



SIEMENS

Ingenuity for life

Siemens Digital Industries Software

Simcenter FLOEFD: Frontloading CFD

Because innovation requires
accurate and fast answers

[siemens.com/simcenter](https://www.siemens.com/simcenter)

Perspective



When I was seven years old my father took me to a computer exhibition. It was at the end of the 1960s and in the Soviet Union such events were very unusual. At the end of the day, I inherently knew that someday all elements of design and production would take place only on computers.

Fast-forward 20 years when my colleagues and I were working on simulation software for use in the Russian aerospace industry. Accuracy was not a luxury – it was a given. In addition, we didn't have access to powerful computers. So our software had to achieve the same speed and accuracy as those which were developed in the West. We had to develop innovative techniques that made our

code as efficient, and fast, but without having access to the same hardware.

Then in the early 1990s my childhood memory and my professional life converged. At the time, a group of my colleagues and I were looking for funding to start a new CFD company. After listening to our pitch, the venture capitalist told us that he was not interested in funding another "me too" CFD software. And that he'd only get involved if we delivered something that was truly innovative... a solution that would combine design and simulation.

That single statement became our mantra when we started work on Simcenter FLOEFD™ software.

Frontloading CFD refers to a concept that's been adopted by thousands of companies and it refers to the practice of conducting simulation early and often.

Alexander Sobachkin,
Product Line Director,
Siemens Digital
Industries Software,
Simulation and Test
Systems

In order for simulation to become a natural step during the design process, it has to be more closely connected with design. That's a simple enough concept to say, but it is not easy to achieve it. The main task of a mechanical engineer is design – it is not simulation. He is under pressure to develop a model that meets the requirements of the project. His main tool is CAD – everything else is supplementary and is used to make his designs better. Therefore, simulation is simply a

method by which trends are examined, and less desirable design ideas are dismissed.

Now CFD by nature is one of the most complicated disciplines in the process and classical CFD requires a lot of time and specialist knowledge to use. Therefore, traditional CFD is ill-suited to the needs of the design engineer who is interested in gaining insight, and quickly. And that's why Simcenter FLOEFD is the perfect solution for the mechanical

engineer and enables companies to frontload CFD.

Frontloading CFD refers to a concept that's been adopted by thousands of companies and it refers to the practice of conducting simulation early and often – as discussed in a presentation titled "Frontloading CFD in the Automotive Development Process" by Dr. Moh Sabeur at the NAFEMS European Conference in December 2015. During the design phase, speed is of the essence. By iterating rapidly, engineers can discard the less attractive ideas and innovate more. And this is exactly where Simcenter FLOEFD fits into the process, it reduces overall time to a solution by as much as 65-75 percent compared to other CFD tools. And as we all know early analysis can translate to a fair amount of savings as the cost to extract defects rises exponentially further down the process.

According to the U.S. Department of Defense (as reported by the Defense Acquisition University), 95 percent of the lifecycle cost for Department of Defense projects is committed by the test phase. In other words, the cost of the product is locked in by decisions made during the early concept stages when very little is known about the final design, but only a fraction of the R&D investment has been committed.

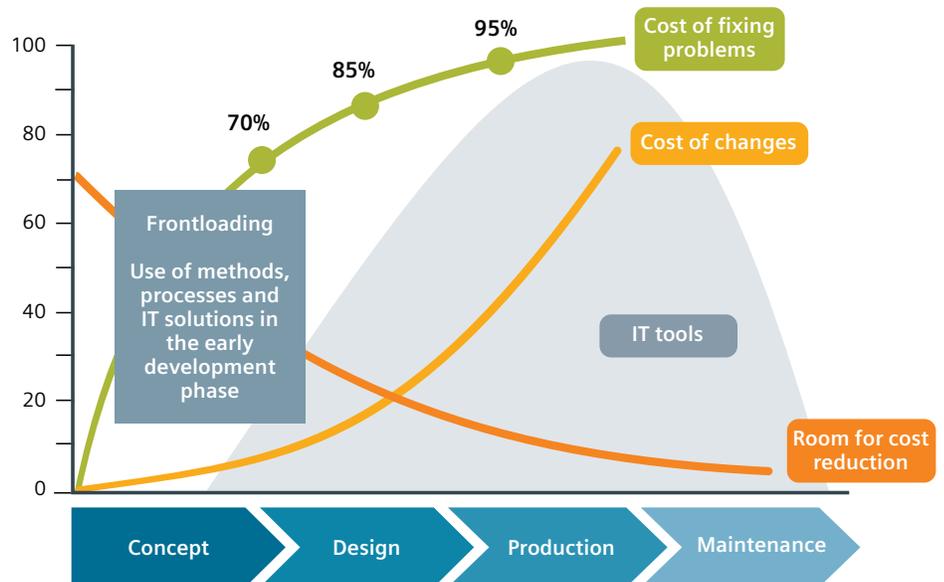
Therefore it is more cost-effective to simulate early and often. Simcenter FLOEFD features groundbreaking technology that provides accurate analysis results quickly thus enabling designers to validate their designs quickly. For example, the Simcenter FLOEFD mesh features SmartCells™ technology that in turn allows Simcenter FLOEFD to create Cartesian-based meshes that are typically 10 times smaller in size than

traditional CFD meshes for the same domain resolution, while retaining high levels of accuracy. And since Simcenter FLOEFD is embedded in today's most popular CAD programs, it takes advantage of CAD technology to enable the user to conduct variant analysis and explore a succession of ideas without risking project deadlines or sacrificing accuracy as discussed in "Validation Methodology for Modern CAD-Embedded CFD Code: From Fundamental Tests to Industrial Benchmarks" by Ivaov, Trebunskikh and Platonovich at the NAFEMS World Congress NWC 2013 and "Simulation Time Saving Approach Based on the Synergy of Numerical and Engineering Methods for Experts and Design Engineers" presented by Dumnov, Kharitonovich, Marovic and Sobachkin at the FISITA Automotive Congress in 2016.

So if your organization is grappling with any of these questions then you need to look for a new solution:

- Do you need to penetrate a new market or get products out to market faster?
- Do you need to develop products that are new and innovative?
- Do you need to minimize your costs to be more competitive?

You need to think about how simulation and design can be combined together efficiently. You need to look at Simcenter FLOEFD. It is the only true solution for frontloading CFD as many companies will attest. Read about their successes and then let us know how we can help you successfully frontload CFD within your organization.



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

Frontloading CFD award winners



Siemens Digital Industries Software is pleased to announce the winners of the 2016 Simcenter FLOEFD Frontloading CFD Award. The award recognizes excellence in implementing frontloading CFD through award-winning Simcenter FLOEFD. Our 6-person panel of judges rated each entry on several criteria including clear demonstration of use of frontloading CFD and the pragmatic approach taken in the application.

The top prize was won by Ms. Aihua Wang, a thermal engineer in the automotive lighting group at Magnetti Marelli. Her presentation titled "Integrating Thermal Analysis into Automotive Lighting Product Design" was delivered at the Integrated Electrical Solutions Forum (IESF) Conference in 2015. IESF is a global conference for electrical/electronic design engineers, managers and executives. Winning the top prize entitled Ms. Wang to \$1,500 in cash and a trophy. Ms. Wang chose to donate the funds to her charity of choice— the American Society for the Prevention of Cruelty to Animals (ASPCA).

Our panel of judges chose Design and integration of cooling systems and power packs/ powertrains to meet the next environmental (tier IV) regulations" by Kolio Kojouharov as one of the two runners up. Mr. Kojouharov, who at the time was working as an engineer at Liebherr-Werk Nenzing GmbH made the presentation at the October 2015 Siemens Digital Industries Software U2U in Munich.

Lastly, Mr. Georg Jäger who is an engineer at Hoval was selected as another runner-up. His whitepaper titled "From conceptual brainstorming to a customized condensing boiler", was written for use at the Simcenter FLOEFD Simulation Conference which is taking place November 8 -9, 2016 in Frankfurt.

We are very proud of the results that our customers are able to achieve with the aid of Simcenter FLOEFD. This award is a celebration of their achievement and we look forward to recognizing others next year.

Winner

Aihua Wang

Runner ups

Kolio Kojouharov
Georg Jäger



Robert Bosch Automotive Steering GmbH steers towards flow optimization

Robert Bosch Automotive Steering GmbH is a pacesetter and trendsetter in the field of steering systems for passenger cars and commercial vehicles. Bosch Automotive Steering develops technically-sophisticated solutions for all major vehicle manufacturers.

Ever-shortening product cycles and decreasing development times in the automotive industry has raised the need for up-to-date simulation tools that can deliver rapid, yet highly dependable simulation results. The use of Simcenter FLOEFD for PTC Creo software enables the evaluation of future automotive components at the earliest possible stage during the development cycle. Allowing problem identification and correction when the concept is first evaluated at the feasibility stage of the project.

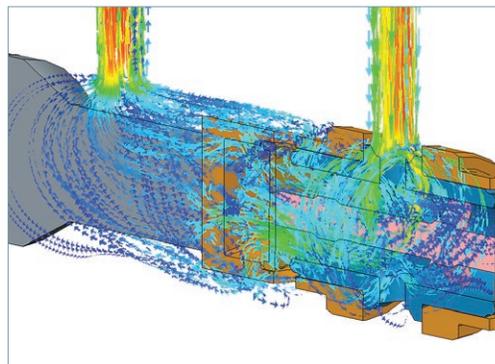
A project recently undertaken by Bosch Automotive was the development of steering assistance in commercial vehicles, which is performed by means of a hydraulic system circuit. A double valve is used to supply the feed pump as a control valve. The objective is to supply the required volume flow, taking into account the given pressure conditions, and keeping the pressure drop at required volume, flow rates to a minimum. Simultaneously, cavitation effects have to be avoided.

Several design variations for the valve were investigated in Simcenter FLOEFD for PTC Creo. Aside from the main geometry modifications, detailed changes to individual

components and their effects were analyzed. The insights gained were incorporated from an early stage in the development of the product concept. The most efficient overall design based on the simulation results was manufactured as a prototype and measured in a test setup. The measurements confirmed that the simulation results were accurate, reducing the number of physical prototypes to just one.

By frontloading the CFD simulations Bosch Automotive was able to optimize the design of the pin in detail, allowing it to be designed for use across a series of such valves in the future. Cost optimizations have already been achieved at the product concept phase for the series.

To learn more about Bosch Automotive design challenges please read <http://go.mentor.com/4Nf9p>



Flow vectors inside the valve.

From an original article by Rolf Haegele, Development Engineer Acoustics / Simulation, Robert Bosch Automotive Steering

“Using Simcenter FLOEFD within our PTC Creo environment has allowed us to frontload full CFD simulation into our design processes, cutting design times and making optimization possible from the very start of the development process. Simcenter FLOEFD has helped us meet today’s requirement for short development cycles.”

Rolf Haegele, Robert Bosch Automotive Steering GmbH

Sports car brake cooling simulation with CAD-embedded CFD

Brake cooling is a crucial area in motorsport and sports car engineering. A recent thesis project by Arne Lindgren of Halmstad University in Sweden considered different cooling solutions for the extreme conditions of super cars. The project, conducted for super car manufacturer Koenigsegg Automotive AB, was to design an efficient brake cooling solution for their latest model, the Regera (figure 1).

Regera, which means “to reign” in Swedish, is the first Koenigsegg car with hybrid-technology, the combined power of its internal combustion engine and electric motors exceeds 1,500 horsepower. Such a powerful car needs effective and reliable brakes. During a braking event from 300 to 0 km/h, the average brake power is over 1 MW in the Regera.

Koenigsegg Automotive AB is a Swedish company founded in 1994 by Christian von Koenigsegg. The first prototype was completed in 1996 and the series production of their CC8S model started in 2002². The Koenigsegg CCR became the fastest production car in 2005 and the CCX model took the Top Gear lap record in 2006 with a time that remained unbeaten for over two years. In 2014 the One:1 model was introduced, the world’s first production car with an hp-to-kg curb weight ratio of 1:1. The company employs approximately 120 staff including an engineering department consisting of about 25 engineers.

Lindgren evaluated several brake cooling designs for the Regera in his thesis. For his CFD simulations he used Simcenter FLOEFD embedded in CATIA V5 from Siemens Digital Industries Software – experimental testing being deemed too expensive and lacking in acquisition of certain flow data¹.

While dedicated brake cooling is not necessary for ordinary passenger vehicles, it is a huge challenge for sports cars that must tolerate racing conditions. The cooling effect of the ambient air is usually sufficient for ordinary car brakes during normal driving conditions. Modern road vehicles are equipped with internally vented brake discs (at least at the front axle, and usually also at the rear). The internal vanes help to pump air through the disc, internal channels are where the biggest heat dissipation takes place¹.

Over-heating of brakes can lead to a number of problems:

- Friction material degradation
- Thermal stress in the brake disc, which can lead to distortion and stress cracks
- Brake fluid vaporization in the brake caliper

These failures can lead to partial or complete loss of braking, which is very serious.

With regard to racing cars and cars for track driving, this issue becomes more complex. Special cooling ducts which direct ambient airflow to the brakes are used to ensure sufficient cooling performance taking into account the more frequent braking intervals that occur during track driving.

Brakes are mainly cooled through the heat transfer method of convection, where a fluid absorbs heat and transports it away from the hot object. Lindgren focused his work on improving the convective air cooling of the brakes, and limited his studies to the front axle brakes (figure 2) since this is where heat generation is at its greatest. The brake cooling solution that was used on the previous generation of Koenigsegg cars was used as a baseline for the cooling simulations.

By Mike Gruetzmacher,
Simcenter FLOEFD
Product Specialist,
Siemens Digital Industries
Software



Figure 1. Koenigsegg Regera.



Figure 2. Koenigsegg Regera front brake.



Image courtesy of Koenigsegg Automotive AB

The baseline design consists of inlet ducts in the front bumper of the car (that captures ambient airflow), and flexible hoses that channel the air to ducts (or nozzles), which are mounted on the wheel bearing carriers and direct the cooling air towards the center of the brake discs (figure 3). For the simulations, CAD models of the relevant geometry were provided by Koenigsegg which were used for the embedded CFD simulations.

The main geometries used in the simulations were:

- The brake discs made of Carbon fiber-reinforced silicon carbide (C/SiC), diameter 396 mm and thickness 38 mm, equipped with radial ventilation channels
- The brake pads
- The brake caliper
- The upright (or wheel bearing carrier)
- The hollow wheel axle
- The 19" wheel rim with tire
- The wheelhouse geometry

After simulating the baseline configuration, different brake duct concepts were generated and their cooling effect was simulated. The same computational model was used for both baseline and concept simulations, only the brake duct geometry and, the position of the duct inlet boundary conditions were changed. The simulations were conducted directly within the 3D geometry models in the CATIA V5 embedded Simcenter FLOEFD CFD software. This allowed for efficient simulation of complex geometries. An efficient and productive workflow was found as a result of the automatic meshing function, the case configuration wizard, the post processing features, and because geometry could be modified directly within the CAD environment¹.



Figure 3. Brake cooling duct mounted on a Koenigsegg Agera upright (the baseline design). Klingelhofer, 2013, Ref 3.

As the objective of the project was to investigate many concepts, a reasonable calculation time was required. Therefore, a complete simulation of the entire vehicle was not expedient. A partial car body with wheelhouse was used as well as wheel and the brake as assemblies (figure 4) with similar dimensions and ground clearance as the Regera.

The ambient velocity was defined as 150 km/h, which is the typical average speed on a race track. An additional airflow from the radiator was applied on the inboard side of the wheelhouse (figure 4, red arrows). The airflow through the flexible hose from the inlet ducts in the front bumper was modeled as a pressure to get a realistic flow rate for all possible brake duct designs. The rotation of rotationally symmetric geometries, such as the tires and brake disc friction surfaces, were defined with wall conditions. The Simcenter FLOEFD sliding mesh approach

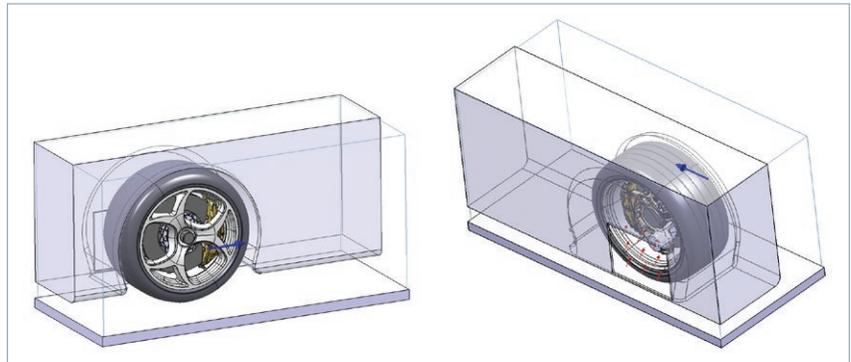


Figure 4. The geometrical wheel arch model (blue arrow indicates ambient airflow direction).

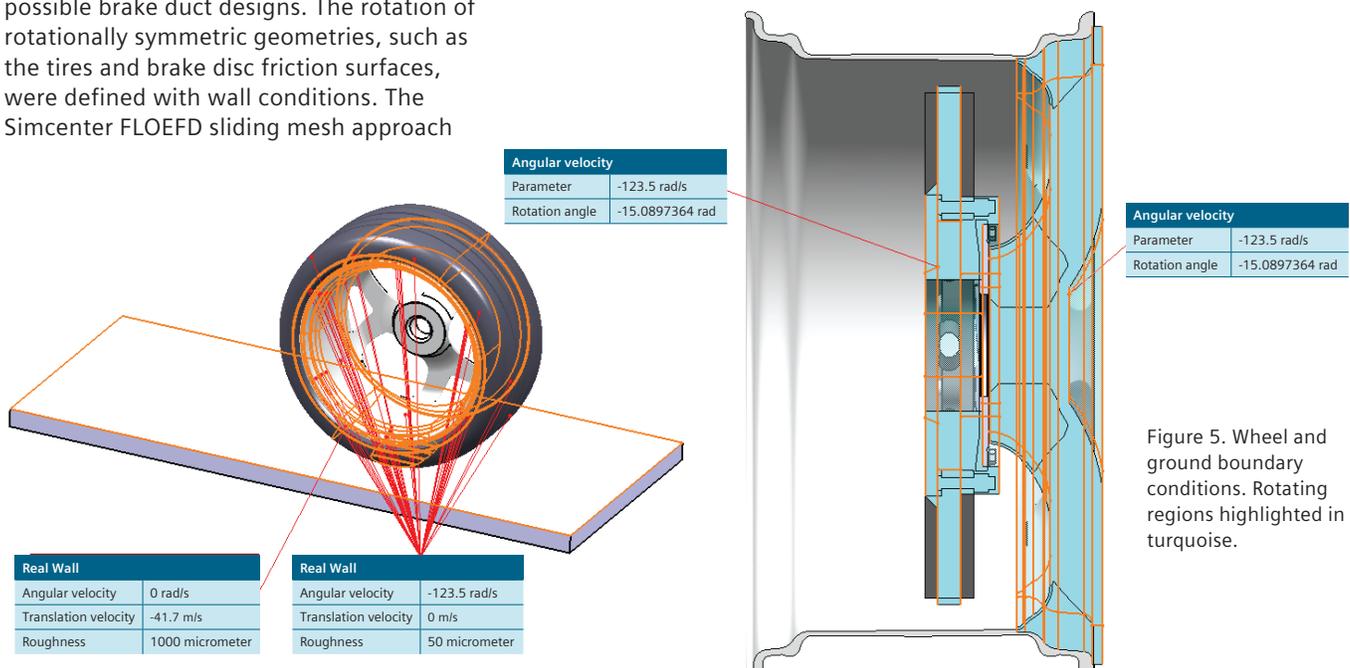


Figure 5. Wheel and ground boundary conditions. Rotating regions highlighted in turquoise.

was applied to the non-rotationally symmetric parts. A 3D body (rotating region) is used to define which geometries should rotate, in this case the rim spokes and brake disc channels. A translation velocity of 150 km/h was applied to the ground (figure 5) to include ground effects. Only convection was considered for the CFD simulations as this is the easiest heat transfer process to influence and has a proportion of about 60-90 percent of the total heat dissipation. The surface temperatures applied on the surfaces of the parts (figure 6) were based on values recorded by Koenigsegg during track testing.

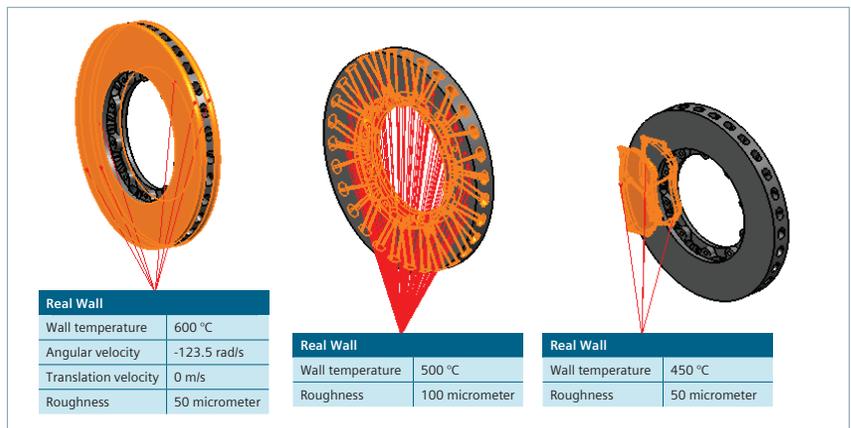


Figure 6. Disc and pad with Simcenter FLOEFD wall boundary conditions.

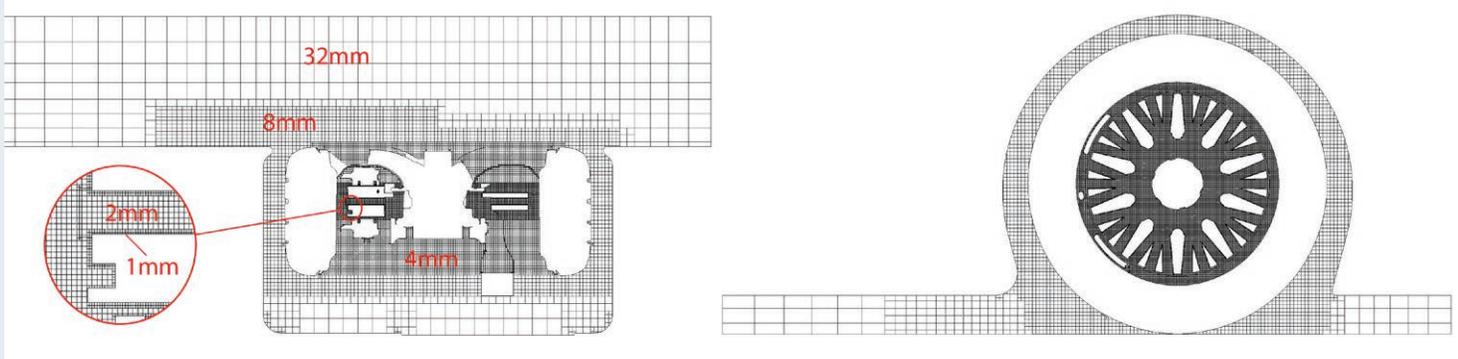


Figure 7. Simcenter FLOEFD Computational mesh of the Wheel, Brake Disc and Wheel Housing.

The simulation was conducted with approximately 3.5 million cells (figure 7) using the Simcenter FLOEFD two-scale wall function technique, which enables the use of a coarser mesh than would be otherwise necessary in traditional CFD codes.

The concepts were based on Lindgren’s own ideas, observed brake cooling designs, and observations in other applications while taking into account only concepts that are possible to manufacture. Twelve different concepts were investigated using the described Simcenter FLOEFD boundary conditions. With the given settings, the simulation time was approximately 24 hours on a computer, with a six-core Intel Xeon E5 CPU at 3.5 GHz and 32 GB RAM.

The baseline simulation results are shown in figure 8:

Each concept was compared with the baseline concept. The investigations focused on the cooling of the brake discs as most of the braking energy goes into the discs. The approach was to increase the convective cooling by breaking up the temperature boundary layer, which can be done by using high air velocity or by introducing turbulence. In addition, other design criteria were considered to ensure that the solution withstands forces, vibrations and temperatures etc., that occur in driving conditions.

The investigations showed that a local improvement of the airflow often led to worse heat dissipation in other areas simultaneously. The airflow rate was simply not sufficient to improve the cooling over larger surfaces. Another early discovery was that the design of the cooling channels in the brake discs could be improved. But the brake disc design was outside the scope of this project so it was not studied further. The break-through was the idea of

putting the brake duct inlet in the center of a wheel axle that has radial channels. This resulted in higher hose flow rate because the radial channels in the axle and the brake disc work together as a centrifugal fan. Finally, this concept was supplemented by a “passive” cooling design (not relying on airflow from the hose), realized by two ring-shaped plates perforated with slots (figure 10). The simulations showed that these plates improved the cooling

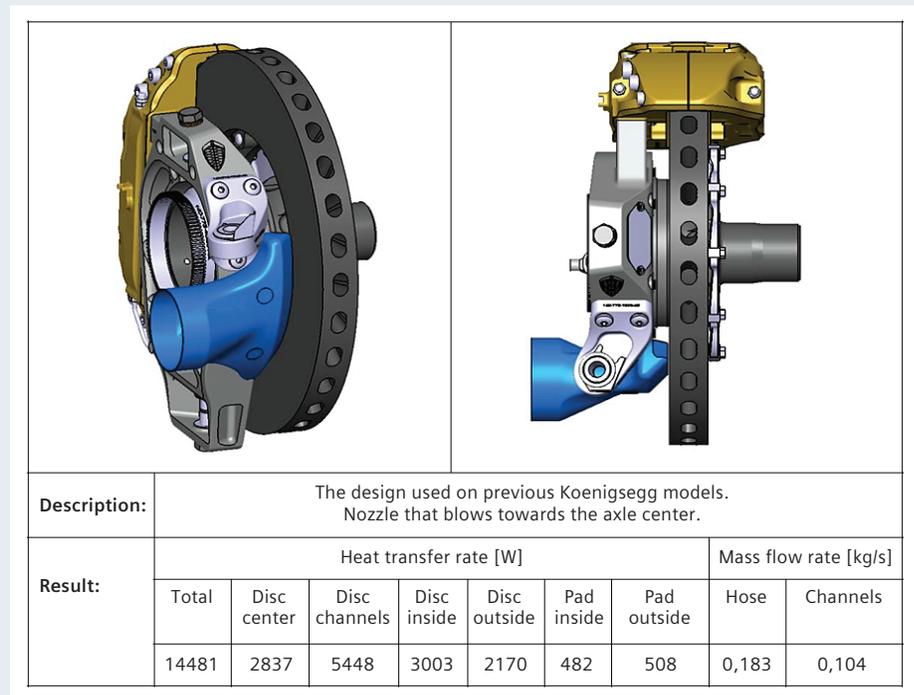


Figure 8. Baseline brake duct highlighted in blue and table with the numerical results from Simcenter FLOEFD.

of the discs friction faces, but Lindgren expresses skepticism of these results as the simulations didn't include radiation (in reality the plates would reflect heat back to the disc). Concepts 8 (figure 9) and 12 were the most promising, although concept 12 needs further analysis or testing to see the effect of radiation. The designs of the concepts and the results given as the difference in percent from the baseline values are shown in figures 9 and 10. Concept 8 has the advantage that only a few relatively simple additional parts are needed.

From the almost infinite number of possible cooling solutions, 12 concepts were analyzed (figures 11-13) and compared with the baseline design from Koenigsegg. The two most promising solutions lead to an overall thermal improvement of 14 percent and 25 percent. Concept 8 was proposed as an enhancement, which had the hose inlet in the center of the wheel axle thus directing the cooling air through radial channels to the brake disc. Simcenter FLOEFD simulations indicated that the proposed design should result in 14 percent higher heat transfer rate compared to the previously used cooling solution. In addition to these cooling ducts, some passive cooling devices could also be considered in future.

In this study Simcenter FLOEFD was able to provide trend predictions, within engineering timescales, for a wide range of concepts although it was recommended that finer meshes and radiation effects be examined more in the future which require more computational resources. The CFD results described here indicate that when compared to brake cooling with the previous Koenigsegg ducts analyzed as a baseline, new concepts could be created, analyzed and developed in an easy iterative process. The simulations with the CATIA V5 embedded version of Simcenter FLOEFD made these investigations possible because creating a real prototype of each concept would lead to high cost and time requirements. The most promising solutions can now be investigated further in terms of structural analysis, manufacturing processes and finally produced as a prototype.

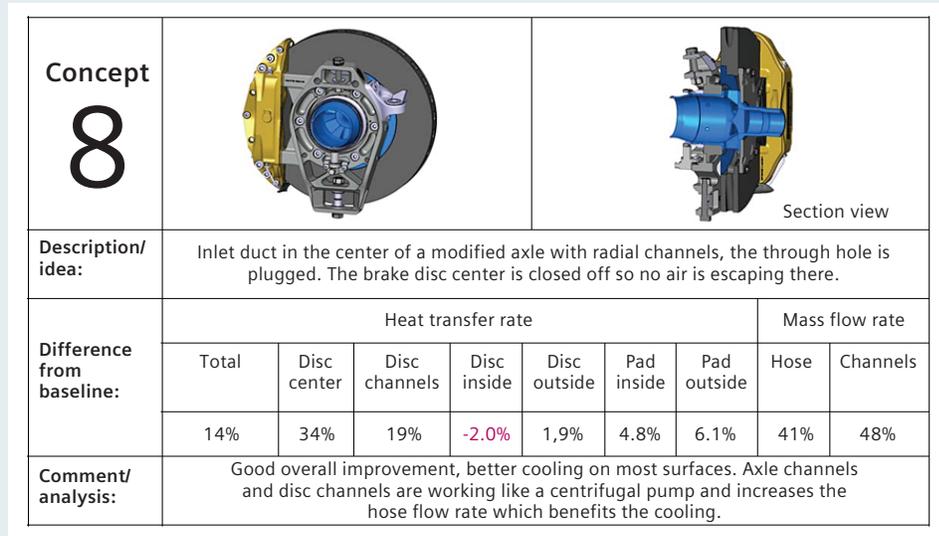


Figure 9. Overall view and Simcenter FLOEFD results for concept number 8.

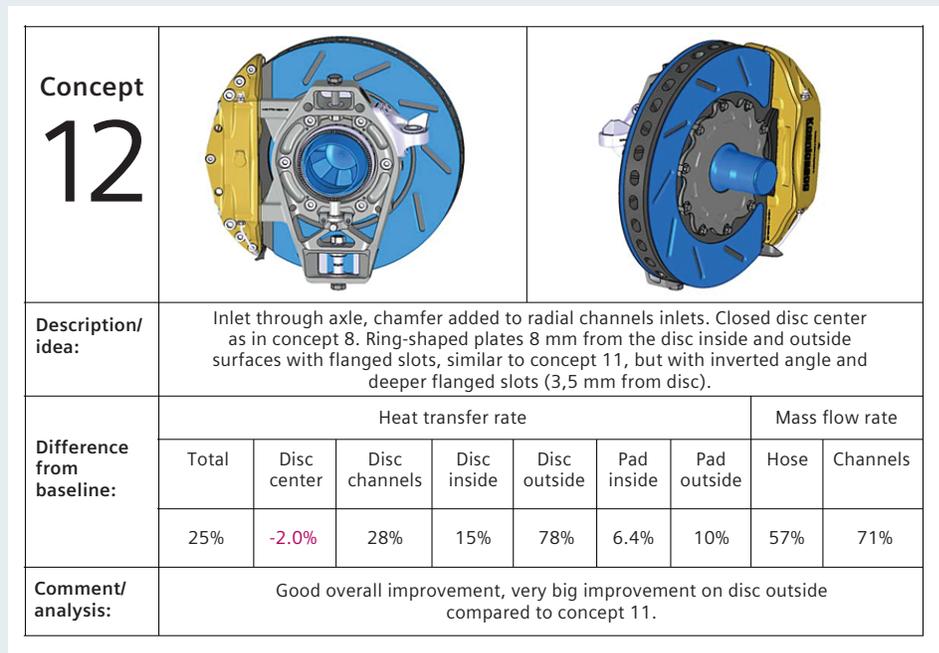


Figure 10. Overall view and Simcenter FLOEFD results for concept number 12.

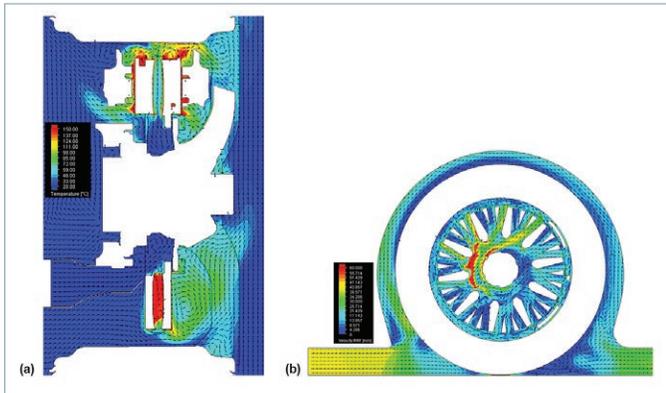


Figure 11. Baseline Case (a) temperature cut-plot on a horizontal plane through the wheel and (b) vector velocity on a vertical plane through the disc channels.

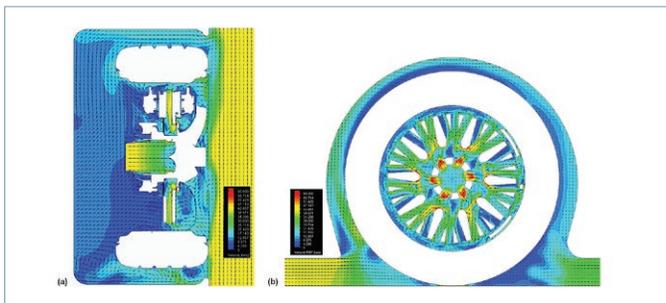


Figure 12. Concept 8 velocity cut-plot horizontal plane (a) through the wheel and cut-plot vertical plane through disc channels (b).

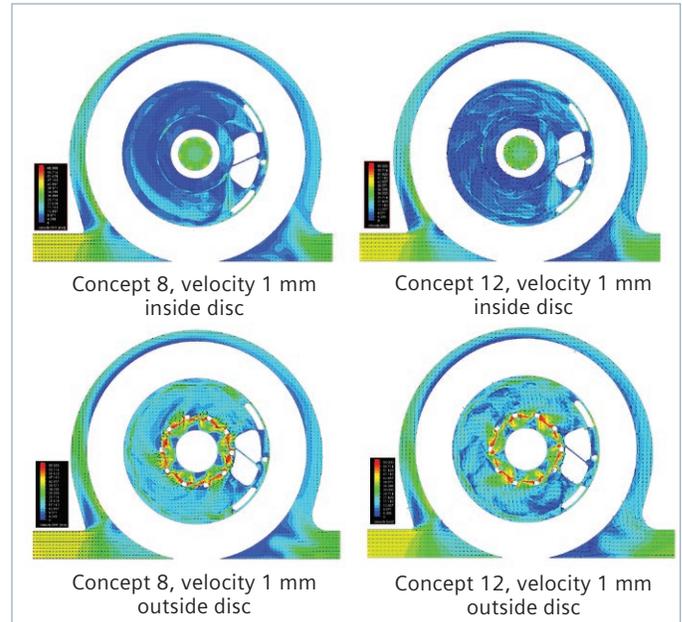


Figure 13. Comparison of Concept 8 and 12

References

1. Development of Brake Cooling”, Arne Lindgren, Bachelor Thesis in Mechanical Engineering, 15 credits, Halmstad 2016-05-20: <http://www.diva-portal.se/smash/get/diva2:938489/FULLTEXT01.pdf>
2. Koenigsegg website: <http://koenigsegg.com/>

“With Simcenter FLOEFD, the configuration of the simulation is made from within the CAD software, which eliminates the need for exporting/importing geometry. The software also has features such as automatic meshing, a case configuration wizard and integrated post processing, all from within the CAD software.”

Arne Lindgren, Halmstad University, Sweden

(After graduation Arne Lindgren was employed as a Design Engineer at Koenigsegg Automotive.)

Enabling engineers to achieve efficient design lifecycles

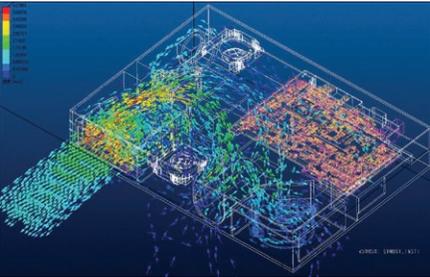


Figure 1. Cooling airflow verification inside the whole enclosure.

The world of technology is constantly evolving but Seiko Epson is at the forefront of projector innovation. The company is in the privileged position of being the pioneers as well as the leaders of their market space. Epson's projectors are widely used in offices, schools, retailers, museums, movie theaters and living rooms.

Heat sources in projectors, like power supplies and lamps, result in high temperatures inside the projector housing. As smaller, more compact portable projectors are continuously being developed the first consideration for the designers at Epson is always managing the heat sources. By the nature of the design and materials used, the units tend to retain heat in their body unless it is vented into the air or through other parts. Radiant heat transfer was traditionally the most important consideration in the development of projectors. When LCD technology was introduced, the development period halved as there was now a demand for projectors that were brighter, had more functionality, and were smaller.

“Simcenter FLOEFD is essential in our daily work. In many cases, we can't predict the results of analysis but Simcenter FLOEFD leads us to correct results automatically.”

Mr. Fumio Yuzawa, Seiko Epson

As with many miniaturized devices, thermal radiation is limited, so air cooling is required within the housing. Development timescales, if underestimated, prove costly if they subsequently overrun. Despite the new challenges faced in thermal analysis there is still a requirement to reduce development times and costs.

The group started to use Simcenter FLOEFD for Creo in 2009 in order to empower designers to analyze their own designs and to speed up productivity. Simcenter FLOEFD users

found new ways to solve difficulties with their newly-acquired skill set in analysis. Simcenter FLOEFD was able to assist in the challenges the team faced with design of semiconductors in projectors. Semiconductors reach high temperatures with natural air cooling and are typically designed under the 60 percent attainment of industry standards. Since designer knowledge and experience would not achieve this level of attainment easily, Simcenter FLOEFD proved to be the key in solving this challenge.

Why was Simcenter FLOEFD the right choice for Epson?

According to Mr. Hiroshi Abe at Epson, “The most important consideration in selecting an analysis software tool was that all team members could use it regardless of their level of ability. We evaluated the following three criteria:

1. The people who don't have much experience of analysis can use it easily. In particular, meshing, as this is one of the most difficult processes. Simcenter FLOEFD's automatic meshing enables you to just set a specific area of a model. As for workflow, we only needed to select “yes” or “no” by using the wizard and then we can also learn what we should set in the analysis process by habit.
2. It was important that the tool integrated with Pro/ENGINEER (now Creo). We didn't want to have to create another model for analysis and being CAD-embedded we could validate various analysis models repeatedly. We also wouldn't have any difficulty in switching between processes (from design to analysis).
3. A comprehensive database. Simcenter FLOEFD has a world-standard database. Especially, we are able to use other databases in the Siemens Digital Industries Software suite of products, such as Simcenter Flotherm™ software. It has real benefits for users.”

To learn more about how Seiko Epson empowers their engineers, please read <http://go.mentor.com/4PhyU>.

Society of Automotive Engineers of Japan (JSAE) benchmark of automotive aerodynamic test measurements

The Society of Automotive Engineers of Japan (JSAE) recently conducted a blind benchmark for commercial Computational Fluid Dynamics (CFD) software to demonstrate their accuracy against test validation data for the standard "Ahmed" body.

Each CFD software package had to analyze the airflow around the model and compare its predicted results to experimental data without knowing the data beforehand. In particular, JSAE wanted to verify how accurate each CFD tool was at predicting boundary-layer separation, pressure distribution and body forces on the model. It was up to the participants in the benchmark to choose the best mesh, turbulence model and other settings in their CFD codes to provide their best prediction. All

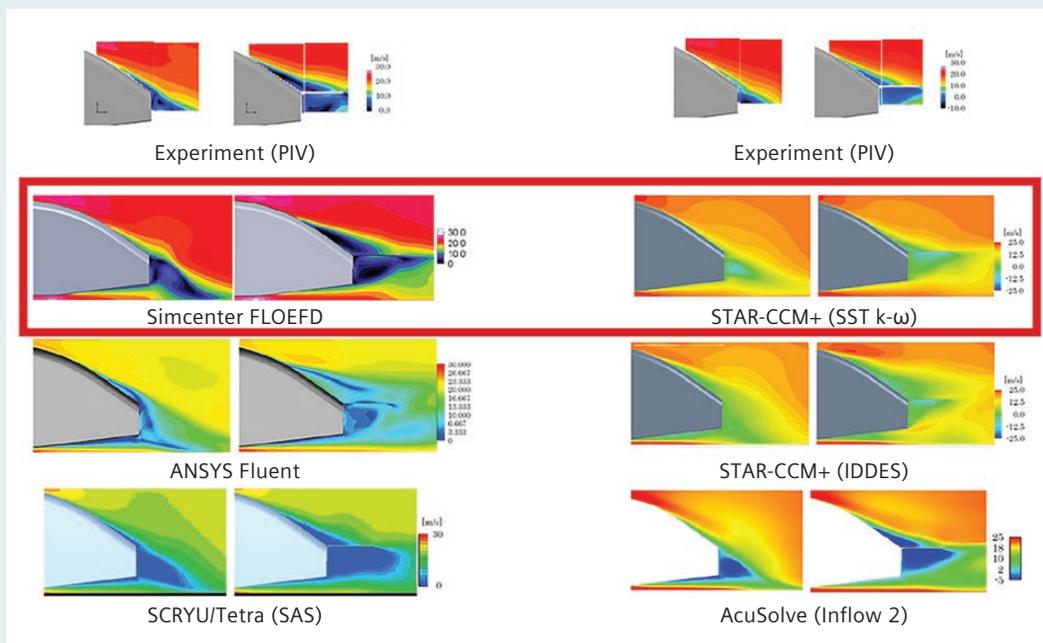
participants had to provide the results for drag, lift, and pitching-moment coefficients, as well as pressure coefficients at various sections of the vehicle body.

Seven organizations took part in the benchmark. KKE Inc., a reseller in Japan, used Simcenter FLOEFD for the project. KKE used the Cartesian mesh approach with solution adaptive refinement on an octree basis and local meshes around the body. Each cell level refinement was easily set-up in Simcenter FLOEFD and the rest of the mesh generation adaptation process was automated. Simcenter FLOEFD's solution adaptive mesh resulted in a cell count that was four to eleven times lower than the other CFD tools for the cases considered. Simcenter FLOEFD was also very

accurate for both drag and lift in Case 1, and had the best pitching moment prediction of all codes tested in Case 2.

In summary, compared to the other tools, Simcenter FLOEFD required less resources and calculation time to come-up with good overall results. It also shows quite good agreement with the wind-tunnel experimental measurements. Although the meshing and solver technology of Simcenter FLOEFD is a non-traditional CFD approach, this JSAE blind benchmark has proven that Simcenter FLOEFD is just as accurate, if not more accurate, than other commercial CFD software for a challenging automotive external aerodynamic study.

For the detailed version of the benchmark please read <http://go.mentor.com/4Phzl>.



Experimental test results from particle imaging velocimetry (PIV) measurement contour plots show that Simcenter FLOEFD and Simcenter STAR-CCM+ (SST k- ω) most closely match the wind-tunnel results.

Bring light into the foggy night: Engineers use Simcenter FLOEFD Frontloading CFD to bring a new LED fog light design to market

Due to their lower carbon-dioxide emissions, lower fuel costs and less noise, electric vehicles have gained in popularity. However, they are also posing some new and interesting engineering challenges for their designers. For example, the basic requirements for a good light design are regulations and safety; however, low power consumption and low weight for more efficient battery use are just as important for a light design used in electric vehicles.

Engineers at TQ Technology (TQT) worked to develop a new design for fog lights. The new design required far less power and weight in comparison to traditional designs (figure 1). The function of fog lights is to show the edges of the road and lane markings, and fog lights are used in poor visibility driving conditions including rain, fog, dust or snow. A good fog light produces a wide, bar-shaped beam of light with a sharp horizontal cut off (dark above, bright below) at the top of the beam, and minimal upward light above the cut off. They are generally aimed and mounted low.

The TQT engineers designed a multi-faceted reflector and compared its performance to three other designs. As a result, the reflector geometry was enlarged from 40 cc to 66 cc, a 65 percent increase, and they were able to reduce the LED from two chips to one. They also gained a better reflector efficiency, from 38.6 to 52 percent, to help deliver sufficient lumen flux to meet EU and UNECE regulations¹.

A better reflector efficiency was obtained by using a well on the edge to light up the two sides of the fog beam, thus the middle part needed less lumen flux to meet the regulation. Their design employs a 1.5-W driver for two fog lights (one pair), and a pair consumes 7.9 W. Compared to a 55-W halogen fog light (to meet the same regulation), this design saves 92.7 percent energy conventional 55W halogen lights and 38.4 percent compared with existing 6.5W fog lights.

The lower power 3.2-W LED design also potentially enables a lower weight light design. They analyzed three housing structures (figure 2). The junction temperature and the surface temperature of the LED was simulated in CAD-embedded Simcenter FLOEFD to evaluate the reliability of different structure effects during the design process. The key parameter was LED junction temperature, which needed to be less than 150 °C. The simulated ambient was 65 °C.



Figure 1. The new TQ Technology ultra-low power LED fog light.

The all-metal case had the best temperature (128 °C) on the junction, whereas the junction temperature reached as high as 243 °C in the plastic case which it cannot – assuming that the maximum temperature from the specification sheet is 150°C then it is already exceeds the temperature and will have a very short lifespan. A hybrid structure that partially used metal on the LED base and plastic for the reflector provided effective cooling and also kept the temperature low on the plastic housing. The light weight was reduced from 260 g (all metal) to 196 g (metal plus plastic), a 24.7 percent saving on weight. This hybrid solution was adopted as the commercial version of the fog light design (figure 3). Their final product delivers higher performance, energy savings and lower weight compared with halogen or other LED fog light designs.

Reference

1. United Nations (2011), "ECE Regulation No.19: Uniform provisions concerning the approval of power-driven vehicle front fog lights contents."

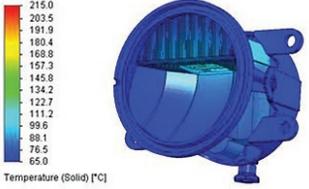
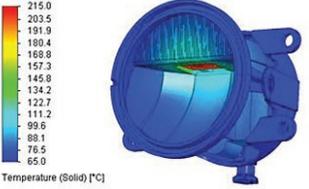
			
LED Ts (°C)	98	107	213
LED Tj (°C) Spec 150 (°C)	128	137	243
Weight (g)	260.4	196.2	170

Figure 2. Thermal simulation results from Simcenter FLOEFD of the different housing materials.



Figure 3. Car road-test comparison of the new TQ Technology LED fog light with a halogen fog light.

“Compared to a 55-W halogen light, this design saves 92.7 percent energy and 38.4 percent energy compared with existing 6.5W fog lights.”

Capturing fugitive emissions at a metal refinery plant

Engineers at Resonant Environmental Technologies completed a study on an existing fume extraction system at a ferromanganese furnace tapping area for Eramet Sauda. Located in Norway, the organization operates two ferromanganese furnaces and a manganese oxygen refining (MOR) units.

Tapping operations at ferroalloy smelters generate a significant amount of fume. Effective capture of fumes is required to meet environmental, occupational and legislative requirements. The study consisted of testing the tap-hole, launder and ladle area onsite as well as computational fluid dynamics (CFD) modeling with Simcenter FLOEFD to identify and evaluate solutions.

Secondary fume emissions are generated at the tap-holes and post-tap-hole operations. The existing secondary fume capture system had good capacity, but fugitive emissions were escaping from the furnace building. The manganese oxides in these fumes can be easily inhaled because of their small size with adverse health effects including manganese poisoning and various lung disorders. As such Norwegian laws for respirable manganese dust have become very stringent.

Testing onsite and observations during furnace operation provided a baseline of system performance and the input parameters for simulation of the fume flow with Simcenter FLOEFD. The engineers at Resonant Environmental Technologies measured gas flow rates and energy contents from each extraction area. They also took photographs and video footage to evaluate areas where fumes would be leaking. They validated the CFD simulations against site observations to ensure the accuracy of input parameters. Then they modeled several concepts and investigated suitable combinations to ensure they provided robust solutions.

They looked at solutions to optimize the secondary fume capture: air curtains (figure 1), adjusting the ladle hood aspect ratio and position, increasing the ladle extraction volume and extending the length of the tapping garage structure. The best overall solution was a combination of 2-m garage extension, increased extraction and air curtain.

They used Simcenter FLOEFD to simulate the fume generated at the tap-hole, launder and ladles and evaluate the capture efficiency of various extraction concepts. They used a steady-state formulation, rather than transient. Heat convection is inherently unsteady. While it was possible to use an unsteady formulation, during the verification process, it was concluded that the steady-state formulation would give satisfactory results to enable a conservative evaluation the capture efficiencies of the hoods.

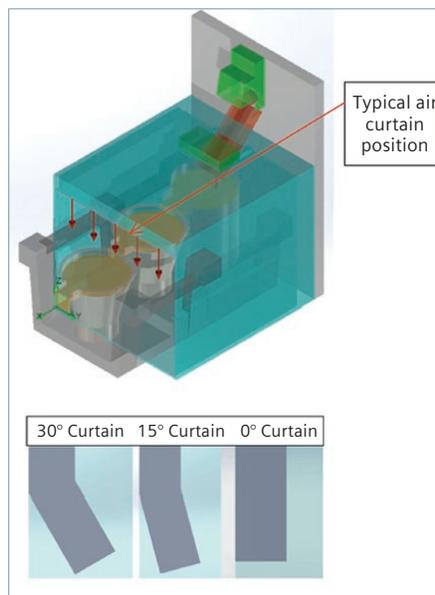


Figure 1. Typical air curtain arrangement.

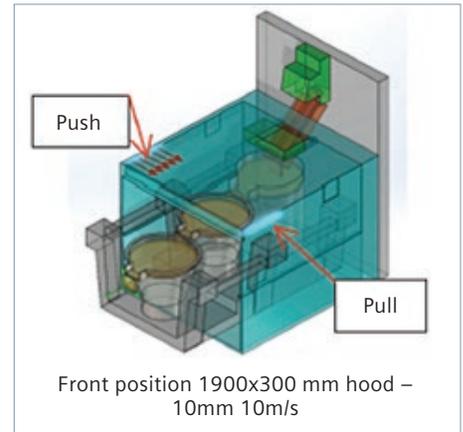


Figure 2.

Among their findings were:

1. A thin lower velocity air curtain is most effective in this application. However, varying the angle of the air curtain did not show any significant benefits (figure 4).
2. The slender hood aspect ratio showed a slight improvement over the current square one. And push curtains tests were found to be ineffective as the additional volume blown into the garage resulted in additional fume leakage. Most importantly, increasing the ladle hood extraction rate was shown to improve the capture significantly (figure 5).
3. An extended garage helped improve capture efficiency. And in order to maximize capture increased ladle hood extraction was necessary.

“Without the help of simulation it would have been financially unfeasible to try as many options to arrive at the most effective option to meet environmental regulations.”

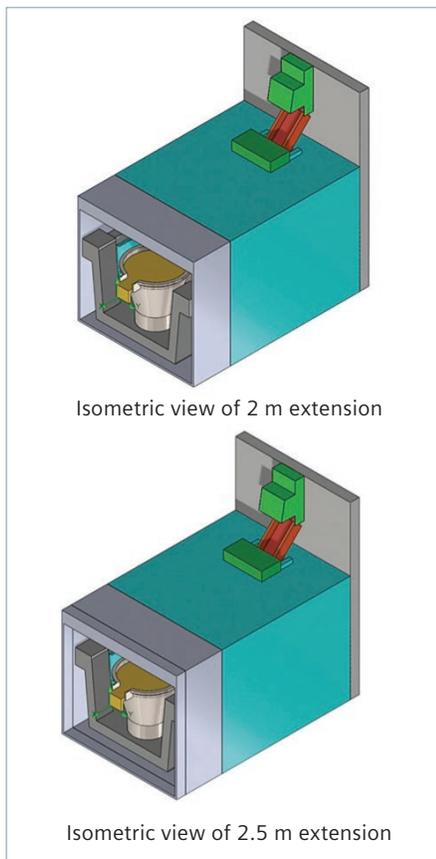


Figure 3. 3D models of the garage extension.

As a result of conducting these analyses the following modifications were highlighted as the most effective:

- 10 mm at 10 m/s air curtain
- Increasing the ladle extraction rate to 18 kg/s
- Extending the garage structure

Without the help of simulation it would have been financially unfeasible to try as many options to arrive at the most effective option to meet environmental regulations.

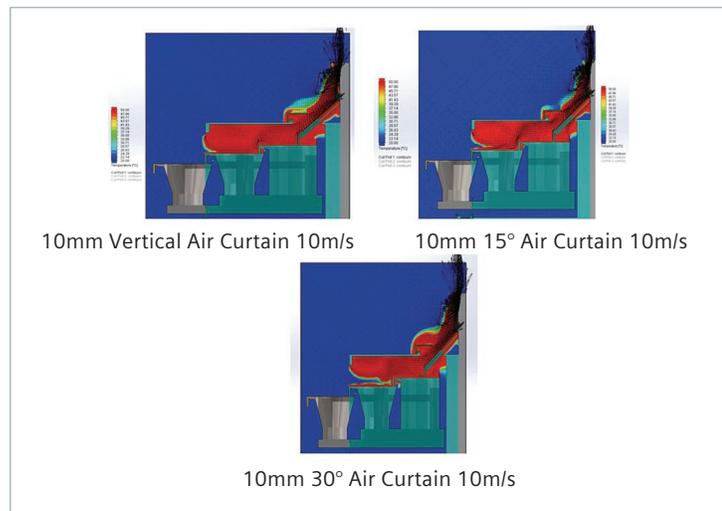


Figure 4. Successful modifications of the air curtain modeled in Simcenter FLOEFD.

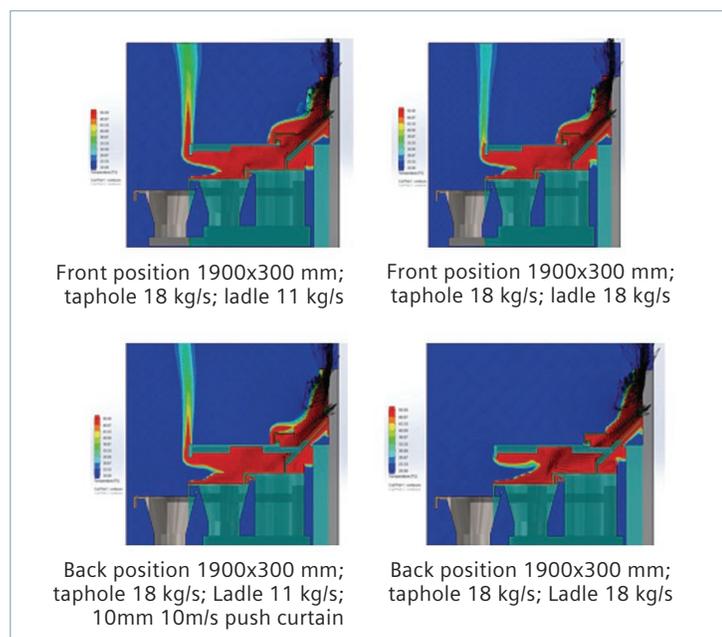


Figure 5. Temperature plots of the ladle hood configurations modeled in Simcenter FLOEFD.

Creating a safer environment for underground coal mines

By Karl de Plessis,
Technical Specialist,
ESTEQ

Continuous miners are machines that mine coal by driving a rotating drum with an arrangement of sharp spikes into a seam of coal and are usually controlled by an operator via a remote coil attached to the machine. Figure 1 shows the development of a heading as the continuous miner is cutting into the coal face. The friction from the cutting action generates heat and can even result in sparks. Add this to a mix of adequate concentrations of methane gas as well as a cloud of coal dust and you have a recipe for disaster. Methane gas can ignite and burn at concentrations between 5 and 15 percent, and at concentrations of 9 percent, the mixtures of oxygen and methane presents the optimum state for an ignition, provided that a significant heat source is present.

Recent incidents of methane gas ignitions in underground coal mines have raised many concerns and questions of “why all of a sudden?”, Coaltech approached ESTEQ for a computational fluid dynamics (CFD) solution to simulate the typical scenarios that could lead to methane ignitions or explosions. ESTEQ provided Coaltech with the NX CAD tool and the Simcenter FLOEFD for NX CFD tool to investigate the reappearance of these methane ignition incidents. CFD simulation with the Simcenter FLOEFD technology was

key in Coaltech’s research and the investigators ability to analyze complex models rapidly.

A typical simulation entails modeling the ventilation inside the heading as it is being developed, complete with continuous miner and scrubber arrangement, shuttle car and methane gas released from the freshly cut coal. Although the work has begun recently, Simcenter FLOEFD has already provided Coaltech with valuable insights into the effects of the various ventilation systems by quickly and easily changing scrubbers or moving the cutting head of the continuous miner up or down and performing “what if” studies.

The reason for the sudden increase in methane ignitions can, on one hand, be ascribed to increased production demands and, on the other hand, poor ventilation design because of the lack of insight as to how the complete system is going to interact. Increased coal demand results in the use of larger machines that are capable of mining more coal at higher rates, which results in more coal dust that has to be removed with larger ventilation, dust suppression and scrubber systems.

The increased ventilation requirements is where the problem arises. Merely increasing the flow rates of the various systems can produce unpredictable behavior and have a

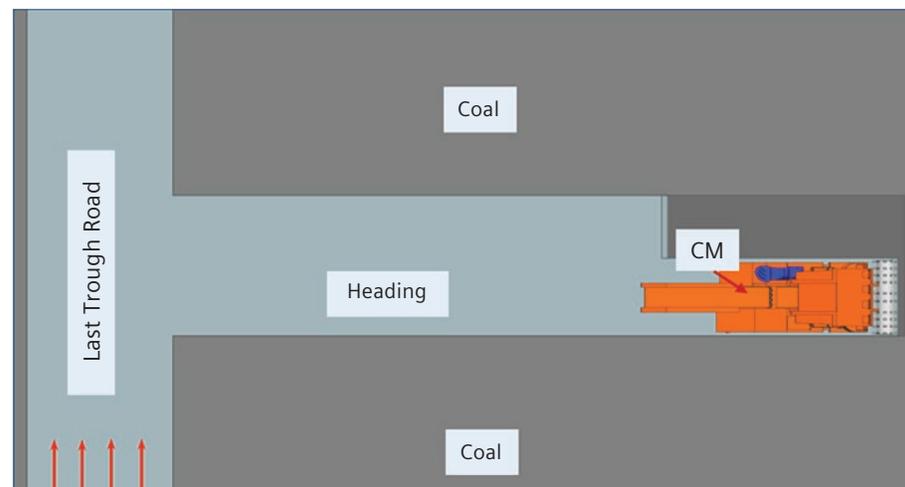


Figure 1. Top view of a heading with continuous miner on the right.

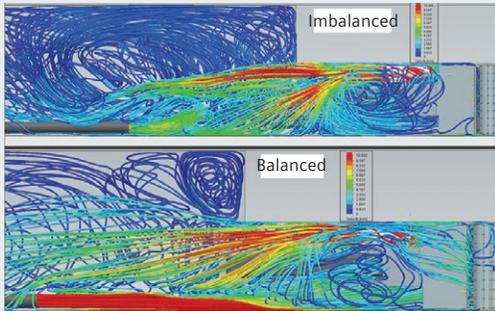


Figure 2. Velocity streamlines of imbalanced (top) versus balanced (bottom) systems.

negative effect on the dilution of methane from the cut face because the system can become imbalanced. Further, careful consideration has to go into matching the different auxiliary ventilation systems, that is, matching a suitable scrubber arrangement with either jet fan, scoop brattice, or ventilation ducting arrangement for the supply of fresh air. Figures 1 and 2 show the effect of an imbalanced system versus a better balanced system, respectively.

In the case of the imbalanced system, the scrubber flow cuts off the fresh-air supply from the duct resulting in a large amount of recirculation of the contaminated air and the build-up of methane. This is evident in figure 2, which shows the iso-surfaces of the methane concentrations. In general, the methane concentrations are lower for the balanced system as the surfaces representing the different levels of concentration encapsulates smaller volumes. In these particular simulations, the rotation of the cutting drum was not considered; thus, the methane is trapped between the drum and the cut-face.

Rotation of the drum will aid in washing out the methane between the face and the drum, provided that sufficient and effective ventilation is ensured. This can also be simulated in Simcenter FLOEFD through the sliding mesh rotation function.

Figure 4 shows a similar scenario (with a different heading and continuous-miner configuration), where the rotation of the cutting drum was simulated. The images are static screenshots of the animations from the transient analyses conducted, but the build-up of methane for the imbalanced system can be immediately identified. Poor penetration of the fresh-air supply results in high levels of recirculation of contaminated air from the

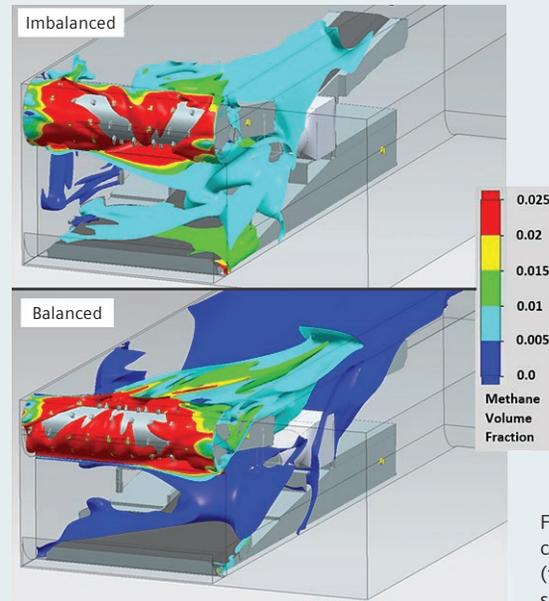


Figure 3. Iso-surfaces of methane concentration for imbalanced (top) versus balanced (bottom) systems.

scrubber to yield the build-up of methane at the cutting face, regardless of the rotation of the cutting drum. The bottom image shows much better dilution of methane. Just to clarify, the slightly higher local methane concentration in the bottom image is attributed to the physical time at which the simulations were stopped. While the imbalanced case showed increasing methane concentrations, up to this point, the balanced case had reached steady conditions by the same time. Now, it is all good and well that complex situations like these can be simulated, but does it have any merit? Can simulations like these really be used to reliably predict what would happen under certain circumstances? Which brings us to the next point, that of verifying the results.

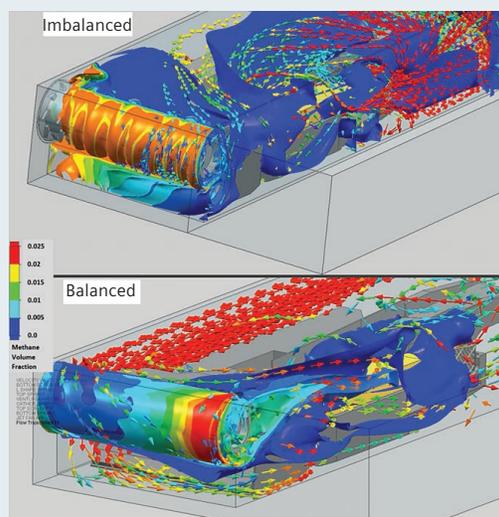


Figure 4. Iso-surfaces and velocity vectors for imbalanced (top) versus balanced (bottom) systems – cutting drum rotation considered.

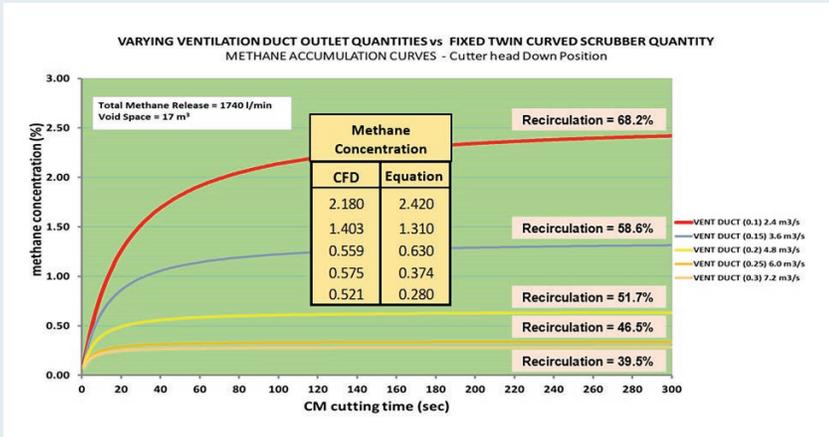


Figure 5. Methane build-up for various recirculation ratios.

Breakthrough in mining ventilation

We need to establish a quantifiable and measurable parameter to verify the CFD results. Considering the flow trajectory images from above, taking velocity measurements to determine the flow profile can seem difficult or even doing some form of smoke tracing because it quickly ends up with no more than a cloud of smoke. In this respect, we can rely on CFD to investigate the interactions of various ventilation system configurations.

By analyzing an extensive number of simulations, made possible by the ease of use of Simcenter FLOEFD, we derived a novel method of determining the actual amount of recirculation. This sounds a lot easier than it really is because it is not immediately apparent exactly how much of the fresh air actually penetrates the cutting region. By performing CFD simulations and making use of the extensive post-processing features available, the amount of recirculation can be calculated and the methane build-up can be estimated subsequently.

Verifying the results

A physical underground test was conducted to verify the simulation results, with controlled levels of methane released and the cutting head moved up and down to simulate typical operation. Comparing the predicted methane levels with actual underground measurements is a more reliable and quantifiable way of verifying the CFD results, as opposed to taking point velocity measurements to derive a flow profile. The scenario considered was with the continuous miner in the right hand sump position, similar to figures 1 to 3.

Figure 6 shows a time-history graph extract from the continuous-miner telemetry during the underground test. The black line represents the cutting-head position as it moves from top to bottom. The blue and red lines are the methane levels detected from the sensors on the left and right side of the cutting head, respectively. We made two observations: one is that the left sensor (blue) constantly measures higher levels than the right side (red), indicating the right to left motion induced by the auxiliary ventilation systems, the other is that the left sensor level peaks whenever the cutting head is in the bottom position (circled), indicating the methane getting trapped under the head as the head moves down. From the graph, it is clear that the process is inherently transient, and conducting such a simulation would pose some challenges. However, with reasonable accuracy, quasi-transient simulations can be done of the methane concentrations measured at the sensors by simulating the transient rotation of the cutting drum, but with the head fixed in either the top or bottom cutting positions.

Figure 7 is a graph of methane build-up over time from the CFD simulation. The graph shows the methane levels averaged across the sensors located all around the continuous miner. Also displayed are the methane levels calculated from datasheets, also averaged across all the sensors. The CFD predictions compare reasonably well with a slight over-prediction when the head is in the bottom position and, conversely, a slight under-prediction when the head is in the top position. This can be attributed to the difference between the quasi-transient modeling approach and the physically transient nature of the cutting sequence.

In the simulations, the head is kept in a fixed position for a longer period of time, which allows for more methane build-up when the head is down and more methane dilution when the head is up. When the head is down, the methane gets trapped because the scrubber cannot effectively draw the contaminated air in as the inlet is obscured, causing the build-up of methane below the cutting head, which results in the higher averaged methane levels. When the head is up, the scrubber inlet is no longer obscured and the contaminated air gets drawn in more effectively, thus aiding dilution of methane. Thus, when comparing the combined effect from the head up and down positions, there is a really good

correlation between the simulated and measured results.

These results prove that Simcenter FLOEFD is capable of producing accurate and reliable results even with such complex systems as underground mining ventilation. The various innovative and advanced technologies within Simcenter FLOEFD have enabled Coaltech to consider the entire system of primary and auxiliary ventilation while maintaining a high level of complexity for the geometry of the continuous miners, scrubbers, shuttle cars and jet fans, etc.

Being able to provide reliable results with confidence has started a paradigm shift in underground mining ventilation, which will lead to more integrated and involved designs where the continuous miner and the ventilation systems will be designed as a complete integrated system from the ground up. Smarter designs for early detection of methane have now been made possible by using CFD to identify the most suitable placement of methane sensors, which would not be possible without modeling the entire system.

The findings based on literally hundreds of different scenarios that have been simulated thanks to the ease of use of the Simcenter FLOEFD technology. Comparisons with actual measurements as part of a verification process provided Coaltech with confidence in the use of FLOEFD as the CFD tool of choice to aid smarter ventilation system designs in the future.

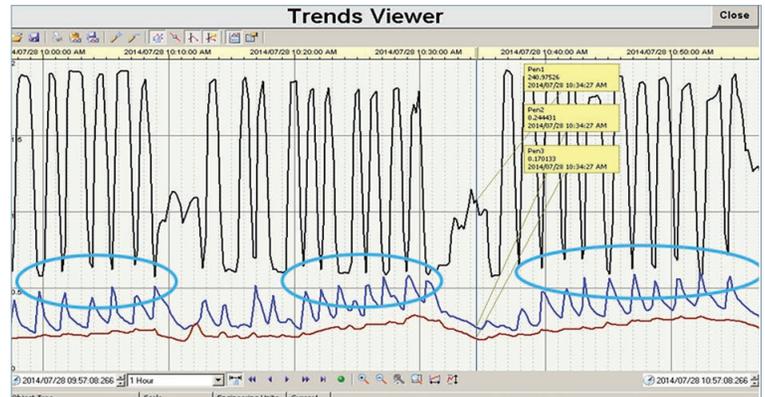


Figure 6. Underground test continuous-miner telemetry. Right-hand sump scenario.

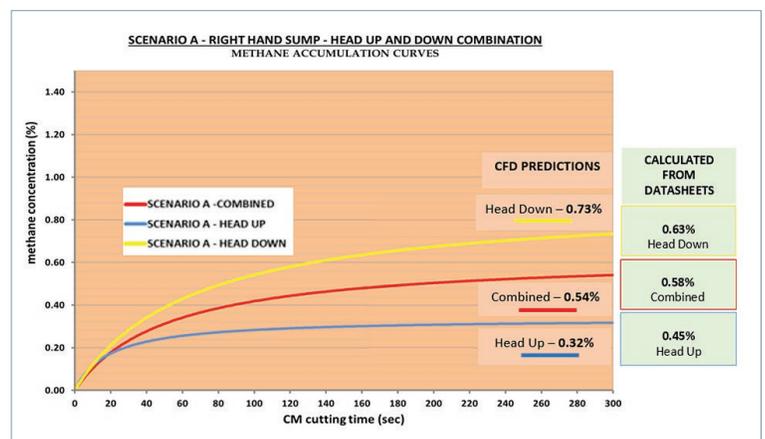


Figure 7. Verification of the average methane levels on sensors comparison.

“These results prove that Simcenter FLOEFD is capable of producing accurate and reliable results even with such complex systems as underground mining ventilation. The various innovative and advanced technologies within Simcenter FLOEFD have enabled Coaltech to consider the entire system of primary and auxiliary ventilation while maintaining a high level of complexity for the geometry of the continuous miners, scrubbers, shuttle cars and jet fans, etc.”

Can we stop this moving train?

By Karl de Plessis,
Technical Specialist,
ESTEQ

Engineers at Transnet Freight Rail in South Africa use Simcenter FLOEFD to investigate the thermo-mechanical response of train wheels. New train wheel designs are governed by design codes such as UIC 510-5 and EN 13979-1. One specific requirement is during braking. Heat is generated while the brake is applied which increases the wheel temperature and consequently causes thermal expansion. The lateral deflections resulting from the thermal expansion are required to be within prescribed limits. Additionally, the residual hoop stress after the cool down period at the end of the test must not exceed the yield stress of the material.

At ESTEQ, we have proposed an approach to solving this problem with minimal effort and within short turnaround time, as outlined in the blog article. Subsequent to the mentioned article, engineers at Transnet Freight Rail adopted this approach and recently put it to good use in a real-world application where a specific wheel had to be investigated for compliance with UIC 510-5 code specifications. The modeling approach and results are discussed and presented below.

Drag braking rig test

The drag braking rig test is outlined in the UIC 510-5, 2007 code as follows: this test must simulate the train traveling at a speed of 80 km/h and the brake being applied for a period of 45 minutes. Thereafter, there is a cool down period of 45 minutes with the train still traveling at 80 km/h. Convection from the external wind is simulated by means of forced rig ventilation and is prescribed by the code as $V_{\text{ventilation}} = V_{\text{train}}/2$. In this particular case, with the wheel considered, the heat generated by the brake was calculated at 15.7 kW (figure 1).

Modeling approach

An accurate distribution of temperature throughout the wheel and axle assembly is required to obtain accurate deflections caused by thermal expansion under these conditions. Considering that the train is traveling at 80 km/h, forced convection is taking place, and because the wheel is turning, the

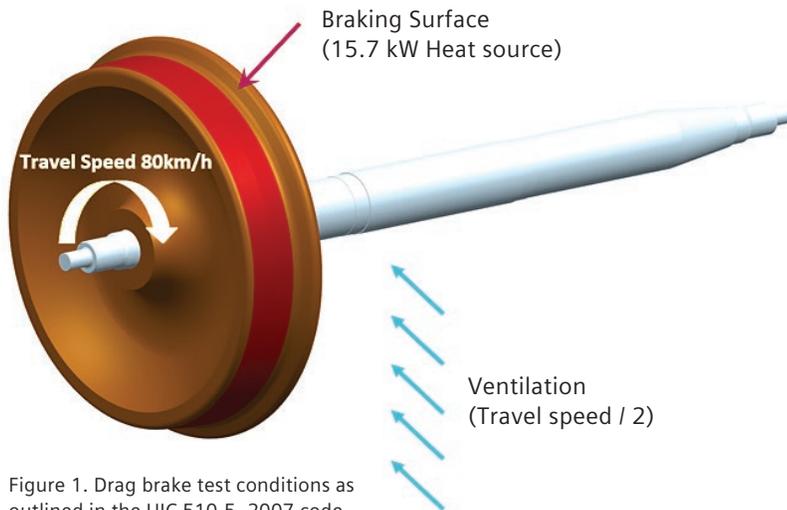


Figure 1. Drag brake test conditions as outlined in the UIC 510-5, 2007 code.

“Simcenter FLOEFD has been instrumental in enabling Transnet Freight Rail to do such thermo-mechanical response investigations with the minimum effort and within the shortest possible time, minimizing the reliance on physical tests, resulting in significant time and cost savings.”

Hendrik Esterhuysen, Transnet Freight Rail

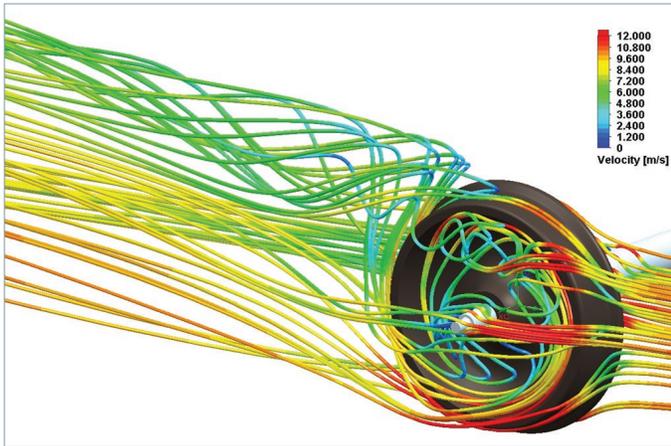


Figure 2. External flow analysis – Velocity streamlines.

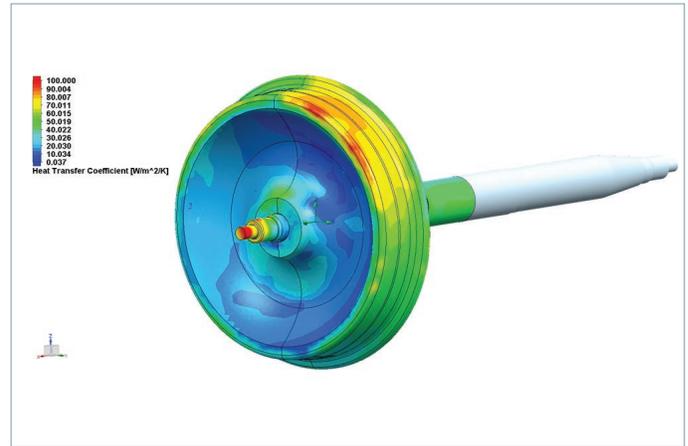


Figure 3. External flow analysis – Heat transfer coefficients.

airflow around the wheel would be disrupted and thus not symmetric around the wheel. Given these conditions, ideally, a detailed CFD simulation should be performed that would solve the conjugate heat-transfer problem, that is, the bidirectional effects of conduction within the solid and the forced convection at the outer surfaces of the wheel, to obtain accurate temperature distributions.

Although Simcenter FLOEFD can solve the conjugate heat-transfer problem, a 90-minute transient simulation is required as per the drag braking test description and performing the full external flow and solid heat conduction would require excessive computational time. When completing various brake rig tests for various levels of wheel wear, shorter calculation times are desirable.

Thus, the analysis was split into two separate steps to speed up the calculation time. First, a steady-state analysis of the external flow without heat conduction in the wheel assembly was performed from which the heat transfer coefficients (HTC) on the surfaces of the wheel and axle assembly are obtained (figures 2 and 3).

The HTCs were then averaged circumferentially and applied as the boundary conditions of the transient analysis. In this case, the circumferential averaging of parameters was deemed acceptable because the wheel is rotating at approximately 700 rpm, for which a single revolution occurs in less than 0.1 seconds, hardly enough time for any measurable temperature changes to occur.

Second, a transient internal heat conduction analysis was performed that ultimately produces the temperature distributions required for the finite element analysis. During this step, the external flow was neglected but accounted for by the outer-wall HTCs boundary condition from step one. The 15.7 kW heat generated from braking was applied to the braking surface by means of a surface heat source. Figure 4 shows the thermal response of the wheel and axle temperatures with respect to time during the heating and cool down phases of the drag braking test.

The resulting temperature distributions from the hot (45 min) and cold (90 min) states

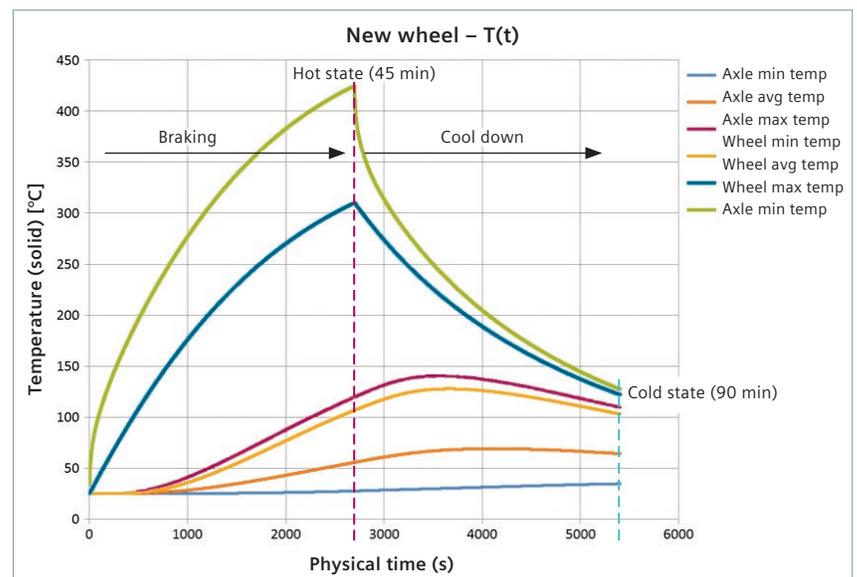


Figure 4. Transient thermal response of the train wheel assembly.

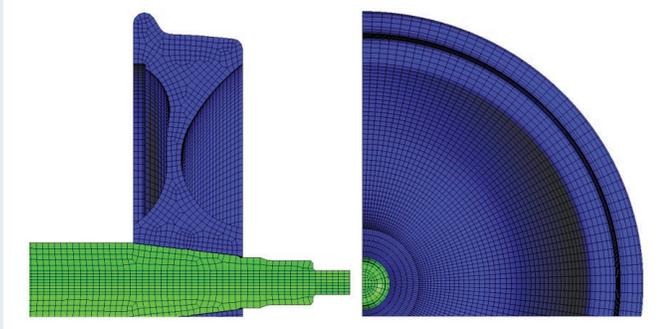


Figure 5. MSC.Nastran finite element mesh, created in MSC.Patran.

were then seamlessly mapped onto an existing Nastran finite-element mesh of the wheel assembly. Figure 5 shows the finite-element mesh of the wheel assembly.

Figures 6 and 7 show the resulting temperature distributions from Simcenter FLOEFD and the subsequent exported temperatures on the Nastran mesh in the hot and cold states. It is clear that the mapping was a success.

Finally, the thermo-structural analysis was conducted by applying the solid temperature distribution in addition to the interference fit between the wheel and axle as prescribed by the UIC 510-5 code. The hot-state deformation and cold-state residual hoop stresses are shown in figures 8 and 9. The lateral displacement in both the positive and negative directions have to be within the limits prescribed by the code for a wheel to pass the specification and be accepted. Although, we cannot reveal whether this particular wheel passed the test or not for proprietary reasons, figure 9 illustrates that the residual hoop stresses are well below the yield stress of the material.

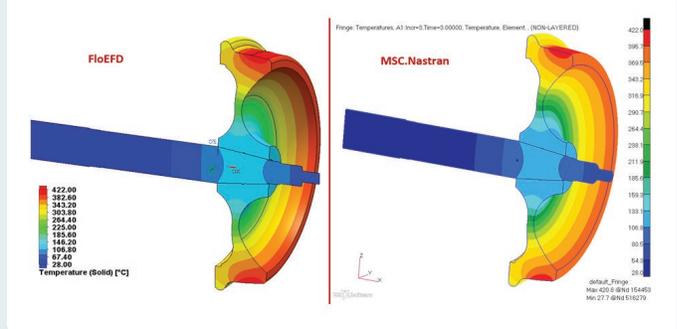


Figure 6. Simcenter FLOEFD to Nastran export – hot-state temperature distribution.

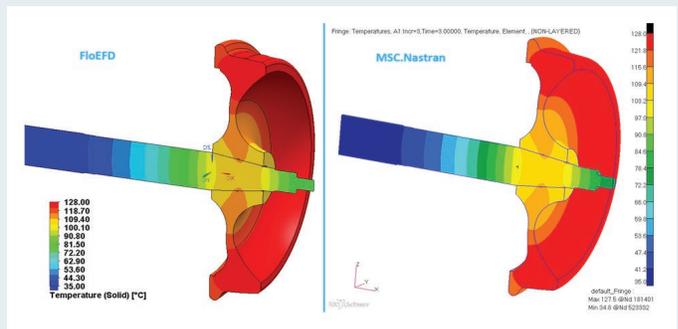


Figure 7. Simcenter FLOEFD to Nastran export – cold-state temperature distribution.

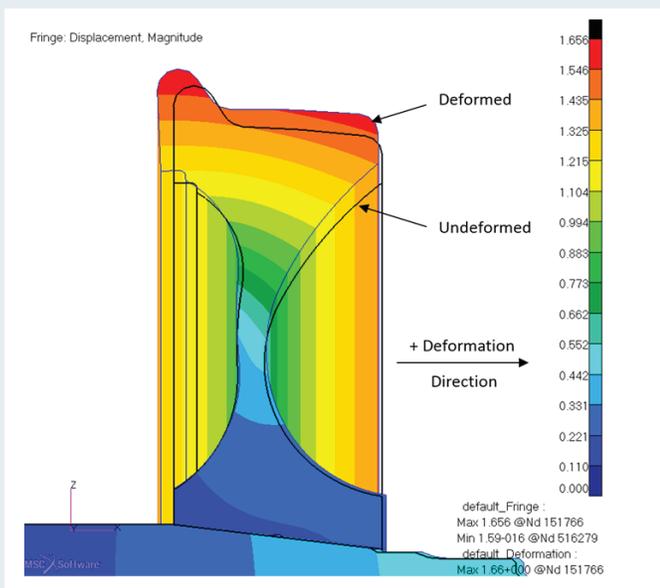


Figure 8. Hot state – deformation.

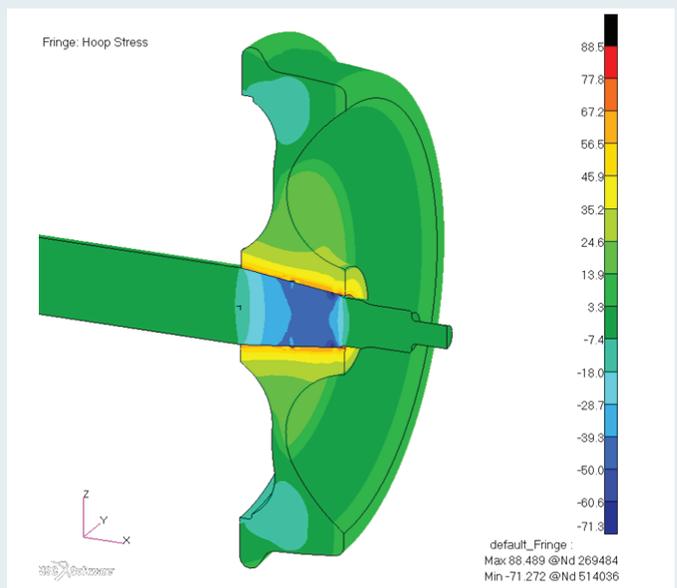
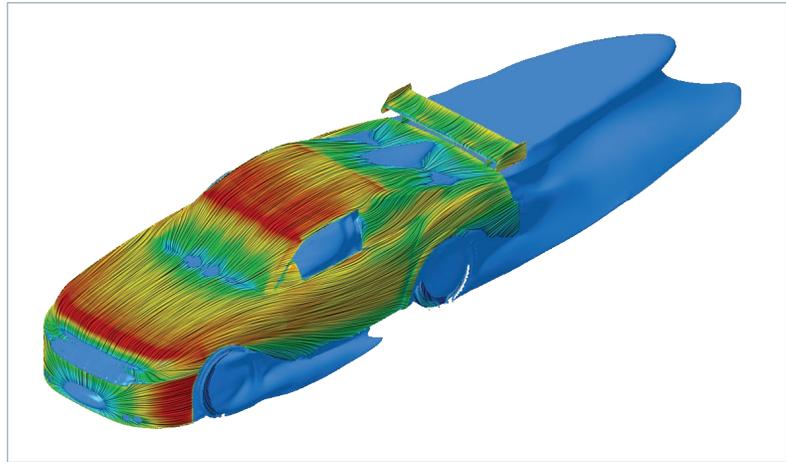


Figure 9. Cold state – residual hoop stress.

Collaboration: The key to engineering a championship winning car

The National Association for Stock Car Auto Racing (NASCAR) has a special place in the hearts and minds of most Americans. And that popularity is spreading. NASCAR Whelen Euro series is now a very successful franchise and just as exciting. Recently Anthony Kumpen, the overall winner of the 2014 NASCAR Whelen Euro Series season talked to Siemens Digital Industries Software about his collaboration with Voxdale, a Belgian engineering consulting team. Managed by Koen Beyers, the CEO of Voxdale, the team at Voxdale used Simcenter FLOEFD thermal and airflow simulation software to model his car's performance on the race track. As NASCAR is a closed racing series with a tight rule book and stringent regulations, the team took advantage of simulation to gain insight into the behavior of the car. The collaboration between engineering and the driver allowed the NASCAR team to proactively improve the car's performance. Watch this short interview <http://go.mentor.com/4OU9m> with Anthony and Koen.



“It’s a lot more than just driving fast in the car. It’s just making the car faster. Improving everything. That’s what racing is all about.”

Anthony Kumpen, 2014 NASCAR Whelen Euro Series Champion



National Institute for Aviation Research (NIAR): Using CFD tools to develop a real-time flight model

Since its inception in 1985, the National Institute for Aviation Research (NIAR) at Wichita State University has made a name for itself as the most capable university-based aviation research center in the United States.

The traditional development of aircraft of any type usually goes through a long design process until the first full-scale prototype is built to test flight behavior. Dr. Olivares and his team at the NIAR set out to develop a method to increase the prediction accuracy of flight behaviors with the real-time flight simulator, MIURA. Conventional aerodynamic calculations used by the simulator were not accurate enough including

predicting stall and other effects such as propeller performance or the wing-fuselage interference.

The NIAR team conducted several simulations with Simcenter FLOEFD for CATIA V5 on their test model, a push propeller UAV, in order to get more accurate data to feed into the simulator for better prediction of flight characteristics. The goals of the research team were to better predict the aerodynamics as well as take into account more models (such as the controls and electrical systems) in the virtual engineering environment while collecting data throughout the flight envelope to feed

back into stress simulations of the aircraft's structure.

Dr. Olivares and his team were able to improve the simulator flight characteristic drastically with the help of Simcenter FLOEFD. The improved accuracy of stall prediction, propeller performance and interference effects, enabled the team to conduct the first steps to develop a virtual engineering method that is superior to the traditional engineering method with regards to product development time and costs.

For more details about the NIAR and comparison data please read <http://go.mentor.com/4Phz3>.

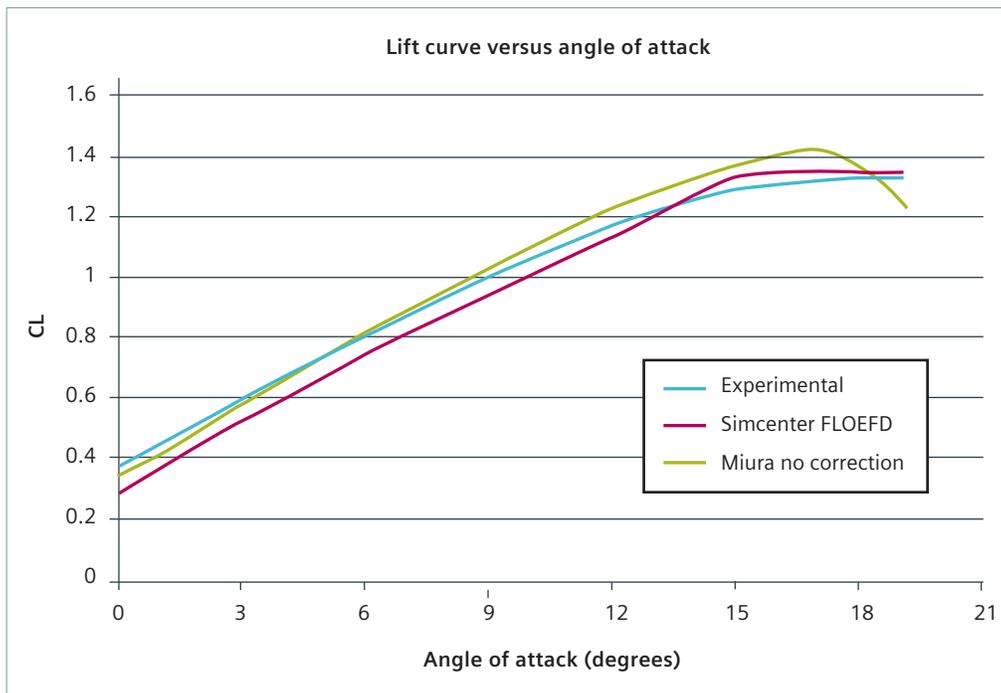


Figure 1. Simcenter FLOEFD has the same lift curve slope and stall pattern as the wind tunnel data. MIURA on the other hand captures the initial slope but over-predicts the stall at 17° AOA.



“The four distinctive features that make Simcenter FLOEFD the best candidate for this kind of application are its CAD embedded approach, the Immersed Body Meshing technology, the parametric study and the solver accuracy.”

Dr. Gerardo Olivares, National Institute for Aviation Research

About Siemens Digital Industries Software

Siemens Digital Industries Software, a business unit of Siemens Digital Industries, 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, we work 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 our products and services, visit [siemens.com/plm](https://www.siemens.com/plm).

Headquarters: +1 972 987 3000
Americas: +1 314 264 8499
Europe: +44 (0) 1276 413200
Asia-Pacific: +852 2230 3333

Restricted © Siemens 2019. Siemens and the Siemens logo are registered trademarks of Siemens AG. Femap, HEEDS, Simcenter, Simcenter 3D, Simcenter Amesim, Simcenter FLOEFD, Simcenter Flomaster, Simcenter Flotherm, Simcenter MAGNET, Simcenter Motorsolve, Simcenter Samcef, Simcenter SCADAS, Simcenter STAR-CCM+, Simcenter Soundbrush, Simcenter Sound Camera, Simcenter Testlab, Simcenter Testxpress and STAR-CD are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries or affiliates in the United States and in other countries. All other trademarks, registered trademarks or service marks belong to their respective holders.

78126-C4 6/19 C