Siemens Digital Industries Software

Air dynamics simulation of the Tamturbo Oy oil-free air turbo compressor

Leading Finnish manufacturer uses Simcenter FLOEFD to realize vision of oil-free air turbo compressors

Executive summary

Simcenter™ FLOEFD™ software makes it possible to analyze turbo compressors at all development stages, including early stage, and allows for on-time design improvements. As a result, the development product lifecycle and its costs are significantly reduced. The combination of being embedded into computer-aided design (CAD) systems, smart cell technology and usability allows Simcenter FLOEFD users to focus on the device's design rather than on operating the software.

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

Abstract

To be competitive in the air turbo compressor industry, companies must use a range of innovative technologies to provide high-performing hardware with the lowest lifecycle cost. Founded in 2010, Tamturbo Oy is a Tampere, Finland-based oil-free air compressor manufacturer. The Tampere region is the birthplace of several compressor innovations, fertile ground for Tamturbo Oy to achieve its goal of transforming the company's view of oil-free technology into a worldwide success story. Tamturbo Oy is fully committed to delivering solutions that bring the highest lifecycle value to their customers.

Predicting the performance of air turbo compressor prototypes is a critical element of the design process. As soon as such predictions are available, the design engineers can 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 can 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 depending 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 are so popular. 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. Those issues include cleaning and healing geometry to make it suitable for mesh creation and manual mesh generation focusing 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 Siemens Digital Industries Software's Simcenter FLOEFD is intended for use in the early design development stages by design engineers. There are two main principals in Simcenter FLOEFD: direct use of native CAD as the source of geometry information, and a combination of full 3D 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, Simcenter 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.

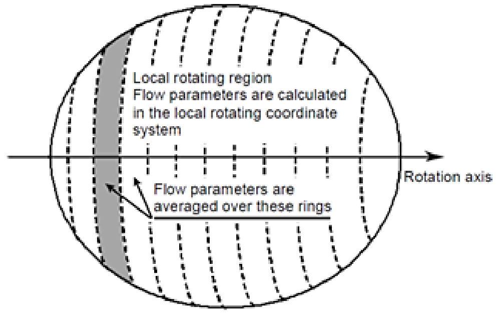

Figure 2. Ring creation in the circumferential averaging rotation approach.

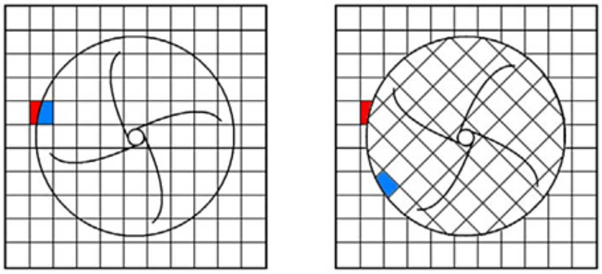

Figure 3. Sliding rotation approach.

The rotation model should be used to simulate the compressor. There are two local rotation models in Simcenter 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 (for example, 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 a "sliding interface." During the calculation, zones linked through the 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 of these techniques allow Simcenter FLOEFD to calculate air turbo compressors. Simcenter is part of the Xcelerator portfolio, a comprehensive and integrated portfolio of software and services from Siemens.

This white paper prominently shows 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 was 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 Oy 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, and shaft predetermined which mathematical models were needed for the analysis.

The turbulence model used in the analysis can detect a flow regime and automatically switched the mode between laminar and turbulent. This solver is unique and allows an engineer to obtain a solution inside narrow channels even on a coarse grid. The calculation was made

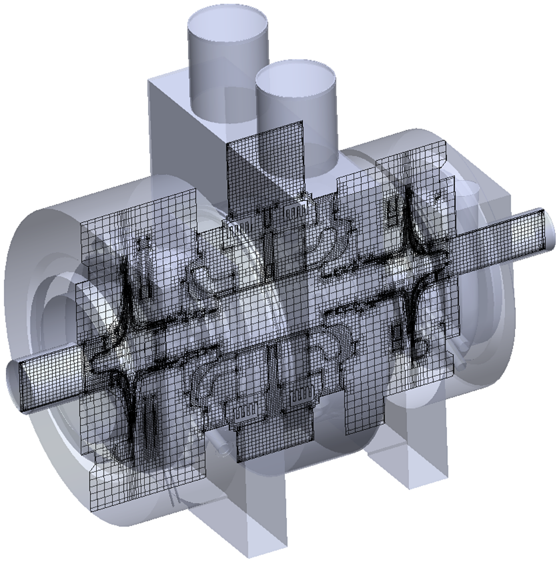

Figure 4. The mesh of the compressor model.

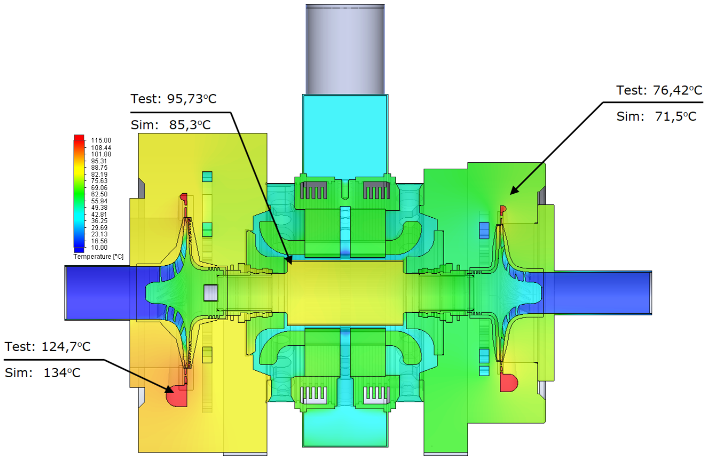

Figure 5. Temperature distribution in the longitudinal section of the compressor.

taking into account conjugated heat transfer, where heat transfer equations were solved in solid bodies. The radiation model was 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, the 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 values instead of using the Simcenter FLOEFD feature emulating a real Joule heating process.

Simulation of the conjugate heat transfer process for all zones and units of the compressor allowed for the investigation of their mutual influence and for the prediction of 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. Bearings work properly when their temperature is lower than 100 to 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 so 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 central processing units (CPU) with a frequency of 3.1 gigahertz (GHz) and only eight gigabytes (GB) of random-access memory (RAM) were used for the task. As a result, the simulation of 30 seconds of physical time took only one and a half 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 to 80 degrees in the calculation results. Tests showed the temperature rise in the nodes close to the bearings up 7 to 12 degrees.

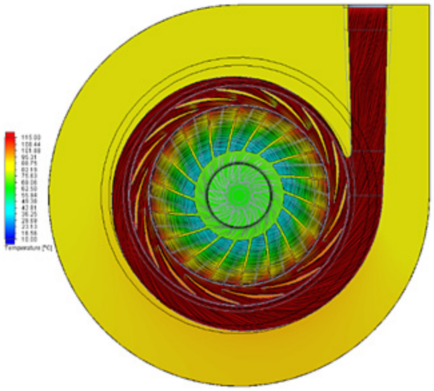

Figure 6. Stage 1. Temperature distribution in the cross section of the compressor with streamlines.

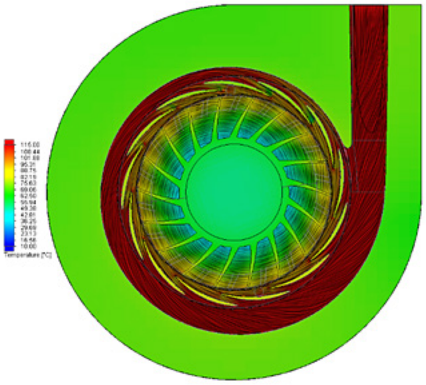

Figure 7. Stage 2. Temperature distribution in the cross section of the compressor with streamlines.

The difference between pressure values of simulation results and natural tests of the real equipment was not more than three to five percent.

Tamturbo Oy uses Simcenter 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 performed by Tamturbo Oy and a number of completed projects. Simcenter FLOEFD has several direct interfaces with other codes which are used in the company, and as a result of this 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.

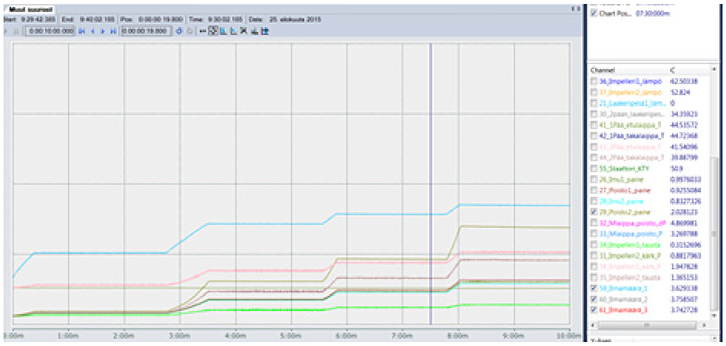

Figure 12. Dynamics of different parameters from the final compressor test.

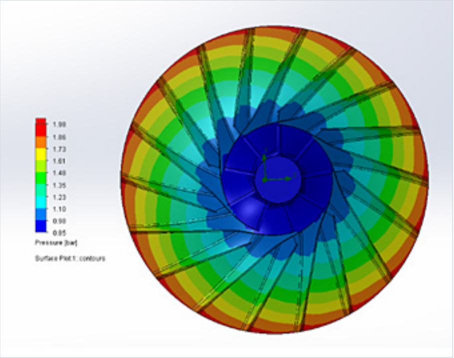

Figure 8. Stage1 impeller. Static pressure distribution on the surfaces at normalized mass flow rate 0.8, normalized rotational speed 0.943. Pressure rise ratio - from simulation 1,9718/ from test 1,9723.

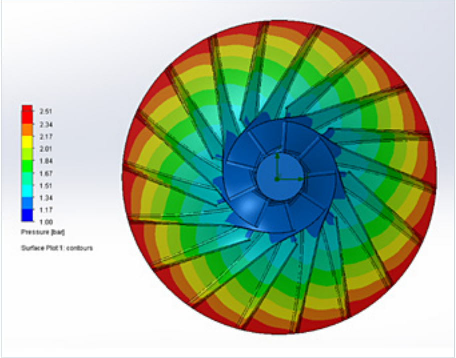

Figure 9. Stage1 impeller. Static pressure distribution on the surfaces at normalized mass flow rate 1.5, normalized rotational speed 0.943. Pressure rise ratio - from simulation 1,9585/ from test 1,9589.

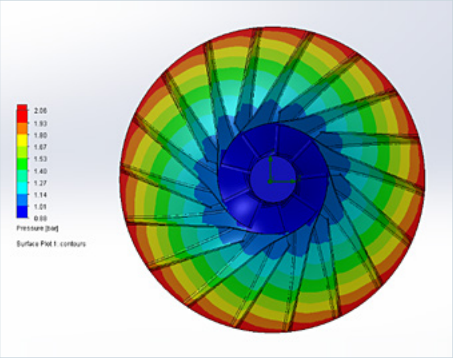

Figure 10. Stage1 impeller. Static pressure distribution on the surfaces at normalized mass flow rate 0.9, normalized rotational speed 0.97. Pressure rise ratio - from simulation 2,0391/ from test 2,0302.

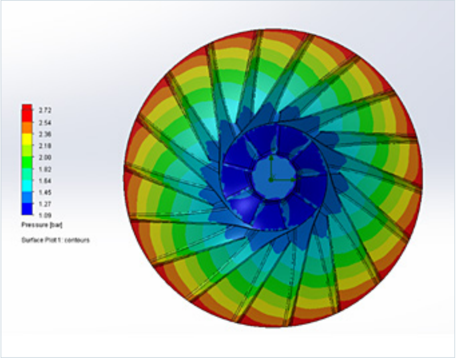

Figure 11. Stage1 impeller. Static pressure distribution on the surfaces at normalized mass flow rate 1.6, normalized rotational speed 0.97. Pressure rise ratio - from simulation 2,0083/ from test 2,0057.

Siemens Digital Industries Software

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

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

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

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

About Siemens Digital Industries Software

Siemens Digital Industries Software is driving transformation to enable a digital enterprise where engineering, manufacturing and electronics design meet tomorrow. The Xcelerator portfolio helps companies of all sizes create and leverage digital twins that provide organizations with new insights, opportunities and levels of automation to drive innovation. For more information on Siemens Digital Industries Software products and services, visit siemens.com/software or follow us on LinkedIn, Twitter, Facebook and Instagram. Siemens Digital Industries Software – Where today meets tomorrow.

siemens.com/software