A 3D CFD simulation of a train nose, showing a pressure contour plot. The train nose is rendered in a grid of green and blue cells. The surrounding air is colored in a gradient from yellow to red, indicating pressure variations. A vertical color bar on the left side of the image shows the pressure scale in Pascals (Pa), ranging from 100500.00 (blue) to 101700.00 (red). The text 'SIEMENS Ingenuity for life' is overlaid on the top left. The text 'Siemens PLM Software' is overlaid on a dark blue box. The main title 'Frontloading CFD – Required technologies' is overlaid on a large teal box. The authors' names are listed at the bottom.

SIEMENS

Ingenuity for life

Siemens PLM Software

Frontloading CFD – Required technologies

Dr. M. Sabeur, DIPL.-ING. M. Gruetzmacher,
Dr. K. Hanna and Dr. A. Sobachkin

Abstract

Today manufacturing product design cycles need to get shorter and shorter as either new or increased numbers of products get to the market faster. In the automotive industry for example, with either facelifts to existing car models, or the next generation of the model appearing almost every three to four years, and an increasing number of new models appearing on the market, the demand on engineering design centers is to produce the same or higher quality products in a shorter time. With engineering simulation technology, increasing product complexity can be designed much closer to its limits and therefore overdesign can be reduced – but it has a knock-on effect of more and more requests for computer-aided engineering (CAE) simulations by stretched simulation experts.

Frontloading of all sorts of simulations, including computational fluid dynamics (CFD), into the design department is the “gold standard” with the highest ROI, and it is playing an increasing role while being accepted widely, especially in the automotive industry.

Such frontloaded CFD simulations need to be done directly on the latest 3D computer-aided design (CAD) model and need to provide results in hours rather than days or weeks. The inherent higher order numerical nature of CFD simulation tools can make them very complicated and demand a deep understanding of both the physics and numerical algorithms involved. The utilization of well-known CFD methodologies such as the finite volume method (FVM) for solving the underlying Navier-Stokes equations are generally required but facilitating technology enablers are actually the synergies between numerical, engineering techniques and analytical methods. These provide the foundation to reducing the numerical skill requirements and the manual time spent on such tools for meshing a given geometry, not only for engineers but also for analysts.

Efficient use of CFD tools embedded into CAD systems requires the development of special approaches. These approaches should allow for the solving of tasks based on complex CAD geometry. By applying analytical methods to numerical tasks, physics describing the behavior of a fluid boundary layer for example can be used to reduce the required skills for building high accuracy meshes and the manual time spent on the geometry and grid generation task by an order of magnitude. A relatively coarse CFD mesh can be applied via this approach and the sub-mesh physics can be overcome with the implementation of analytical data. As a result, the considered approach enables a reduced cell count compared to meshes with thin boundary layers being meshed. This approach also enables the use of an automated Cartesian mesh with Cartesian-based polyhedral cells at the fluid-solid interface as well as solid-solid interface, thus eliminating the laborious manual work of creating a fine boundary layer mesh. Instead, it enables the automation of the meshing process completely with very low numerical skills or time requirements for the engineer or the analyst.

The SmartCells™ based frontloading CFD approach will be elaborated in this paper. It has proven successful in a wide range of applications and is applied by OEMs and Tier 1 suppliers in the automotive, aerospace and other industries. Several benchmark examples will be presented to illustrate the approach. The methodology described here ultimately enables the use of CFD in the design process by CAD engineers and automates tedious CFD meshing by an order of magnitude within common CAD environments and product lifecycle management (PLM) workflows without compromising the resulting accuracy compared to typical traditional CFD solutions even with coarse meshes.

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 around \$1 billion per year in revenues worldwide today (Hanna & Parry 2011). The bulk of CFD simulation (over 90 percent) 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. CFD is based on well-accepted numerical methods for solving the fundamental Navier-Stokes equations that govern fluid flow, heat and mass transfer (Boysan, Choudhury and Engelman, 2009). But the technology enablers for the CFD industry invariably are a synergy between numerical and engineering techniques and analytical methods that are 30 to 40 years old (Spalding, 2007). Indeed, the vast majority of CFD carried out in the world today is based on variants of the tried and-proven $k-\epsilon$ turbulence model, the accepted workhorse of the industry that is now over 40 years old (Launder & Spalding, 1974).

Users of CFD simulation tools for the first time can find them very difficult to use because the user has to master very complicated preprocessing (geometry and grid generation) approaches and frequently the codes themselves demand of users a deep understanding of the physics and numerical algorithms underlying them because of their inherent mathematical nature. And invariably the quality of a CFD prediction is very much affected by the preprocessing 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 approach described in this white paper enables the use of CFD in design processes by CAD engineers and experts alike through the automation of the pivotal meshing task without compromising the final result accuracy even with coarse meshes compared to typical CFD meshes. An efficient simulation approach has to 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. This approach is remarkably robust and can be employed by CFD analysts as well as designers allowing them to “frontload” their CFD simulations (Eigner, 2010; Marovic, 2013; Sabeur M., 2015; Dumnov G. et al., 2016) in order to yield maximum simulation productivity at least time cost in a product manufacturing workflow.

The top challenge for design engineers in the world is ever increasing product complexity. Many manufactured products have evolved into complex systems of mechanical components, electronics and software, involving multiple engineering disciplines. In addition, the increasing number of components, often combined with miniaturization, requires an even greater understanding of how these components will interact, while making sure they do not overheat. To add yet another level of complexity, products are often offered in multiple configurations, and design engineers must understand the performance of each configuration.

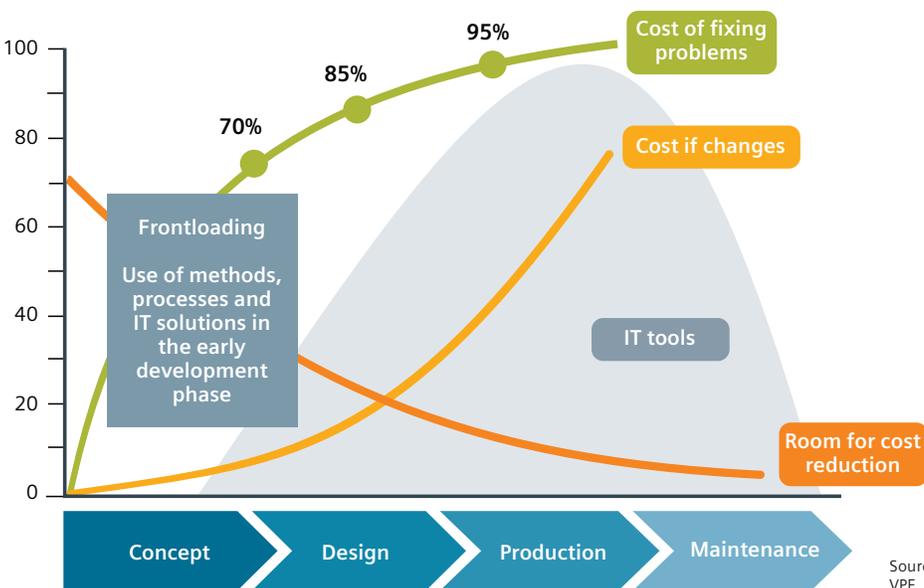
Frontloading CFD – Why?

Figure 1 is based on work by Prof Martin Eigner (2010) and it shows that the earlier that simulation can be done in a design process, the lower the cost of change is and the higher room for cost reductions, hence the biggest return on investment for such tools. He calls this “frontloading” for all sorts of software simulation tools.

Surely CFD is a mature technology and it is used extensively throughout a product’s design cycle workflow today. So, why try to frontload it? Well, the reality is that many industry analysts reckon that CFD is only being used by 5 to 10 percent of the users who could and should be using CFD today in the world. Traditional CFD approaches (that account for ~85 percent of all CFD in the world) are very time consuming and frequently prohibitive for practical design cycle workflows especially in the early design stage because a user may be dealing with:

1. Complex geometries that may need simplification
2. Time scales that are too short – traditional CFD approaches cannot be automated and/or require intense user implementation

3. Risks because of over-simplifications in a geometry to save time: it may not map onto the “real world” properly
4. Complex geometries that require manual meshing by an expert
5. Traditional CFD simulations are only done by experts (who are in short supply) because of the legacy tools a company may have
6. The CFD analysis team in a company is overloaded because of long turnaround times and product engineering workflows can therefore be delayed
7. Expensive CFD experts frequently are required to conduct routine simulation work such that they have no time for innovation
8. Different internal systems for CFD analysis and design analysis and a need for data transfer between them (other than “throw it over the wall”)
9. Transferring geometry to CFD and back can have long turnaround times
10. Hard to track design variations: design changes are not synchronized due to simulation delays



Source: Prof. Dr. Martin Eigner, VPE, TU Kaiserslautern

Figure 1: Frontloading economical value (Eigner, 2010).

Frontloading provides the best environment for simulation-centric CFD. This is similar to what was called upfront CFD except that here we are talking about CAD-embedding of CFD which has benefits throughout a product’s manufacturing process. The key thing is to simulate early inside and CAD and frequently to have the most impact on the product’s design (figure 2).

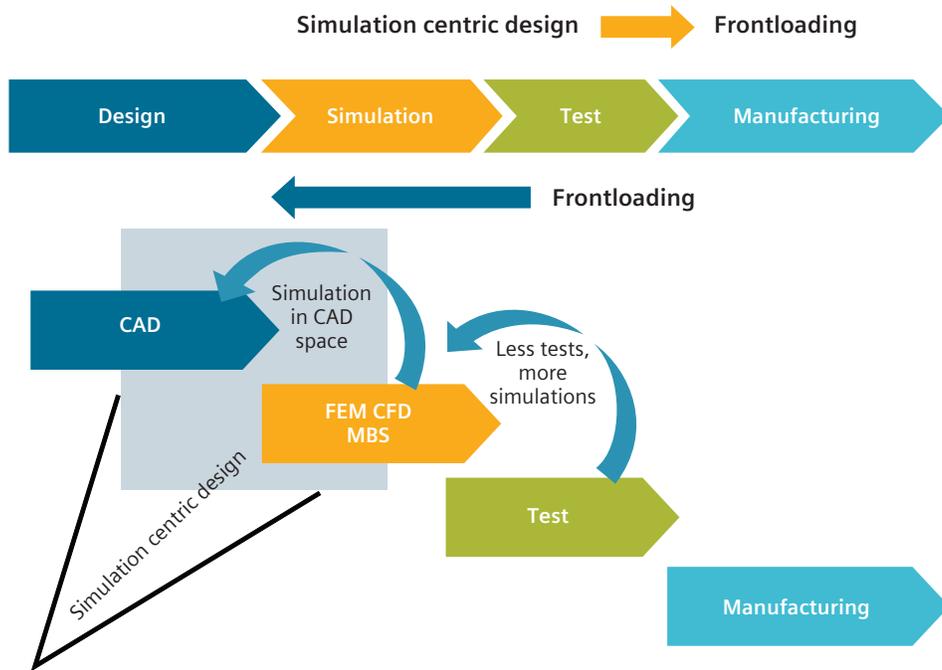


Figure 2: CAE centric design – CAE frontloading (Sabeur, 2015).

Frontloading CFD – Enabling technologies

To understand Frontloading of CFD requires first an understanding of a traditional CFD process where the CFD code is separate from the CAD package from which it gets its geometry. All CFD simulations require you to deal with a CAD model, geometry preprocessing (usually CAD clean up and repair), meshing, solving, post-processing and reporting (figure 3). It also illustrates the sequential (and time consuming) process for a traditional CFD simulation (LHS) and a frontloading CFD approach (RHS). The former approach requires steps inside and outside of a CAD package and repeated return to the CAD tool with inherent risks of geometry approximations coming into the CFD simulation. CAD-embedded CFD by comparison is all within the native CAD and it changes as the geometry changes inside the one CAD environment.

Hence, to produce an efficient CFD approach requires CAD-embedding and several pieces of technologies to work well together. The main component elements of these frontloading technologies are:

Geometry

It is essential to note that all major CAD systems were created 20 to 30 years ago and were optimized as design tools and only later the necessity for embedding CAE (and in particular CFD) was 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 (Eigner, 2010; Marovic, 2013; Dumnov G. et al., 2016; Sabeur, 2015; Weinhold & Parry, 2013). The biggest obstacle to achieving this is the high level of human resource requirements for performing CFD calculations when 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.

The most efficient CFD approach we believe is based on two 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

Geometry preparation, which includes model cleaning and healing to reduce complexity or for closing of gaps (if necessary), is done within the CAD system. A reasonable timeframe for geometry processing ideally should not exceed a few hours.

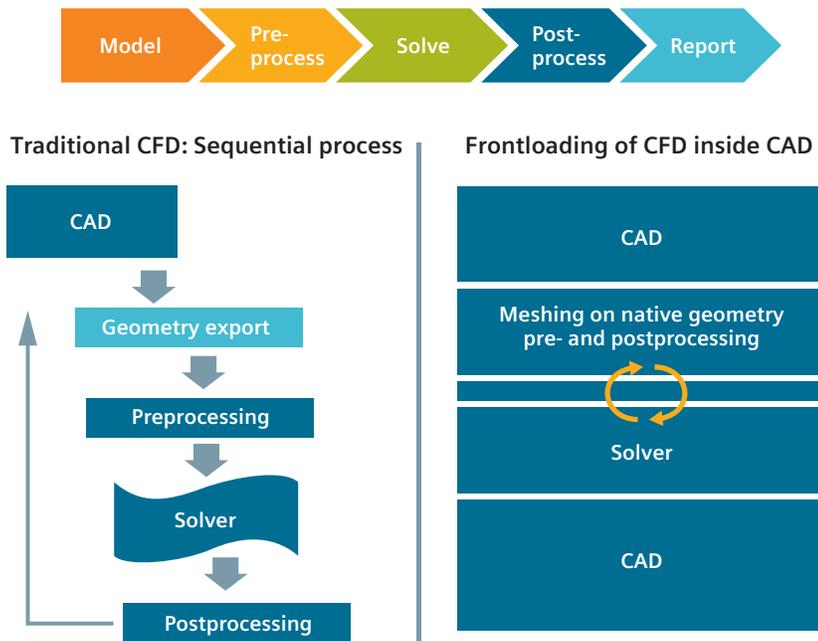


Figure 3: The CAE simulation process (Sabeur, 2015).

Mesh	Unstructured triangular		Structured triangular		Structured curvilinear		Cartesian [Aftosmis]		Simcenter FLOEFD SmartCells	
Results	Cells	$ LTE _{L1}$	Cells	$ LTE _{L1}$	Cells	$ LTE _{L1}$	Cells	$ LTE _{L1}$	Cells	$ LTE _{L1}$
	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 the numbers of cells of CFD mesh types.

(Marovic B., 2013)

Meshing

Typically in traditional finite volume CFD, meshing types used tend to include unstructured triangular, structured triangular, structured curvilinear as well as immersed boundary Cartesian meshes (Marovic, 2013). Table 1 shows a mathematical formulation of CFD simulation accuracy, $||LTE||_{L1}$, related to the various CFD meshing approaches considered (the lower the LTE number the more accurate the CFD prediction is).

What we call “SmartCells” technology in the CAD-embedded general purpose CFD code, Simcenter FLOEFD™ software, 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 unstructured triangular, structured triangular and structured curvilinear meshes. This obviously has both a substantial memory and a CPU saving associated with it. These savings are made possible in part due to the numerical methods employed and to Cartesian cells not suffering from skewness, hence the name SmartCells.

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 postprocessing overheads that comes with these large models. And 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 to hand in a company, and more besides. 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 (Sobachkin & Dumnov G, 2013).

This paper contends that there is another approach to industrial-level RANS CFD that is both smarter, computationally more efficient, just as effective, well validated, but uses orders of magnitude fewer cells, and therefore uses much less computational resource for the same level of accuracy as traditional CFD approaches. And it is also embedded within CAD and PLM workflows which is intuitively the most optimal place for CFD simulation to be thus enhancing user productivity inside their familiar CAD/PLM interface. In engineering design simulation practice today, whatever the industry, CAD/PLM concepts (such as Dassault Systèmes CATIA and SolidWorks, PTC Creo, Autodesk Inventor, Siemens Solid Edge® software and Siemens NX™ software) are widely deployed by engineers as the

means by which 3D manufactured product data are used and maintained consistently during an entire product's lifecycle and across all its design changes (Hanna & Parry, 2011).

The basis of the PLM concept is the availability of complex 3D product model data within a mechanical CAD system as its central element. 3D product model data are 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.

This synergy of CAD and 3D numerical CFD models is a critical element which allows to reduce resource requirements on grid generation and numerical solution stages by an order of magnitude. It can simplify getting 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 (Marovic, 2013). 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 4 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 eight smaller cuboids (figure 4) for better resolution of geometry or fluid flow singularities. It should be pointed out that the immersed body 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 two 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, three in the case of one thin wall (fluid CV-solid CV-fluid CV) or more in the case of several layers of materials with different properties inside of a thin wall (figure 5). Simcenter FLOEFD can typically cope with 20 control volumes inside one Cartesian SmartCell™.

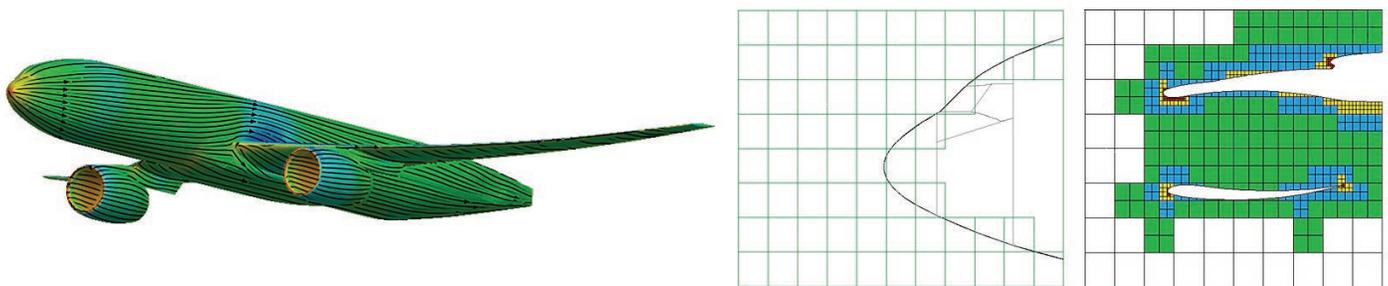


Figure 4: Aircraft CFD flow surface streamlines and Cartesian immersed-body grid without grid cell refinement (middle) and with grid cell refinement (right) using SmartCells.

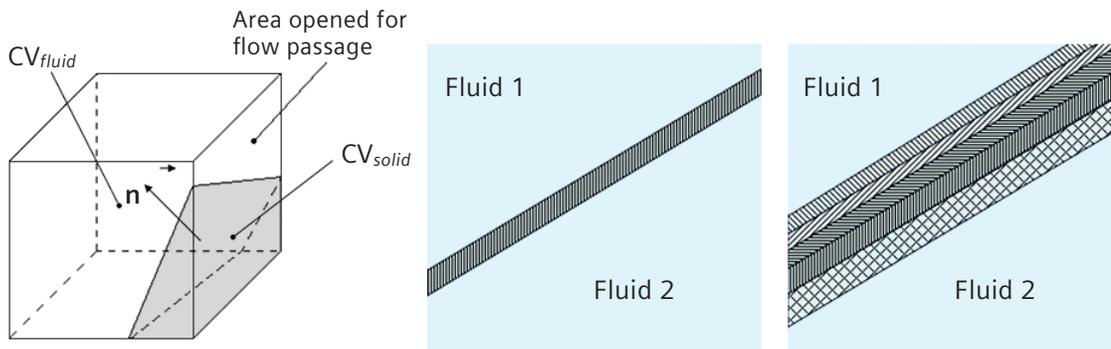


Figure 5: SmartCell with two control volumes (CV) (left), with three control volumes (fluid-solid-fluid) in case of a thin solid wall (middle) and with seven control volumes in the case of a thin solid wall having five layers with different material properties (right).

This approach to CFD simulations will lead to a coarser mesh being applied to a given application to capture the physical phenomena being resolved (for example, turbulence vortices, thin channels, etc.) due to the implementation of more modern engineering data approaches (Hanna & Parry, 2011). As a result, this approach enables a reduced cell count for a CFD simulation compared to CFD meshes 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. Typically, an appropriate time frame for the automatic meshing is normally a few minutes or hours.

Turbulent boundary layer simulation and multiple fluid-solid control volumes

Within fluid regions of a cell, fluid flow phenomena can be described by a system of 3D differential equations of mass conservation of the fluid media, its momentum and energy and turbulence characteristics. Simcenter FLOEFD software for instance, which is based on these techniques, is even able to consider both laminar and turbulent flows (Uppuluri et al., 2013) 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 solver naturally transitions smoothly to turbulent flow.

To simulate turbulent flows, the Favre-averaged Navier-Stokes equations are used, 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, transport equations for the turbulent kinetic energy and its dissipation rate, using the modified k- ϵ turbulence model with damping functions proposed by Lam and Bremhorst (1981) can be employed.

Within solid regions of a cell, two kind of physical phenomena can be calculated, heat conduction and direct electrical current, with 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 can be 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 ranging from Ray Tracing, also known as discrete transfer radiation model (DTRM), through discrete ordinates (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 can be taking into account 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 in Simcenter FLOEFD (Van Driest, 1956). The key idea behind this approach is similar to the wall function approach already used in CFD codes. However, the wall treatment that forms part of the described 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:

1. 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 6)
2. A thick boundary layer approach when the number of cells across the boundary layer exceeds that requirement to accurately resolve the boundary layer
3. In intermediate cases, the code automatically employs a compilation of the two above-mentioned approaches, ensuring a smooth transition between the two models

Essentially such a turbulence modeling approach can be applied for flow as well as for temperature and concentrations boundary layers. In the thin-boundary-layer approach, Prandtl boundary layer equations are used along fluid streamlines covering the walls and for their solution an integral boundary layer technology is applied (Lam & Bremhorst, 1981). In the case of turbulent flows, for the determination of turbulent viscosity, the Van Driest hypothesis (1956) on the mixing length in turbulent boundary layers is used. 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 that correct 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 can be 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, the full profile proposed by Van Driest (1956) is used. All other assumptions are similar to the classical wall functions approach in CFD software. The incorporation of a thin-boundary-layer approach is a key technique of this frontloading CFD approach via SmartCells.

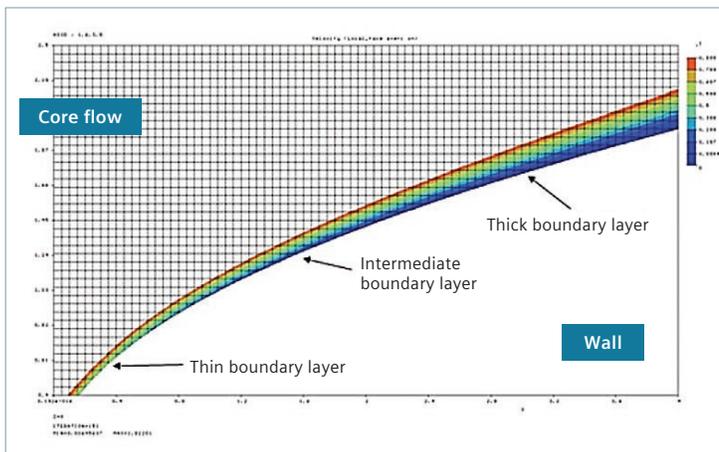


Figure 6: Thin, intermediate and thick boundary layers.

Solving and results processing

An automatic and robust convergence behavior is essential for an efficient CFD simulation process. The described approach for Simcenter FLOEFD has an implicit flow solver for incompressible and low compressible flows, an explicit solver for high Mach Number and hypersonic flows and a hybrid solver for liquid flows with cavitation, thus allowing for both high simulation efficiency and high accuracy of the technology. The first CFD run usually converge without additional numerical diffusion, it avoids multiple re-runs and allows multiple variant scenarios.

Parametric studies inside Simcenter FLOEFD within CAD packages enable early design variant analysis and what-if analysis without further definition of model data. The full support of product CAD configurations or instances allows users to modify the design and run automatically the simulation. Multiple parametric simulations result in optimum design.

The engineering outputs from this CFD solver appear in both a timely and intuitive manner, including reports in Microsoft Excel and Microsoft Word, which most engineers are familiar with. This combination of good performance for relatively coarse meshes, CAD-embedded capability, and a high level of automation and usability regarding the model set up, meshing and solution ensures an effective fast and robust CFD analysis.

Today's product development processes inside CAD involve several areas of simulations, for example structural analysis, heat transfer and moving body analysis. With the described CFD approach, pressure and temperature loads can be exported relatively easily which avoids export and import manual efforts, interface problems and attendant approximation errors.

CAD-embedded CFD also offers a unique and cost effective approach to characterising components for 1D system simulation applications. As an alternative to physical test bench methods, the CFD tool can simulate complex flows in CAD geometries. The highly 3D interacting nature of the flow can be resolved, without having to resort to assumptions or text book relationships. A simulation-based characterization (SBC) workflow can therefore be used to characterize complex 3D geometries for subsequent use within a 1D simulation code.

A typical time frame for solving and results processing in CAD-embedded CFD is a few hours. For more complex applications and several design points within a

parametric CAD study or for optimization studies may take days, but there is no additional time required for the engineer to control the simulation.

Additional empirical engineering models

Another engineering approach within Simcenter FLOEFD 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 Simcenter FLOEFD technology platform is programmed to 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, the Simcenter FLOEFD code has also been applied successfully to heat transfer phenomena in solid thin walls and even over thin multilayer structures within one computational cell. Use of other engineering methods also extends these models to various electronics devices such as PCBs, 2-resistor models, heat pipes, etc. with minimal grid cell counts. Extensive validations and verifications of these underlying technologies have been done by Ivanov et al (2013).

The underlying SmartCell philosophy of accommodating multiple control volumes (CVs) inside one mesh cell can typically cope with 20 CVs inside one Cartesian cell element. In addition to resolving two or more CVs inside one cell some unique engineering techniques have been developed over the years (for example 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 accurately if there is not enough grid resolution to resolve such phenomena by direct numerical modeling.

This Simcenter FLOEFD approach to CFD involves a combination of fluid and solid control volumes inside one cell 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 because the code is CAD-embedded.

The basis of modern PLM software is the availability of complex 3D product model data within a mechanical CAD system as its central element. 3D product model data in the PLM package are therefore both the foundation and starting point for all virtual prototyping and physical simulations today. The performing of fluid flow simulations using CFD in such a CAD-embedded context is very attractive as it can accelerate the design process and make it more predictable and reliable. It also allows for the pushing of CFD simulation to the front of the design process thus providing the biggest return on investment.

This approach allows for the specification of all aspects of the geometry and takes the PLM data of a MCAD package into parametric CFD simulations very efficiently. The above described techniques also include 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 cell (see for instance figure 7).

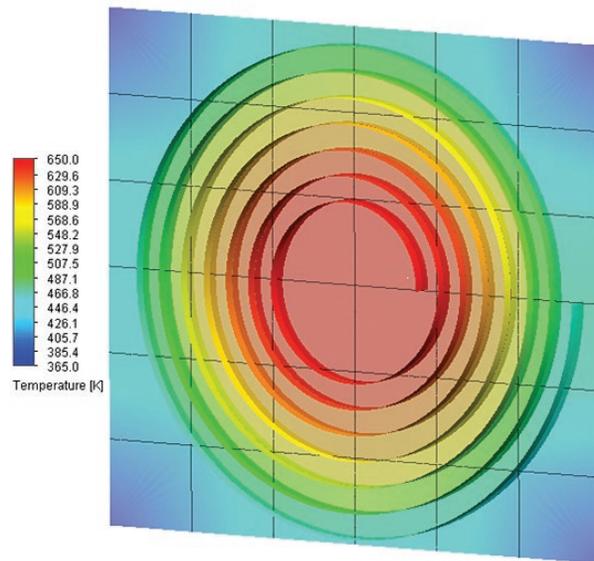


Figure 7: Multiple control volumes (solid-fluid-solid-fluid-, etc.) for simulating a Joule heating coil.

Industrial examples of SmartCells and frontloading CFD

Plane fin heatsinks

These elements are widely used in various electronics devices today. Using the described 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 above (figure 8), thin channel technology is used, where the number of cells across the channel is one to two. The 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 (Jonsson & Palm, 1998) in table 2. In this example the 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 8 top right) corresponds with the grid that has 3,900 cells in total for this example (table 2).

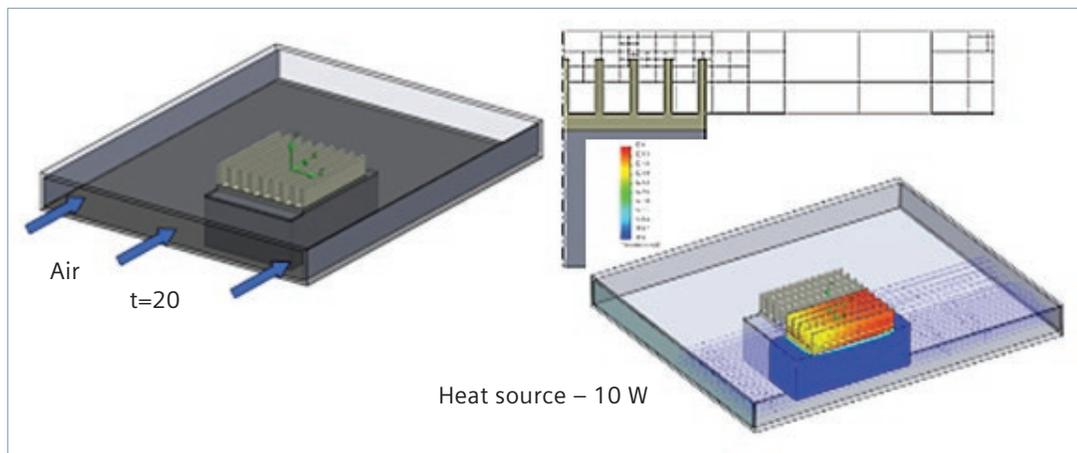


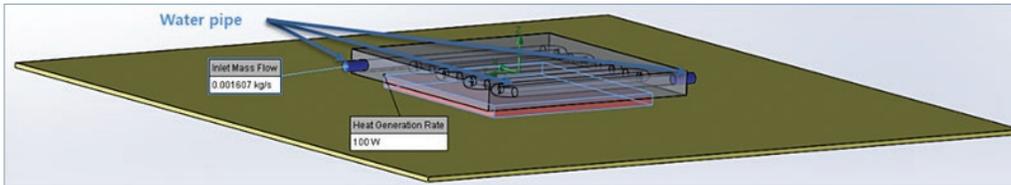
Figure 8: Thin-channel technology for a pin fin heatsink.

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: Simcenter FLOEFD calculation predictions for the pin fin heatsink example for coarse and fine mesh versus experimental measurements.

Cold plate heat exchanger with thin channels

This example illustrates the thin channel technique used by Simcenter FLOEFD:



Pipe diameter: 4 mm
Water mass flow: 0.0016 kg/s
Water comes in with 20° C
Heat source: 100 W

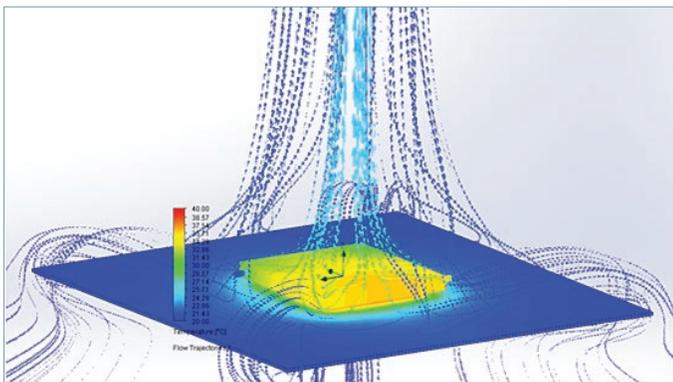
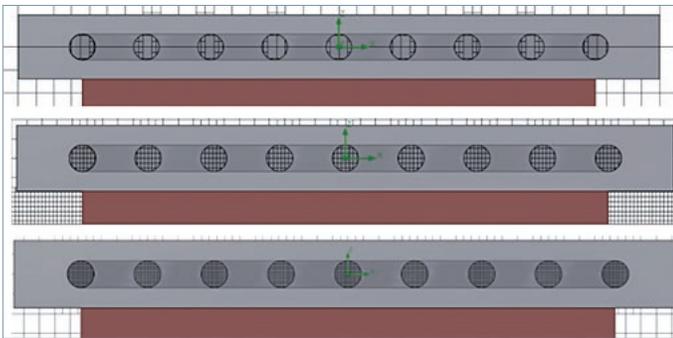


Figure 9: Water pipe heat exchanger.

ASMO automotive aerodynamics benchmark

The Aerodynamisches Studienmodell (ASMO) car body calculation is shown below and its simulation prediction is compared with experimental data (Dumnov et al., 2016). 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 10.

Simcenter FLOEFD CFD calculations were done for an oncoming air flow speed of 50 m/s. During the simulation, another automated technology of adaptive grid refinement for flow singularities was used (figure 11). The initial computational grid consisted of 200,000 Cartesian cells with the final adapted mesh being about 2 million grid cells when adaptive grid refinement was applied to high flow gradient regions.

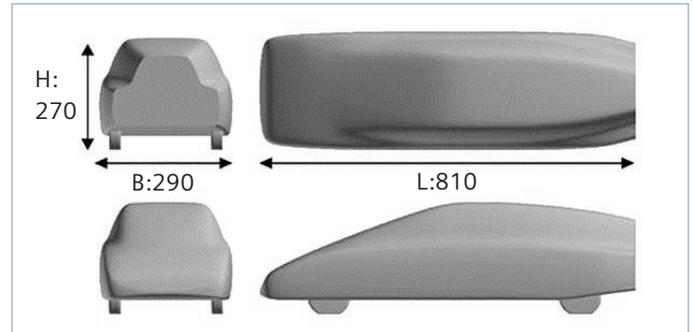


Figure 10: ASMO model geometry (dimensions are in mm).

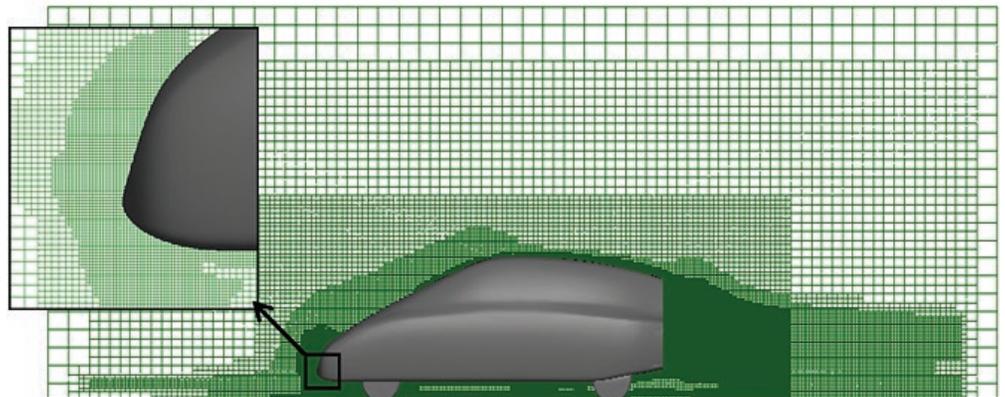


Figure 11: Final SmartCell grid for the ASMO validation model.

It can therefore be seen that the synergy of 3D numerical methods within the engineering technique of boundary layer resolution can achieve good accuracy without the detailed resolution of the boundary layer by large numbers of computational grid cells (figure 12). In addition, significant computational resources are saved. Pressure coefficient distributions on the ASMO model surface are shown in figure 12 together with experimental data (Dumnov et al., 2016). The value of drag coefficient obtained was 0.158 whereas the experimental data from Volvo experiments was 0.158 and from

Daimler Benz was 0.153 highlighting good agreement between Simcenter FLOEFD predictions and wind tunnel data using this approach.

The described approach has been used for the solution of various tasks in the automotive industry like LED lighting applications (Marovic, 2013; Watson et al., 2015), internal combustion engine applications (Ivanov et al. 2013), etc. Such technologies can also be extended to more sophisticated physical models like cavitation and condensation (Watson et al., 2015).

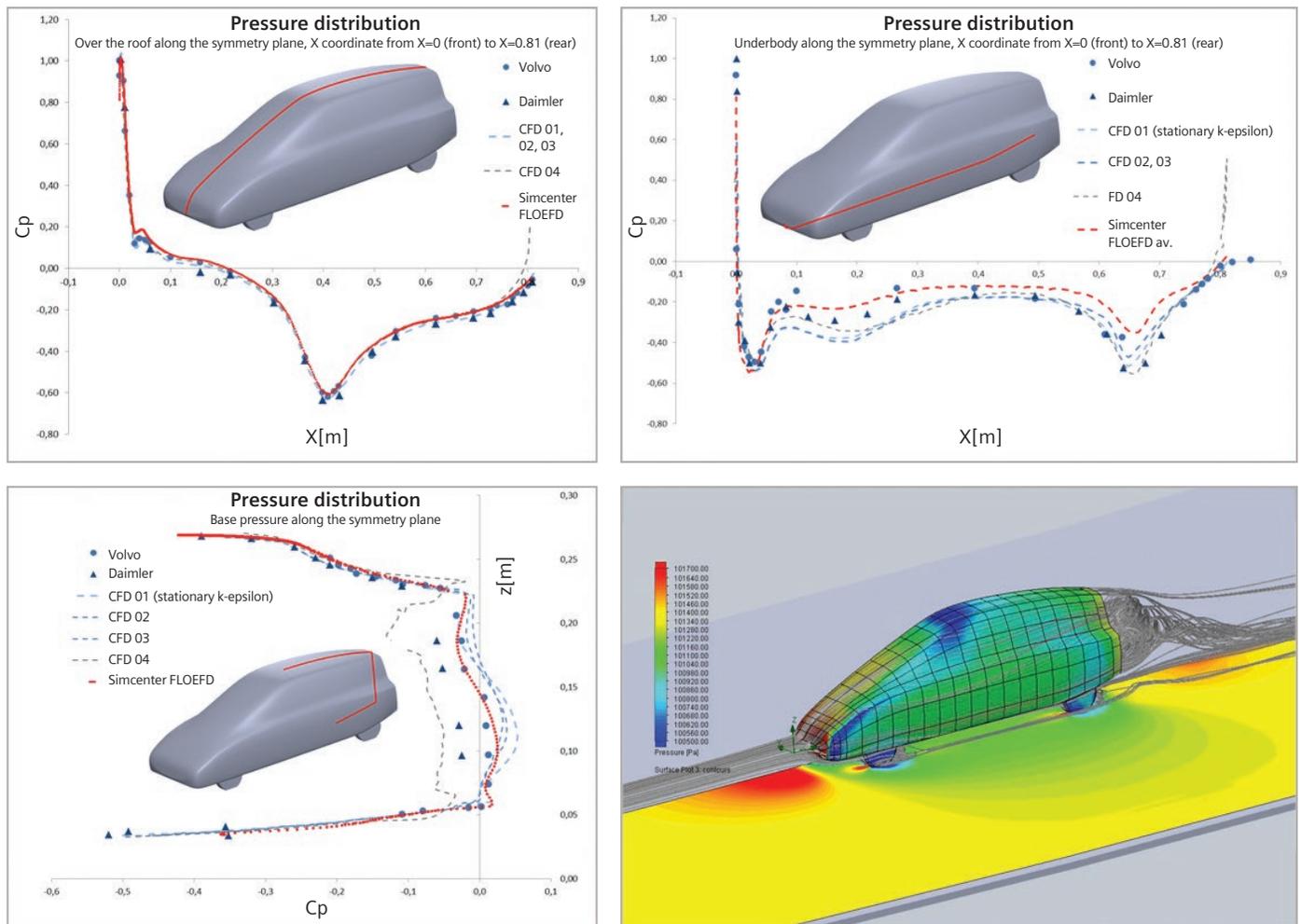


Figure 12: Comparisons of Cp calculated by Simcenter FLOEFD with experimental data and depiction of the overall flow field.

Summary and conclusions

The biggest barrier to user productivity with industrial CFD tools today is dealing quickly and effectively with complex CAD geometries through the generation of usable meshes within realistic (short) engineering timescales. Efficient usage of CFD tools embedded into CAD systems – the ideal situation – 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. Software based on this technology, like Simcenter FLOEFD, shows high levels of accuracy in an efficient, practical tool for CFD experts and CAD engineers alike to solve various tasks in many industries.

The SmartCells approach has been designed to mesh complex geometries in seconds and minutes with relatively coarse CFD meshes. 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 industrial product manufacturing competition. It also solves the age-old “Achilles heel” of CFD, the time-consuming specialist nature of mesh generation, thus paving the way for the democratization of CFD usage.

The SmartCell approach to CFD described in this paper has proven successful over the last 20 years for a wide range of industrial benchmarks and applications (some shown herein), and is regularly employed by OEMs and Tier 1 suppliers in the automotive, aerospace and other industries. Ultimately, the benefit of these synergies of numerical and engineering techniques must be seen in comparison with traditional CFD approaches where up to an order of magnitude in productivity has been seen. With the speed of manufacturing design cycles ever increasing, and always in the context of ever-present PLM software that all engineers use in order to improve their designs, engineers need CFD simulation results ever faster but without loss in accuracy (Eigner M., 2013). This compression of CFD workflows is shown schematically for the various CFD stages in figure 13.

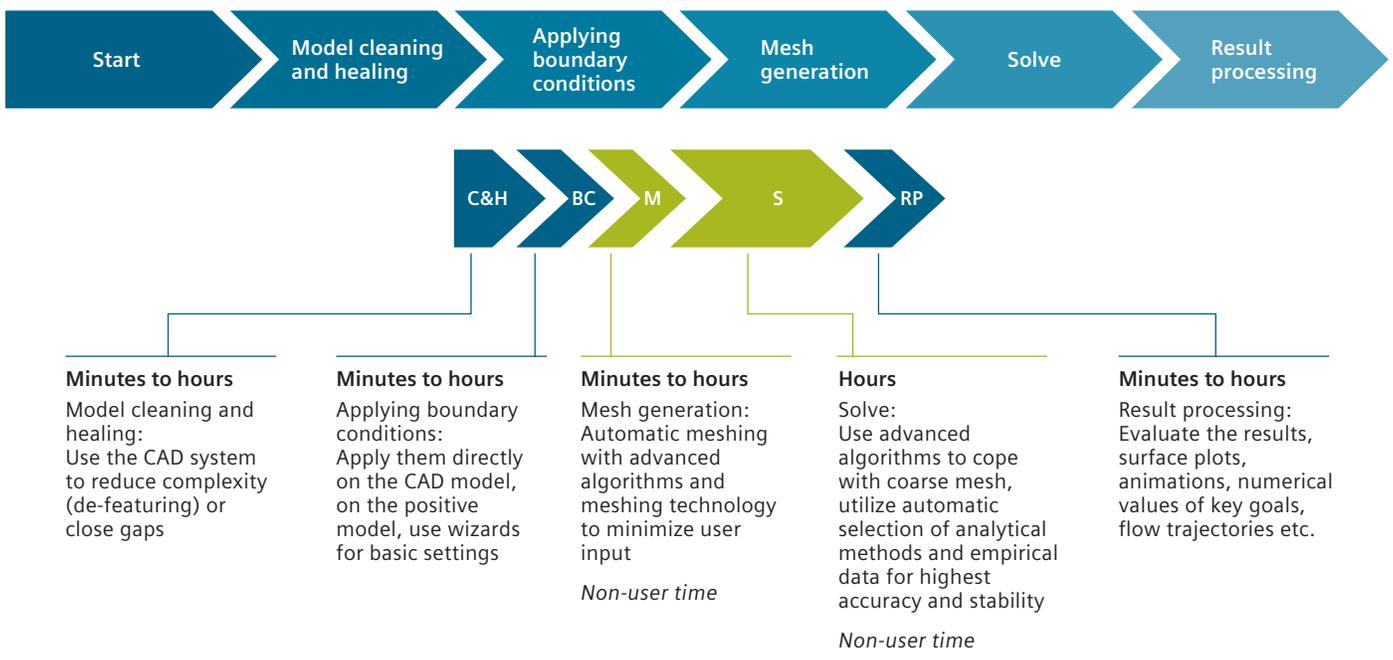


Figure 13: SmartCell enabled efficiencies of the CFD simulation process.

References

- Boysan, H.F., Choudhury, D. & Engelman, M.S. (2009). "Commercial CFD in the Service of Industry: The First 25 Years" in *Notes on Numerical Fluid Mechanics and Multidisciplinary Design*, Volume 100, pp. 451-461.
- Dumnov G., Muslaev A., Streltsov V., Marovic B. "Cavitation Process Simulation for Automotive Applications with an Isothermal Solver Approach," SAE World Congress 2013 (Detroit, MI, USA).
- Dumnov, G., Kharitonovich, A., Marovic, B.; Sobachkin A., (2016). "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.
- Eigner, M. (2010), Future PLM – "Trends aus Forschung und Praxis," University of Kaiserslautern Blog, 2010.
- Hanna, R. K., Parry, J., (2011). "Back to the Future: *Trends in Commercial CFD.*" NAFEMS World Congress, Boston, USA.
- Ivanov A.V., Trebunskikh T.V., Platonovich V.V. "Validation Methodology for Modern CAD-Embedded CFD Code" from *Fundamental Tests to Industrial Benchmarks*, NAFEMS World Congress NWC 2013, Austria, Salzburg.
- 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.
- Lam C.K.G., Bremhorst K.A. (1981). "Modified Form of Model for Predicting Wall Turbulence," *ASME Journal of Fluid Engineering*, 1981, 103: 456-460.
- Launder, B.E. Spalding, D.B., (1974). "The numerical computation of turbulent flows," *Computer Methods in Applied Mechanics and Engineering*. 3 (2): 269–289. doi:10.1016/0045-7825(74)90029-2.
- Marovic B. (2013). "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.
- Nakashima T., Sasuga N., Ito Y., Ikeda M., Ueda I., Kato Y., Kitayama M., Kito K., Koori I., Koyama R., Shimada Y., Hanaoka Y., Higaki T., Fukuda K., Yamamura J., Li Y. (2013) "Benchmark of Aerodynamics CFD of Simplified Road Vehicle Model," JSAE paper number: 20134343, 2013: 8-28.
- Sabeur, M. (2015). "Frontloading CFD in the Automotive Development Process," NAFEMS European Conference, December 2015, Munich, Germany.
- Sobachkin A.A., Dumnov G.E. (2013). "Numerical Basis of CAD-Embedded CFD," NAFEMS World Congress Austria, Salzburg.
- Spalding, D. B., (2007). "CFD Past, Present and the Future," Lecture at the Sixteenth Leont'ev School-Seminar Saint Petersburg, Russia.
- Uppuluri S., Proulx J., Marovic B., Naiknaware A. (2013) "Characterizing Thermal Interactions between Engine Coolant, Oil and Ambient for an Internal Combustion Engine." SAE World Congress 2013 (Detroit, MI, USA).
- Van Driest E.R. (1956). "On Turbulent Flow near a Wall," *Journal of the Aeronautical Science*, 1956, 23(10): 1007.
- Watson J.C., Dumnov G., Ivanov A., Muslaev A., Popov M. (2015). "Evaluating Water Film and Radiation Modeling Technologies in CFD for Automotive Lighting: NAFEMS World Congress," NWC 2015 and the 2nd International SPDM Conference, San Diego, CA, USA, June 21-24, 2015.
- Weinhold, I., Parry, J. (2013) "The Third Wave of CFD," NAFEMS World Congress, Austria, Salzburg, June, 2013.

Siemens PLM Software

Headquarters

Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas

Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe

Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific

Unit 901-902, 9/F
Tower B, Manulife Financial Centre
223-231 Wai Yip Street, Kwun Tong
Kowloon, Hong Kong
+852 2230 3333

About Siemens PLM Software

Siemens PLM Software, a business unit of the Siemens Digital Factory Division, is a leading global provider of software solutions to drive the digital transformation of industry, creating new opportunities for manufacturers to realize innovation. With headquarters in Plano, Texas, and over 140,000 customers worldwide, Siemens PLM Software works with companies of all sizes to transform the way ideas come to life, the way products are realized, and the way products and assets in operation are used and understood. For more information on Siemens PLM Software products and services, visit www.siemens.com/plm.

www.siemens.com/plm

© 2019 Siemens Product Lifecycle Management Software Inc. Siemens and the Siemens logo are registered trademarks of Siemens AG. Femap, HEEDS, NX, Simcenter 3D, Solid Edge and Teamcenter are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries. Simcenter and Simcenter FLOEFD are trademarks or registered trademarks of Siemens Industry Software NV or any of its affiliates. This document contains information that is proprietary to Mentor Graphics Corporation and may be duplicated in whole or in part by the original recipient for internal business purposes only, provided that this entire notice appears in all copies. In accepting this document, the recipient agrees to make every reasonable effort to prevent unauthorized use of this information. Creo is a trademark or registered trademark of PTC Inc. or its subsidiaries in the U.S. and in other countries. Microsoft and Microsoft Office are registered trademarks of Microsoft Corporation. CATIA and SolidWorks is a registered trademark of Dassault Systèmes SolidWorks Corporation. All other trademarks, registered trademarks or service marks belong to their respective holders.

76293-A4 1/19 C