovich V.V. "Validation Methodology for Modern CAD-Embedded CFD Code: From Fundamental Tests to Industrial Benchmarks." Proceedings of NAFEMS World Congress NWC 2013, Austria, Salzburg, June 09-12, 2013.
11. Uppuluri Sudhindra, Proulx Joe, Marovic Boris, Naiknaware Ajay. "Characterizing Thermal Interactions between Engine Coolant, Oil and Ambient for an Internal Combustion Engine." SAE World Congress 2013 (Detroit, MI, USA), April 2013.
12. Lam C.K.G., Bremhorst K.A. "Modified Form of Model for Predicting Wall Turbulence." ASME Journal of Fluid Engineering, 1981, 103: 456-460.
13. Van Driest E.R. "On Turbulent Flow near a Wall." Journal of the Aeronautical Science, 1956, 23(10): 1007.
14. Jonsson, H. and B. Palm, 1998, "Thermal and Hydraulic Behavior of Plate Fin and Strip Fin Heat Sinks under Varying Bypass Conditions", Proc. 1998 Inter. Society Conf. on Thermal and Thermomechanical Phenomena in Electronic Systems (ITHERM ´98), IEEE, pp. 96-103, ISBN 0.7803-4475-8.
15. Watson J.C., Dumnov G., Ivanov A., Muslaev A., Popov M. "Evaluating Water Film and Radiation Modeling Technologies in CFD for Automotive Lighting." Proceedings of NAFEMS World Congress NWC 2015 and the 2nd International SPDM Conference, San Diego, CA, USA, June 21-24, 2015.
16. Dumnov Gennady, Muslaev Alexander; Streltsov Viatcheslav; Marovic Boris. "Cavitation Process Simulation for Automotive Applications with an Isothermal Solver Approach." SAE World Congress 2013 (Detroit, MI, USA), April 2013.
17. Nakashima Takuji, Sasuga Nobuhiro, Ito Yuichi, Ikeda Masami, Ueda Iichiro, Kato Yoshihiro, Kitayama Masashi, Kito Kozo, Koori Itsuhei, Koyama Ryutaro, Shimada Yoshihiro, Hanaoka Yuji, Higaki Tatsuhiko, Fukuda Kota, Yamamura Jun, Li Ye. "Benchmark of Aerodynamics CFD of Simplified Road Vehicle Model." Japan, JSAE, Paper Number: 20134343, 2013: 8-28.

SIM Technologies Pvt. Ltd.

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

+91-8754447021 **marketing@simtek.in**