Air Dynamics Simulation of the Tamturbo Oil-Free Air Turbo Compressor

By Timo Pulkki, Mihail Lopatin, Tamturbo Oy, Tampere, Finland & Alexandr Nikulin, Axis Engineering (Mentor Graphics distributor), Saint Petersburg, Russia & Tatiana Trebunskikh, Mentor Graphics, Moscow, Russia

Nowadays to be competitive in the air turbo compressor industry, the company must provide high performing hardware with the lowest life cycle cost. In most cases to achieve such characteristics the company has to use a range of innovative technologies. Tamturbo Oy, was founded in 2010 in the Tampere region, the birthplace of several compressor innovations, has achieved their goal of transforming their view of oil free technology into a worldwide success story. They are fully committed to delivering solutions that bring the highest life cycle value to their customers.

Figure 1. The Tamturbo compressor

Performance prediction of the prototypes of air turbo compressors is one of critical questions during the design process. As soon as such predictions are available, the design engineers are able to optimize the device, taking into account a number of factors like the cooling of the compressor stages or cooling of the electrical engine driving the compressor. Moreover all issues with new design should be discovered as soon as possible before hardware testing, to reduce development time and cost. The required changes can be minor or major for larger parts. Without dependency on the scale of the changes, a deeper understanding is required into processes inside the particular parts as well as the whole device. This is the reason Computer-Aided Engineering (CAE) software and Computational Fluid Dynamics (CFD) software in particular are so popular in the modern industrial world. CFD in particular is more effective in investigating directly, during the product design process as an integral part of the product lifecycle management (PLM) in the early development stages. There are several approaches in CFD including traditional and frontloading. In addition to relying on the vast knowledge and experience in CFD, the traditional approach usually requires transferring geometry from a CAD system to CFD software, via different exchange formats, where some issues with geometry can occur. Issues such as cleaning and healing geometry to make it suitable for mesh creation and manual mesh generation with focus on boundary layers. Investigations of a wide range of designs using such an approach is very time-consuming and as a result only a few particular cases are examined by CFD experts after all changes are made by the design engineer.

The frontloading approach presented in Mentor Graphics' FloEFD™ tool, is intended for use in the early design development stages by design engineers. There are two main principals in FloEFD: direct use of native CAD as the source of geometry information, and a combination of full three-dimensional CFD modeling solving Favre-averaged Navier-Stokes equations with simpler engineering methods in the cases where the mesh resolution is insufficient for direct simulation.

To overcome a traditional CFD code restriction of having a very fine mesh density near walls in a calculation domain, FloEFD describes boundary layer, with the "Two-Scale Wall Functions" method including the near wall functions and the sub-grid model of the boundary layer.

For the simulation of the compressor, the rotation model should be used. There are two local rotation models in FloEFD – circumferential averaging and sliding. The circumferential averaging approach is employed for calculating transient or steady-state flows in regions surrounding rotating solids, which are not bodies of revolution (e.g. impellers, mixers, propellers, etc). To connect solutions obtained within the rotating regions and in the non-rotating part of the computational domain, special internal boundary conditions are set automatically at the fluid boundaries of the rotating regions. The rotating region's boundaries are sliced into rings of equal width and the values of flow parameters, transferred as boundary conditions from the adjacent fluid regions are averaged circumferentially over each of these rings.

The sliding rotation model produces a time-accurate unsteady solution of the flow fields, where the rotor-stator interaction is strong. The sliding technique takes into account the relative motion between stationary and rotating regions. Rotor and stator control volume (CV) zones are connected with each other through "sliding interface". During the calculation, zones linked through "sliding interface" remain in contact with each other. The sliding interface has CVs on both sides and as a consequence each face of the sliding interface has two sides belonging to both rotor and stator zones. All these techniques allow FloEFD to be used for the calculation of air turbo compressors.

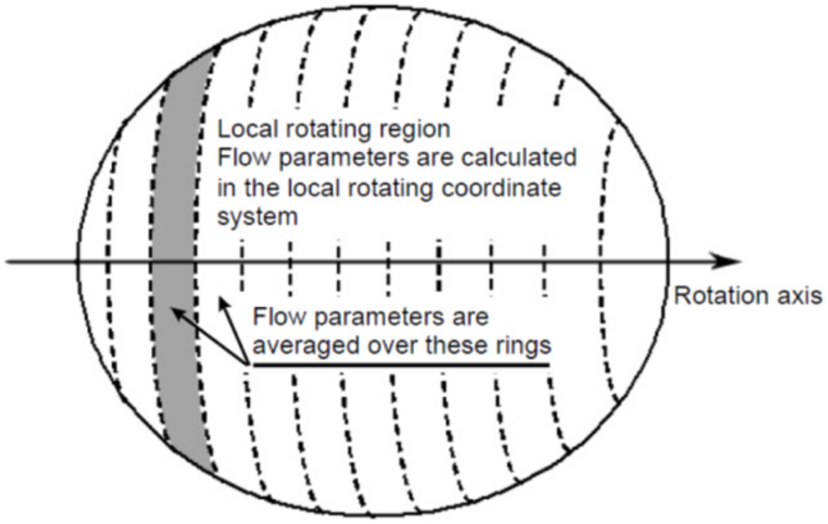

Figure 2. Ring creation in the Circumferential Averaging Rotation Approach

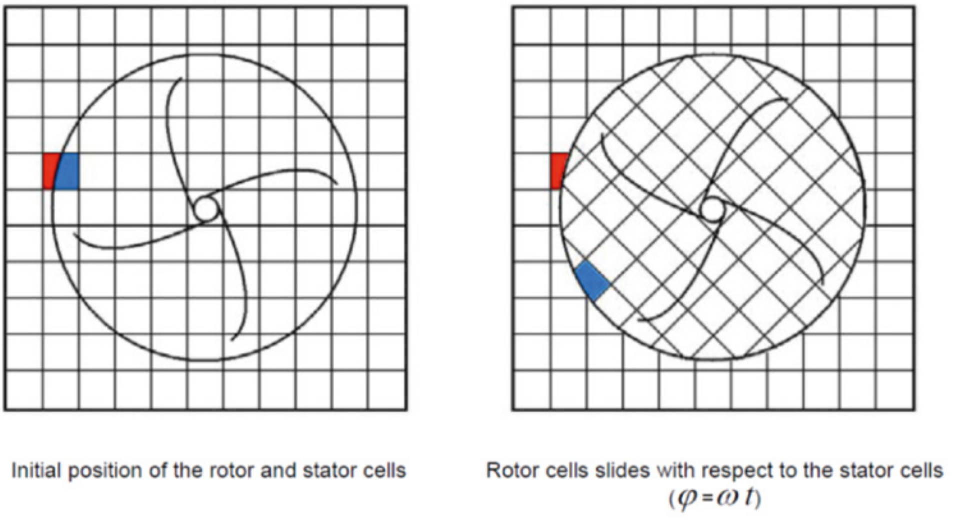

Figure 3. Sliding Rotation Approach

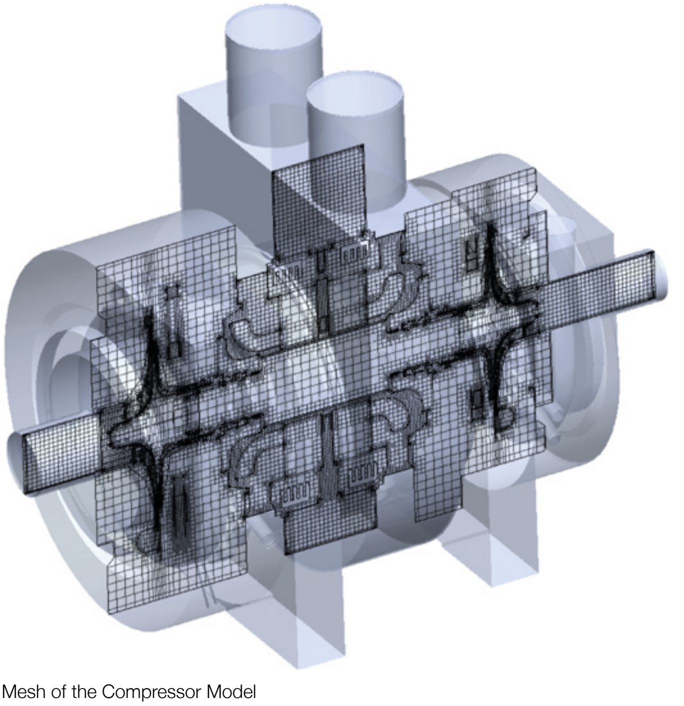

Figure 4. The Mesh of the Compressor Model

This article shows a great example of the two stages of oil free compressor investigation, completed by Tamturbo Oy with support from Axis Engineering.

For analysis, the full assembly of the compressor was used, the construction of which includes several contours: compressor flow area, and several cooling contours. Each contour includes components manufactured from different materials, such as titanium steel or aluminum, which were chosen based on the results of the preliminary calculations of thermal loads and strengths of the various parts of the compressor.

The geometry of flow area has been determined by required compressor characteristics and has a very complex internal structure. The CFD analysis of the compressor, where final pressure rise ratio was calculated, allows for the assessment of air temperature rise. To decrease the temperature of the compressor's body, Tamturbo engineers added the cooling system for the compressor with a complex internal structure.

All geometry features of the compressor, presence of the rotating parts and special requirements for temperature in some critical places like bearings, volutes, shaft and so on, predetermined mathematical models which were needed for the analysis.

The turbulence model used in the analysis can detect a flow regime and switch the mode between laminar and turbulent automatically. This solver is unique and allows an engineer to obtain a solution inside narrow channels even on a coarse grid. The calculation was made taking into account conjugated heat transfer, where heat transfer equations were solved in solid bodies. The radiation model has been disabled due to relatively low temperatures.

In the analysis of the flow, to consider the rotation of the impellers, special rotating model "sliding" was used, which helps to predict characteristics of any type of turbomachines with better quality.

As boundary conditions for the airflow inside the compressor were specified so were the conditions for pressure at the volutes outlets, mass flow rate at their inlets, and shaft rotating speed. For cooling paths, pressure difference between inlet and outlet boundaries was determined. To simplify the thermal analysis of the engine and the shaft, a heat emission process was simulated by adding heat sources with constant power

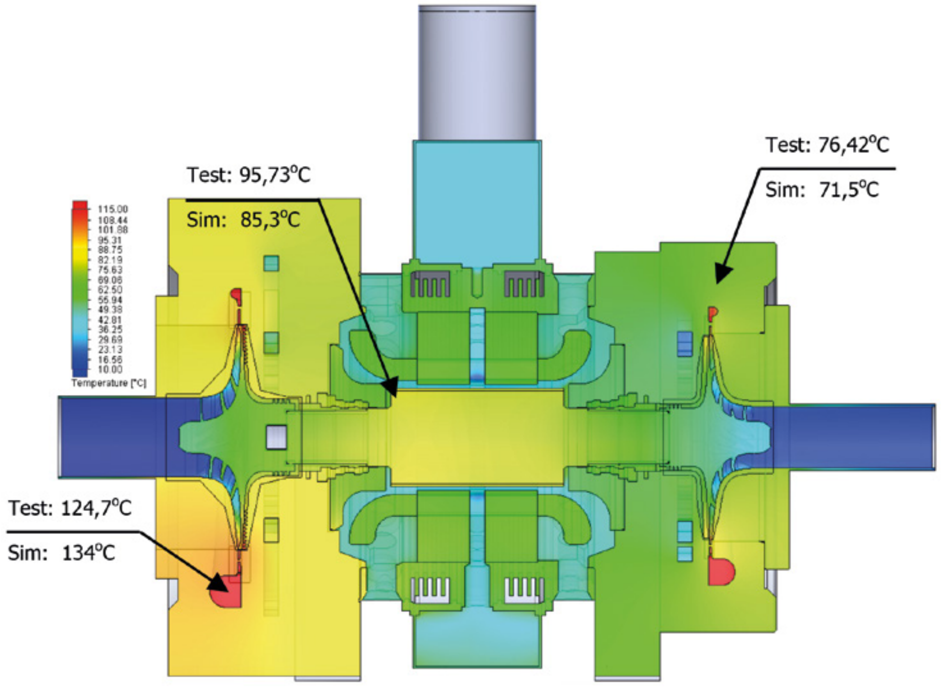

Figure 5. Temperature Distribution in the Longitudinal Section of the Compressor

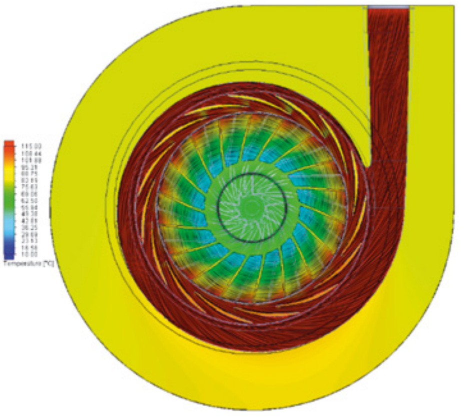

Figure 6. Stage 1. Temperature Distribution in the Cross Section of the Compressor with Streamlines

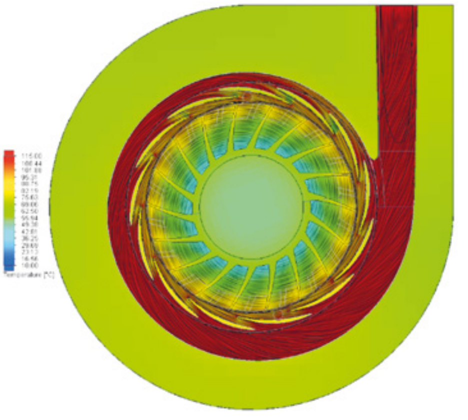

Figure 7. Stage 2. Temperature Distribution in the Cross Section of the Compressor with Streamlines

values instead of using the FloEFD feature emulating a real Joule heating process.

Simulation of conjugate heat transfer process for all zones and units of the compressor, allowed for the investigation of their mutual influence and predicting the thermal state of the compressor components.

The most important parameters under investigation were air temperature before the bearings the volute and the shaft temperature. Mainly bearings work properly when their temperature is lower than 100-120°C, but on the other hand the temperature of the compressed air can be higher than 200°C. And of course every calculation has to predict the compressor pressure rise ratio for the required mass flow rate range.

Initially homogeneous structured mesh was created on the entire geometry, where the number of cells along the main axis of the compressor did not exceed 90. The number of cells for the other axis was chosen in the way that the final size of the basic cells was similar in all three dimensions. After creating the basic mesh, some local regions for better mesh resolution around compressor impellers and in the bearing's area were added. Additionally mesh generator settings were specified to allow for automatic detection of the narrow channels and further splitting mesh in these areas. As a result the total number of cells in the entire two-stage machine was approximately 4.9 million.

The analysis was run in the transient regime on the usual workstation for CFD calculation:

two CPU with frequency 3.1 GHz and only 8GB RAM were used for the task. As a result the simulation of 30 seconds of physical time took only 1.5 days of computer time.

According to the calculation results, the additional seals have been designed and the optimal mass flow rate of the cooling agents were chosen. The temperature in the bearings, was in the range of 60-80 degrees in the calculation results. Tests showed the temperature rise in the nodes close to the bearings up 7-12 degrees.

The difference between pressure values of simulation results and natural tests of the real equipment was not more than 3-5%.

Tamturbo Oy uses FloEFD for a wide range of other tasks, from the turbo compressors developing area, closing all questions of aero and thermal calculations. The quality of the results was proved by the series of experiments which were provided by Tamturbo Oy and a number of completed projects. The software has several direct interfaces with other codes which are used in the company, and as a result of this fact and due to the unique solver technology as well as all other features mentioned above, the development product lifecycle and its costs were significantly reduced without compromising the quality of the results.

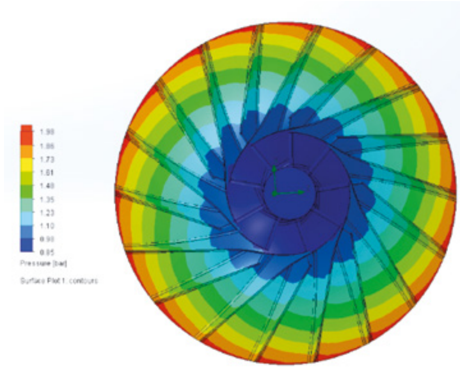

Figure 8. Stage1 impeller. Static pressure distribution on the surfaces at normalized mass flow rate 0.8, normalized rotational speed 0.943. Pressure rise ratio - from simulation 1,9718/ from test 1,9723

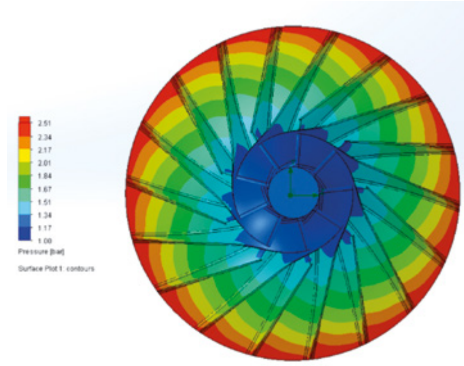

Figure 9. Stage1 impeller. Static pressure distribution on the surfaces at normalized mass flow rate 1.5, normalized rotational speed 0.943. Pressure rise ratio - from simulation 1,9585/ from test 1,9589

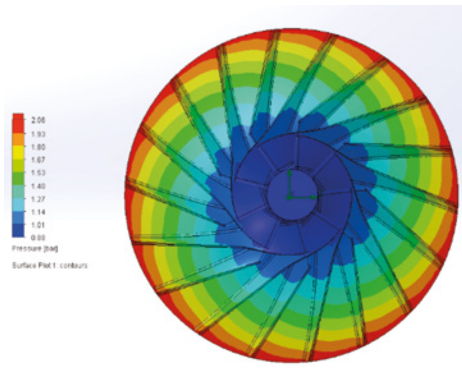

Figure 10. Stage1 impeller. Static pressure distribution on the surfaces at normalized mass flow rate 0.9, normalized rotational speed 0.97. Pressure rise ratio - from simulation 2,0391/ from test 2,0302

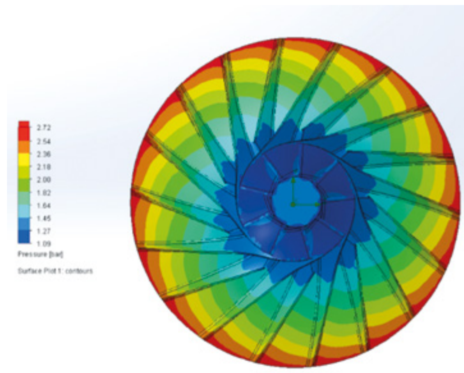

Figure 11. Stage1 impeller. Static pressure distribution on the surfaces at normalized mass flow rate 1.6, normalized rotational speed 0.97. Pressure rise ratio - from simulation 2,0083/ from test 2,0057

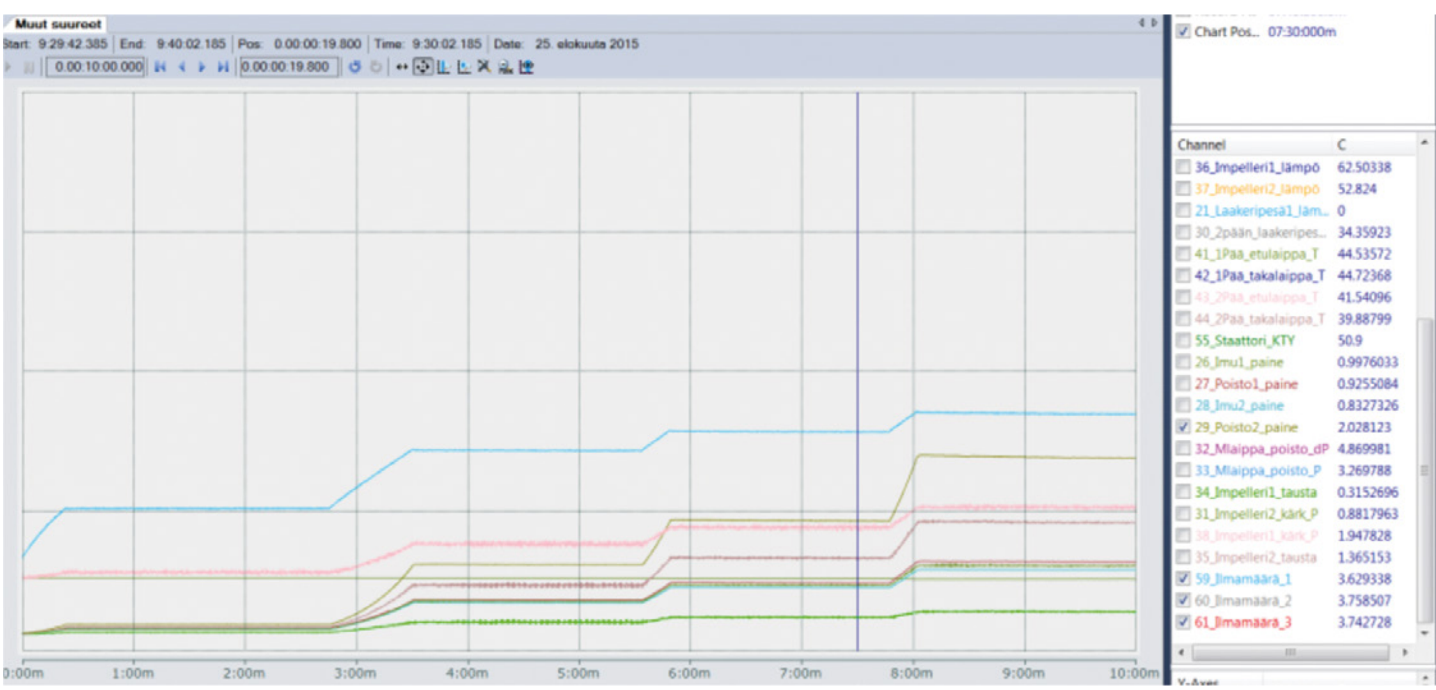

Figure 12. Dynamics of different parameters from the final compressor test.

Dr. Uwe Lautenschlager
Senior Technical Expert - Simulation

Q. Tell us about Continental AG
A. Continental can look back on a successful history for 145 years, founded in 1871, when the company started manufacturing soft rubber and rubberized fabrics. Although most consumers recognize Continental for its tire division, we are a leading diversified German automotive manufacturing company specialized in brake systems, automotive safety, powertrain and chassis components, as well as instrumentation or driver HMI, connectivity, infotainment, access systems and many more. Today, Continental has sales above €39 Billion with around 215000 employees worldwide. My place is inside the "Infotainment & Connectivity" Business Unit's Mechanical Engineering team within the Interior Division. We are located here in Wetzlar, Germany, a town known for optical and fine-mechanics industry.

Q. Tell us about what you do in your role at Continental?
A. I started my career at our Wetzlar site in 1999 as a structural analysis specialist. However, the task quickly extended into the electronics cooling and thermal simulation area. Today I manage our BU's mechanical simulation team which is based at several sites around the world. It covers optical, structural, EMC, molding and thermal simulation of our products. My team role is to innovate the simulation processes by setting the simulation methodology, tools and knowledge sharing and targeting the same level at all sites. Nevertheless, I'm always passionate about developing models and doing simulations by myself. Besides my work in the simulation team, I'm also fortunate because Continental supports me to be an "ambassador" at the nearby University of Siegen where I have been giving lectures on "Modeling and Simulation" since 2009. The students love the close connection to Industry they can get from my hands-on course.

Q. What are the products that you and your team are involved with?
A. Speaking about our Business Unit, our product portfolio includes radios, which have evolved from simple tuners and amplifiers to proper entertainment hubs, multimedia systems, and telematics modules. Connectivity devices allow the communication between the vehicle and the internet and, in future, increasingly to other peers as part of the Vehicle-to-X (V2X) trend. A permanent exchange of data from the vehicle is aimed in our "Always On" vision. Infotainment is a fast growing part of the automotive industry and is rapidly changing from year to year in both car connectivity and car communications. Our products have to deliver new answers for future mobility and intelligent technologies, they are networked together by taking on more driving tasks.

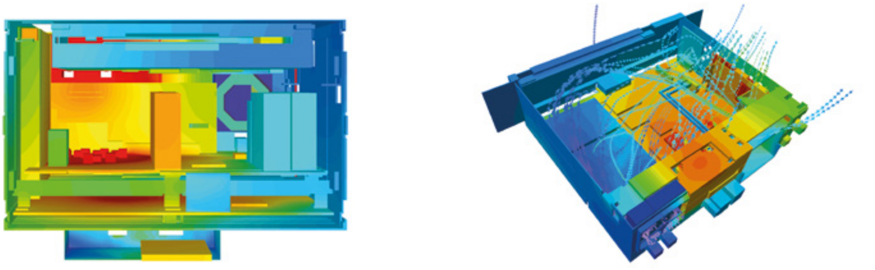

Figure 1. Thermal analysis of an infotainment system with multiple components and devices.

Q. Do you work mostly within your Business Unit or does your work stretch to other Continental Business Units in Germany or even Worldwide?
A. We have two aspects here, one being the integration of products across our division in the future. The second aspect is our simulation network. Not only do we have our experienced and agile simulation team, we have established a close simulation network and User and Working Groups across Continental for several disciplines. For example, I'm the speaker and organizer of our FloTHERM® User Group. We meet regularly with all international users in varying locations and support each other across all divisions if necessary and possible. Our BU's simulation team itself is positioned worldwide like in Singapore, France, Germany, Romania etc. so it really is a worldwide working group. Our team has the desire to serve our customer's, (internal and external) needs quickly and reliably; we have a high documentation standard in order to provide the same high quality in all our simulation results.

Q. How different are the requirements or thermal limits of your products and how different are the environmental boundary conditions for them?
A. Thermal limits become increasingly obvious as package dimensions shrink, power dissipation increases locally or globally and packages have lower temperature specifications, e.g. from consumer electronics. Boundary conditions as part of the requirements are given by the OEM and can therefore vary according to the standards they specify. The ambient temperatures vary, the requirements on housing temperatures and the thermal management options differ and are dependent on the location of where the product is installed (roof, dashboard etc.). Since our infotainment systems can be found in trucks, we have to fulfil a higher lifetime expectation (25,000 hours) for them compared to cars (with around 10,000 hours being more normal). Hence, the overall variation is more between the OEM or vehicle type and we try to re-use the platforms across OEMs to keep the costs for development to a minimum. There are many parallel thermal challenges nowadays, not "only" designing a good amplifier heatsink as maybe in the past.

Q. When you use FloTHERM in your work, how many simulations do you run on one product on average until the design has matured?
A. In 2000 we chose FloTHERM as our thermal simulation tool, convinced primarily by its quality and robust solution capabilities. We use FloTHERM to run thermal simulation very quickly and it is deeply integrated into our design process. It allows us to do lots of variant simulations very fast. If my boss says "we need something today", I can get a satisfactory answer quickly. Productivity is important but it depends on what type of answer is needed, or how much detail and data to include. In the very early phase of a product's design, such as the quotation phase, the simulation detail is secondary as the product cannot be defined as accurate as compared to the final product development stages. So it is not a number, it is a level of confidence that can be achieved rather quickly.

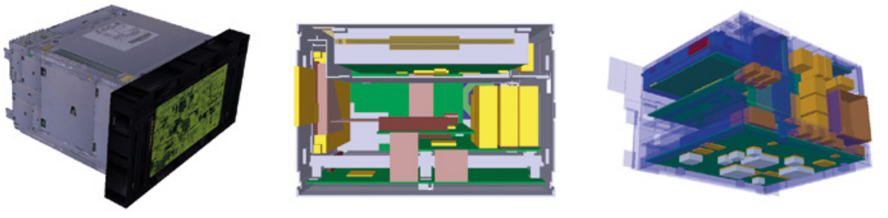

Figure 2. A FloTHERM model with texture map and the internal view of an assembly with its components and modules.

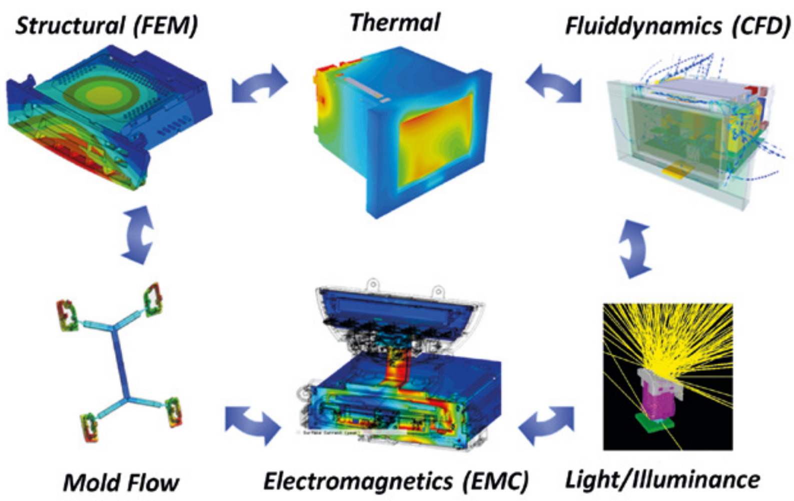

Figure 3. The overall connectivity of various simulation disciplines in multimedia system simulations.

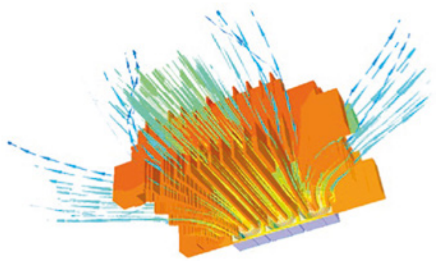

Figure 4. Thermal analysis of an infotainment system with multiple components and devices.

Q. Do you think the products you develop today would be realizable without simulations?

A. We always need to understand our products - that's where simulation tools come in. Of course, developing a product on time and to cost was a goal when there was not much virtual analysis done 20 years ago. Often the products were overdesigned and we had many prototypes and samples created. Today, however, we can design the products closer to their limits as we understand the physics behind them better through simulation. The development cycle we have today could not be done without simulation. Our first shot has to be correct. A good simulation is also not only about being correct, but it also helps to understand the product. The design of our automotive multimedia systems has to fulfill requirements from mechanical stability and thermal management to electromagnetic compliance, hence a multi-disciplinary design and optimization problem. Conflicting objectives among these disciplines require a concurrent design approach with different modeling levels, parameter variation (e.g. DOE techniques), optimization techniques and the consideration of uncertainties and changes within the design process. Such up-front simulations enable us to test and modify system and components as virtual prototypes before performing physical tests and building real prototypes.

Q. How important is it to interact with the other simulation groups from electrical and circuitry design teams and how often do you need to communicate the changes and the results?

A. Extremely important! Every change with a thermal benefit can greatly impact the structural behavior or Electromagnetic Compatibility (EMC) and electrical circuitry aspects. A larger heat sink in the back of the system could increase the vibration and cause rattling for instance; or moving a processor closer to a colder zone for better cooling, but also closer to a high frequency zone near an antenna for example can cause EMI issues. As said before, we have a multi-disciplinary design problem.

Q. Where do you see electronics thermal simulation in the future? Will there still be physical testing or will it be all virtual testing through simulation?

A. We typically skip the A-Sample that we had in the past and almost directly go into the tools as we expect already a good sample from the simulation. Potential design risks are typically identified by simulation early enough and eliminated. This is besides the many tests we put our devices under with physical laboratory stress tests from long term use of buttons, shock and random vibration to environmental tests in climate chambers. For us the testing will not be replaced in any foreseeable future; final qualification testing will always be done on the matured design in a laboratory. However, the number of prototypes is reduced and the risk of failure should be at a minimum.

Q. When thinking about the future, where do you see the biggest challenges in your work with the future technology developments that will go into your products?

A. Several years back CPUs generated 1-2 W of heat in a car; now they are more in the range of 10W. We are also seeing the trend towards bigger touch-screen displays and multimedia systems are getting more and more condensed into smaller spaces of non-rectangular shape. Those are some of the technical challenges we face, additionally we must put a big focus on developing the skills of our simulation team to be able to understand and handle new technologies and simulation tool features accordingly. I always see the resource aspect as we will never have enough simulation people to do everything that could be done, but we do everything that's necessary.

Q. Do you have any goals over the coming years to increase your simulation productivity?

A. Our daily work is changing faster than ever. Improving the post processing and documentation of simulation tools and results will help us to communicate answers more efficiently to our customers. We try to reduce the time for modeling and documentation and spend more time for creativity and solutions. We want to implement highly efficient workflows with the right tool landscape. Continental is on the way to being a digital company for sure.

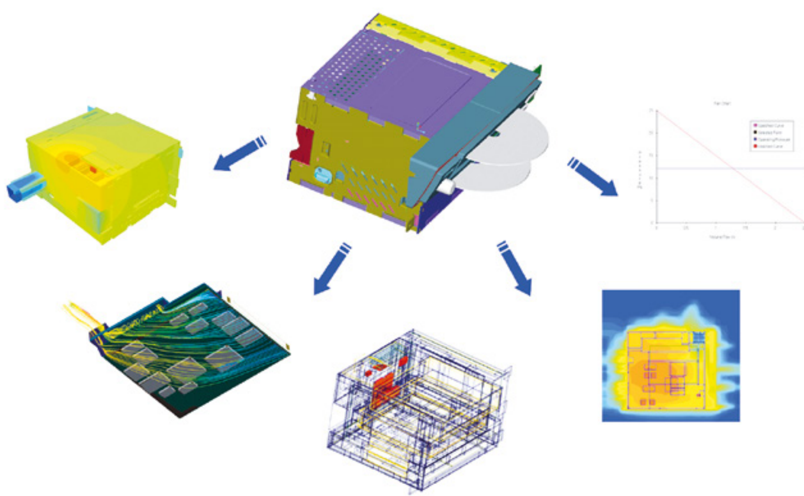

Figure 5. Showing a range of simulation tasks and result evaluations in an infotainment thermal simulation.