Diesel-engine ATR 220 Pesa railcar with a cooling system from Voith Turbo

Fan of Simulation

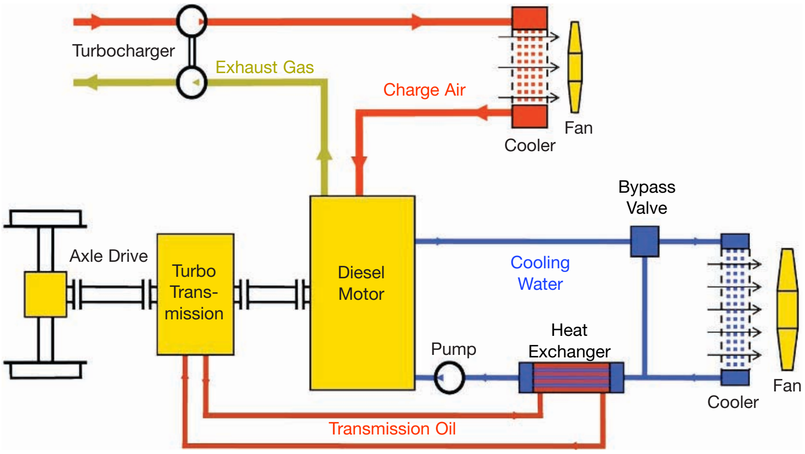
Voith Turbo reduces costs while developing quiet fans by simulating a complete railcar cooling system.

By Bernd Horlacher and Steffen Kämmerer, Development Engineers, Voith Turbo, Crailsheim, Germany

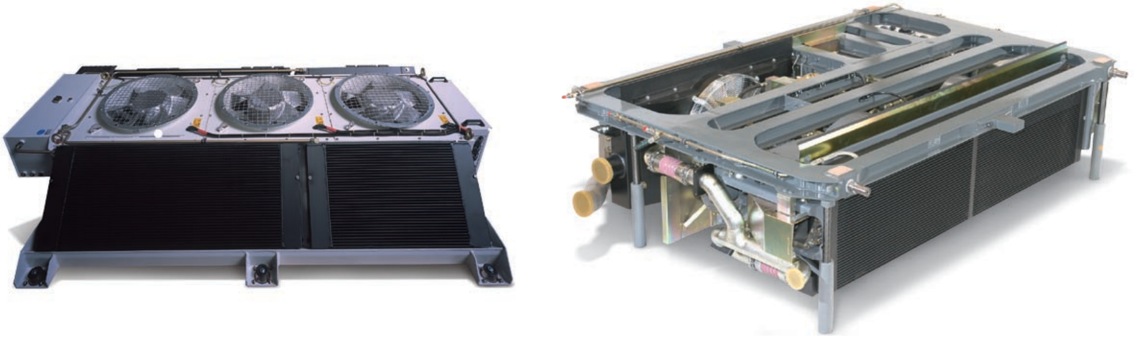
At certain operating conditions, the highest noise levels in rail vehicles come not from the engine but the cooling units, especially the fans. Increasingly stricter exhaust regulations and growing output requirements call for higher and higher cooling performances, which could lead to greater noise pollution. Voith Turbo in Germany developed a plan to address these competing parameters in the rail industry.

Voith Turbo is a leading company specializing in power transmission used in industry as well as on the road, rails and water. Equipment from Voith Turbo Cooling Systems, a division of Voith Turbo, operates safely and reliably in railcars and locomotives all over the world, including high-speed trains (diesel–hydraulic, diesel–electric and electric). These cooling systems, used to cool diesel engines, transmissions, transformers, inverters, throttles and drive motors, require an induced air mass flow that is created using high-capacity fans.

One disadvantage of classic fan design is that excessive noise is emitted at rotational speeds of 3,500 rpm and blade tip velocities up to 360 km/hr. To reduce the sound level, engineers from Voith Turbo Cooling Systems and researchers at the University of Siegen in Germany developed Voith SilentVent™ technology. Applying this technology to railcars and locomotives alike was a challenge: The cooling systems in railcars are installed on top of the roof or under the floor and, therefore, are quite compact when compared to the cooling systems for locomotives.

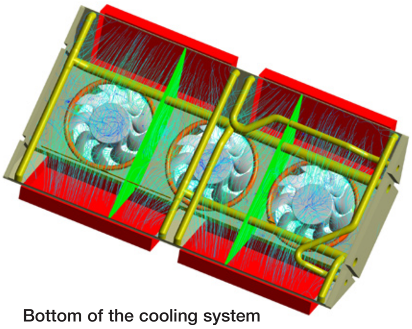


Voith flow system for cooling fans



Roof-mounted cooling system (left) and under-floor cooling system (right)

The inlet to the fan is often blocked, so the flow entering the blade passages are often highly distorted. This can reduce efficiency and increase noise generated by the fan.



Bottom of the cooling system showing obstructions

Another key issue is that there is little installation space, although high air throughput is required. The installation space is limited by the clearance of the vehicle. (The railcar's height and roof contour are limited because the train must pass through tunnels.) In addition, the flow may be blocked by other systems installed in the vehicle, and the fan housing has a specific axial installation height. From an acoustic point of view, a rail vehicle cooling system is an open system because coolers are sound permeable.

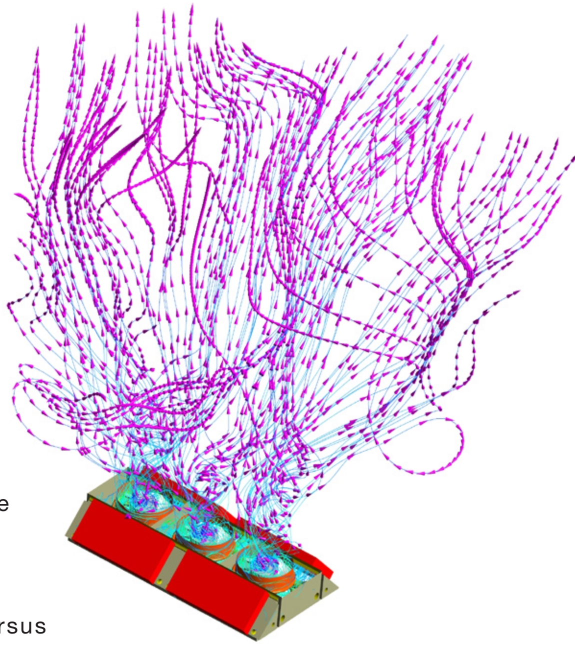
To investigate the efficiency and noise emission of the SilentVent fan technology compared to a standard fan, design engineers in the Basic Development (research and development) department used fluid dynamics in addition to their own expertise. The goal was to demonstrate the operational reliability of a complete cooling system using ANSYS CFX fluid flow software.

The cooling system consists of four heat exchangers and three axial fans in which the fan inlets are separated by metal partitions and blocked by tubes/pipes and holding plates. The scenario studied was a railcar that had come to rest in a railway station after traveling at high speed. Its fans are still rotating at full speed to dissipate the heat, but, because there is no natural wind, the warm air can be drawn back into the heat exchangers. This recirculation flow could degrade the operation of the entire cooling system. To accurately simulate these conditions, a large domain outside the cooling system and railcar was taken into account. With nearly 50 million elements, the mesh of the cooling system was necessarily very large. The grid generated needed to strike a balance between attention to detail and available resources to ensure that the simulation time was reasonable, but parts such as the fan were finely discretized to predict areas of separation, critical in obtaining accurate solutions.

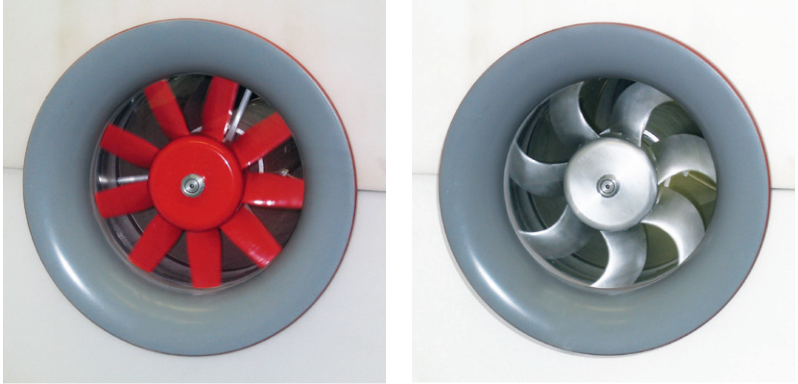
For this model, the heat exchangers were idealized and defined as a porous medium. Using the full porous model available in ANSYS CFX software, which is based on Darcy's Law, the model of the heat exchanger was calibrated by adjusting the loss coefficient to emulate the characteristic curve (loss of pressure versus volume flow).

Visualization of the flow using streamlines did not show any recirculation flow through the heat exchangers for the cooling system model with either the Voith standard fan or SilentVent technology. Each fan showed a clean outflow. By using fluid dynamics technology from ANSYS, the engineers were able to determine the volume flow rate of each fan and distribution of flow from the heat exchangers to the fans. The team established fan design data such as torque, rated input, pressure increase and efficiency for the complete cooling system — information that is not easy to obtain on a fan test under idealized conditions.

The engineers also used the ANSYS CFX Turbo Noise macro,



Exhaust air from the fan



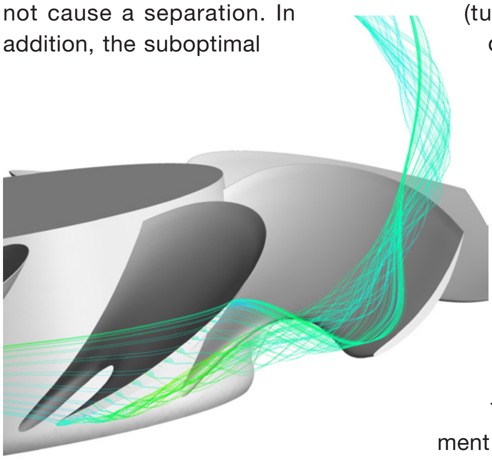
Voith standard fan (left) and Voith SilentVent fan (right)

based on the Lowson model, to assess tonal noise behavior. The Turbo Noise tool showed trends that were equivalent to experimental results, and the tool was, therefore, used to compare different designs to ensure that the sound emission of the SilentVent was much lower than the standard fan.

The distance from the SilentVent's inlet to the outlet is longer than the standard fan's, causing the inlet to be closer to the obstructing pipes and, in some cases, the ground. The direction change (deflection) of air between the heat exchangers and the inlet is also larger. As a result, the SilentVent fan blades were designed so that the inflow does not cause a separation. In addition, the suboptimal inflow due to obstructions can be improved by rearranging the pipes and their attachments. Many of these findings — obtainable only though fluid dynamics — will be further investigated in future projects.

Depending on the stage of product development, developing a single prototype fan for an existing cooling system through simulation can reduce costs by two-thirds.

Simulation tools from ANSYS are well established at Voith Turbo. Engineers at the Basic Development department depend on engineering simulation including fluid dynamics. These engineers use hydrodynamic fundamentals to design torque converters, hydrodynamic couplings (turbo couplings) and hydrodynamic brakes (retarders) as well as a variety of turbomachinery applications. The ANSYS CFX product is employed at Voith Turbo to simulate a very complex simulation portfolio — from incompressible flow to compressible flows and multiphase flows to cavitation investigations. This software is a requirement to simulate the fluid flow in rotating machinery.



Tip vortex

Turbo Post-Processing with ANSYS CFD-Post

To complement the rich collection of features for general fluid dynamics post-processing, the ANSYS CFD-Post tool provides a set of additional capabilities that specifically address the needs of rotating machinery designers and analysts. These capabilities ensure that users can maximize the benefit gained from the insight provided by their simulation results.

The turbomachinery post-processing capabilities include the ability to generate plots in the relevant two-dimensional coordinate systems. Meridional plots in axial–radial coordinates can be used, for example, to assess circumferentially averaged flow quantities. Unrolled, blade-to-blade plots are indispensable in helping to identify possible design improvements, allowing the flow to be visualized at any desired span-wise position between hub and shroud.

Specific charting options for rotating machinery also are incorporated, allowing users to easily examine blade loading at a desired position on the blade or to look at flow variations along lines from hub to shroud, inlet to outlet, or in the circumferential direction.

All these plots and graphs can be integrated into automatically generated reports based on templates provided for all types of rotating machinery, from pumps and fans to turbines and compressors. The report templates supply standard machine-specific definitions of quantities such as head rise, thermodynamic efficiency and loss coefficient, for example. ANSYS CFD-Post contains a macro for the prediction of noise from low-speed fans for quick and simple assessment of fan acoustics.

John Stokes, Requirements Manager, ANSYS, Inc.