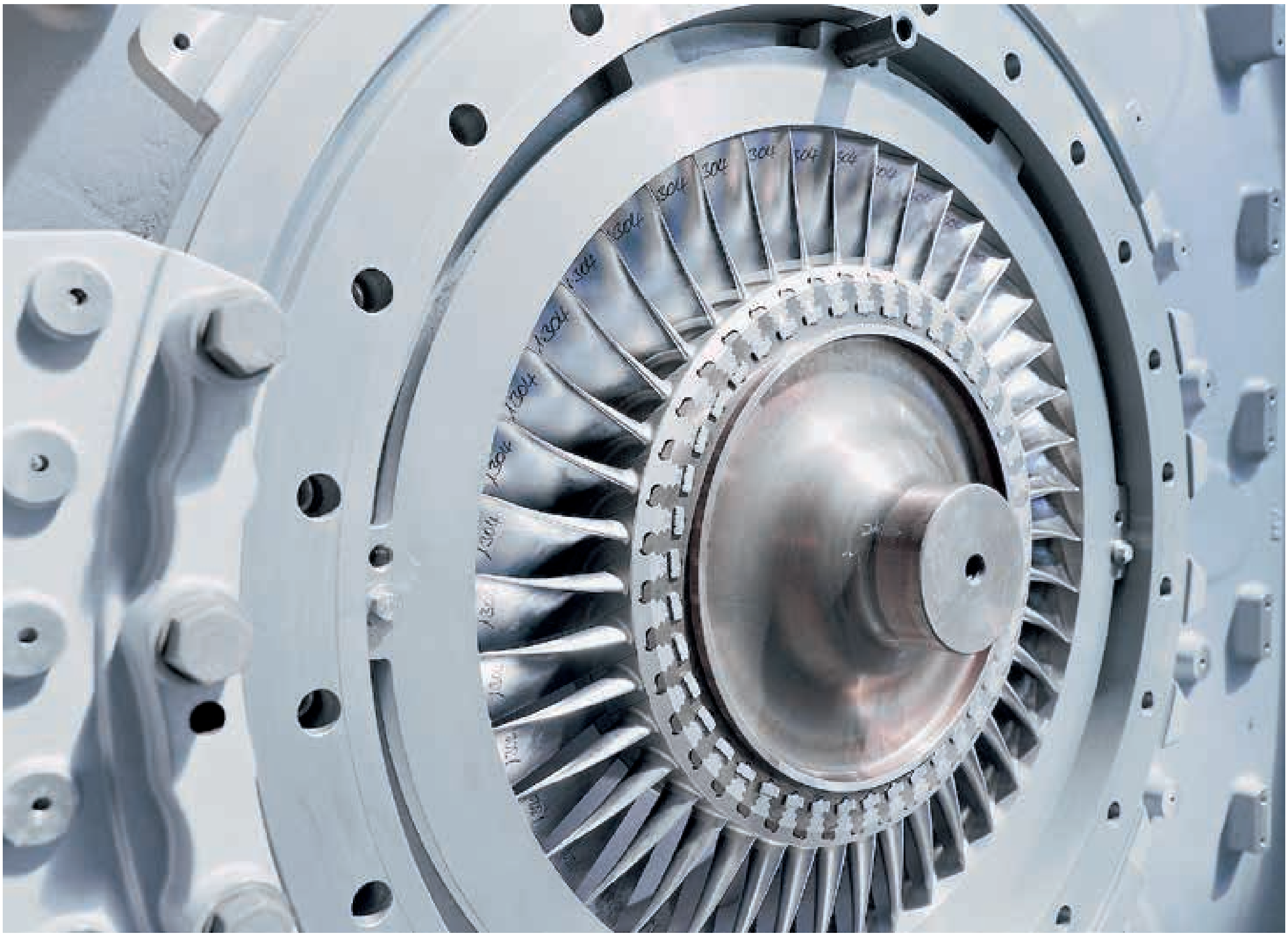


Pushing the limits

Turbine simulation for next-generation turbochargers

KWOK-KAI SO, BENT PHILLIPSEN, MAGNUS FISCHER – Computational fluid dynamics (CFD) has matured and is now an indispensable tool for turbomachinery design and development. In particular, aerodynamic analysis and optimization of axisymmetric rotors and stators based on CFD has become a standard procedure in the development process. Measurements of turbocharger turbines on a test rig have indicated that distortion of the symmetric flow at the inlet and the outlet of the turbine can have a significant influence on turbine performance. Based on CFD simulations of the complete turbocharger turbine from inlet flange to outlet flange, this influence was confirmed and a turbine geometry less sensitive to the distortions was found. These simulations have pushed the limits of the application of CFD to turbine design.

1 Computational model of the turbine stage. Left to right: intake manifold, nozzle ring, rotor and exhaust manifold.

2 Turbocharging multiplies power

ABB is at the helm of the global industry in the manufacture and maintenance of turbochargers for 500 kW to 80+ MW diesel and gas engines. ABB's technology and innovation enables customers' equipment to perform better and produce fewer emissions, even in the toughest environments. Approximately 200,000 ABB turbochargers are in operation across the globe in ships, power stations, gensets, diesel locomotives and large, off-highway vehicles. ABB has over 100 Service Stations in more than 50 countries and guarantees original parts and original service.

A typical turbocharger consists of two main systems ➔ 3: the turbine stage (orange section in figure) that recovers the power from the diesel engine exhaust gas; and the compressor stage (blue section) that boosts compressed air to the diesel engine. With a turbocharger, a diesel engine delivers four times the output power a naturally aspirated engine would. In other words, the turbocharger multiplies engine power. In addition, turbocharging technology is the main driver for the reduction of fuel costs and NO_x emission in diesel and gas engines.

A turbocharger for a high-power application typically has an axial turbine stage that includes a nozzle ring and a rotor. The turbocharger's intake manifold directs the gas flow into the axial turbine stage upstream of the nozzle ring. The gas expands in the turbine stage and drives the rotor. Downstream from the rotor, the exhaust manifold functions as a diffusor – an element critical to the performance of the turbine stage ➔ 1–2. The gas flow through the turbocharger components must be engineered to ensure maximum turbine performance and, in this regard, CFD plays a critical role in the development of turbochargers.

Challenges in turbine simulation

Conventional CFD simulation of an axial turbine stage is usually performed on two components, namely the nozzle ring and the rotor. Typically, only the flow inside one segment of the nozzle ring and the rotor is simulated – the simplification of a periodic axisymmetric flow condition is assumed. Traditionally, the simulation and design of the intake and exhaust manifolds are carried out in a separate, standalone process. The advantage of this sequential design approach is the relatively low computational demand of flow simulation for one channel of the nozzle ring and the rotor. Also, the approach works well for regular axial inflow and outflow conditions. However, the simulation of the manifolds alone, without the turbine stage, is not a complete simulation as the interaction between the turbine stage and the manifolds is not taken into account.

> The flow through the components must be engineered to ensure maximum turbine performance and, in this regard, CFD plays a critical role.

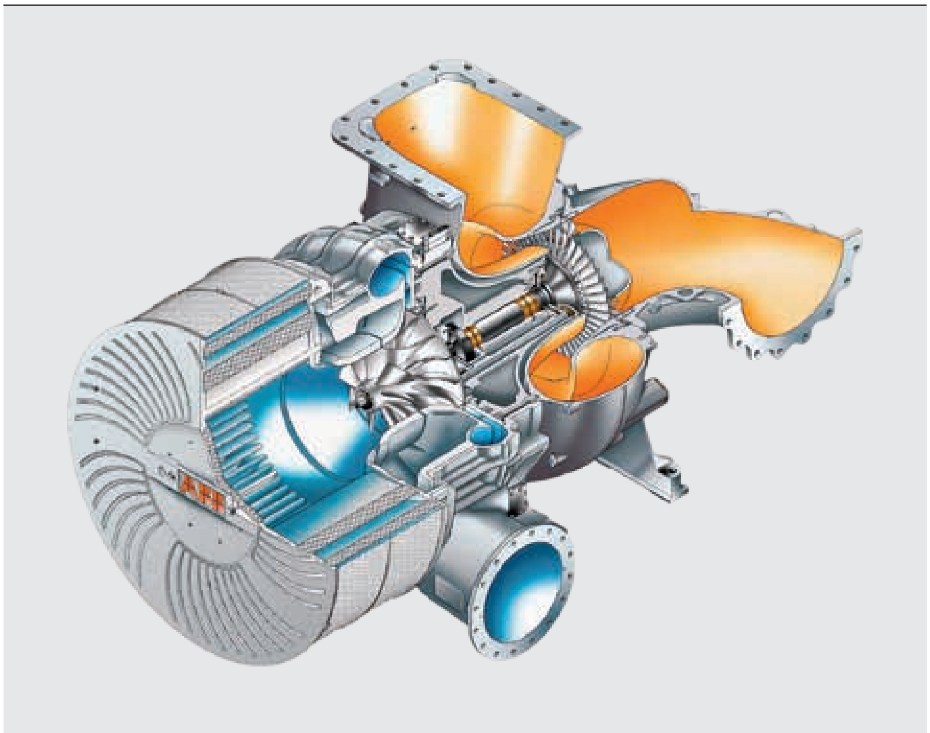
The challenge in turbine-stage simulation arises when the intake manifold and the exhaust manifold exhibit severe turns and radial bends, due to spatial requirements. In this case, the assumption that the inflow and outflow paths are axially symmetric no longer holds. These flow deviations can have a significant effect on the diffusor and turbine performance as undesired flow recirculation and separation can occur in the diffusor region and result in a drop of turbine efficiency. Up until now, finding a suitable diffusor involved several design iterations with extensive test runs.

In order to investigate the flow of the axial turbine stage with various intake or exhaust manifolds without having to test these design variants on the turbine test rig, multicomponent CFD simulations can be performed. In this regard, simulations of the radial intake manifold together with the nozzle ring were conducted to study inflow inhomogeneity originating

Title picture
How can the full capabilities of CFD be exploited to make the design of turbines in turbochargers even better?

The challenge associated with a multicomponent CFD simulation is the higher computational demand due to the full 360° computational domain.

3 ABB turbocharger A175-L



from the radial intake manifold – in particular, the flow performance in the diffusor within the exhaust manifold under nonaxisymmetric inflow and outflow conditions. The challenge associated with a multicomponent CFD simulation is the higher computational demand due to the full 360° computational domain that is now required because the axisymmetric flow assumption no longer holds. As is the case generally in multicomponent CFD simulations, suitable mesh matching, numerical models appropriate for multicomponent situations and reliable definition of boundary conditions are all essential.

Despite these challenges, multicomponent CFD simulations do offer the opportunity to gain insight into the flow across different components. In particular, the coupled effect at the turbine rotor exit on flow conditions in the diffusor can be studied in detail. Furthermore, because the CFD simulation of the full turbine stage from inlet flange to outlet flange corresponds to the configuration in the standard test bench measurement, numerical CFD results can be validated against experimental measurements.

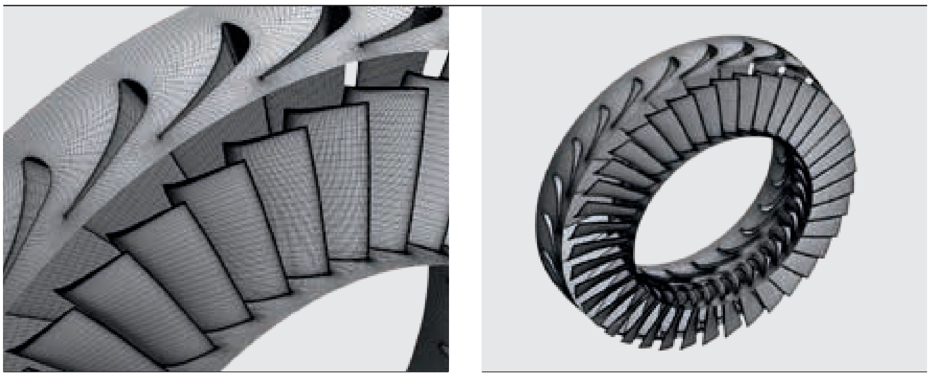
For flange-to-flange simulations however, a balance must be struck between computational effort and solution accuracy when investigating the influence of nonaxial inflow and outflow conditions.

The challenge in turbine-stage simulation arises when the intake manifold and the exhaust manifold exhibit severe turns and radial bends.

High-fidelity CFD to the rescue

Given the objective of flange-to-flange simulations, the CFD meshes, needed for the numerical calculation and the models, must be defined accordingly. A specific meshing tool that can cater for the vane and blade geometry in the turbine nozzle ring and the rotor is employed there → 4, while a general-purpose meshing tool is adopted for the intake and exhaust manifolds, where its flexibility allows it to efficiently tackle the various curved profiles.

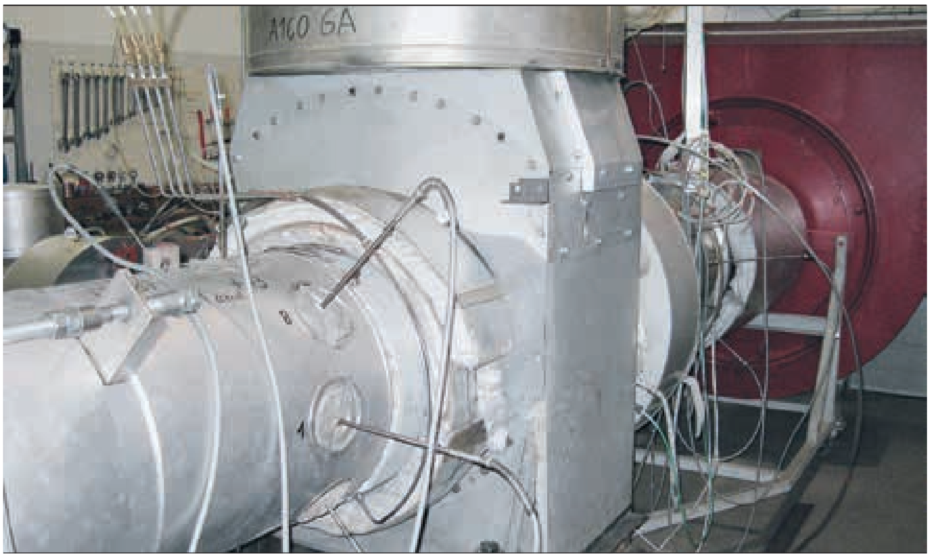
4 CFD mesh adaptions



4a Nozzle vane

4b Rotor blade

5 The turbine stage on a test bench



As the CFD simulation of the full turbine stage corresponds to the configuration in the standard test bench measurement, results can be validated against experimental measurements.

In the steady-state CFD simulations, the Reynolds-averaged Navier-Stokes equations serve as the flow-governing equations. The k-ω shear stress transport model, in conjunction with a wall function approach, is adopted to predict the turbulence, onset and amount of flow separation. The CFD simulation is set up so as to avoid resolving small-timescale transient effects.

The stationary nozzle ring and rotating turbine wheel of the computational domain are coupled by a so-called frozen-rotor domain interface model. This interface model is an efficient algorithm for computing a steady-state CFD solution for stationary and rotating parts.

The CFD model and setup above is based on a widely-adopted industrial CFD platform, the accuracy of which is already verified. The computational domain typically consists of around 20 million cells and nodes. The large-scale simulations are carried out on an ABB high-performance computing (HPC) cluster so that the simulations of various components in different operating and test conditions can be conducted within a few weeks.

Physical validation

Setting up and running the simulation model is only half the story: The simulation results must be validated against physical observations, typically test measurements, to ensure that they accurately describe the physical phenomena.

In the test facility ➔ 5, an extensive list of turbine-stage characteristic quantities, including static and total states of pressure and temperature at various locations of the turbine, mass flow rate, turbine work, turbine power and turbine stage efficiency, can be measured. This provides an excellent setting to accurately define the boundary conditions in the CFD model and to form the basis for comparison and validation of the CFD simulation results.

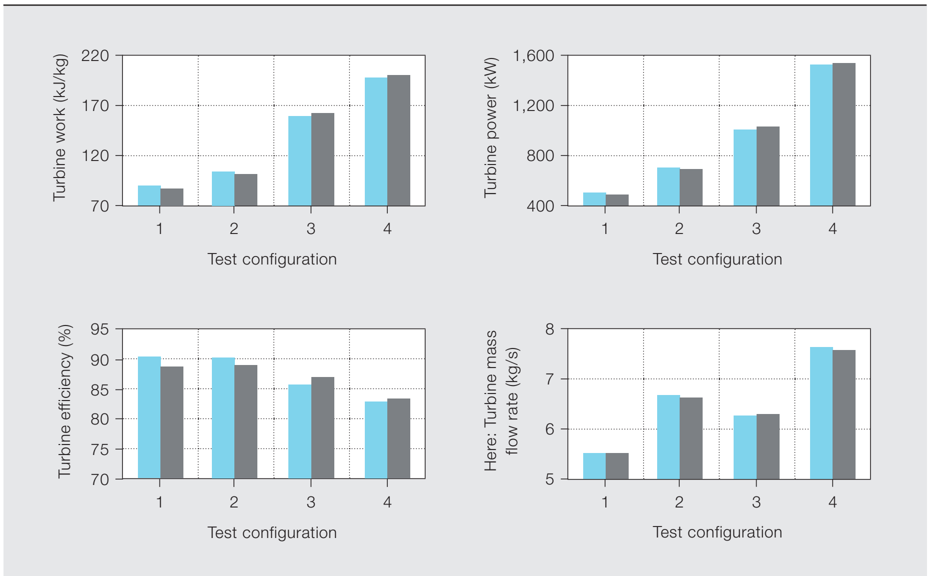
Results delivered by CFD

Different scenarios were simulated by the CFD model. These scenarios involved different designs and combinations of the nozzle ring, turbine rotor, intake manifold and exhaust manifold, as well as different operating conditions and cold-gas and hot-gas test conditions.

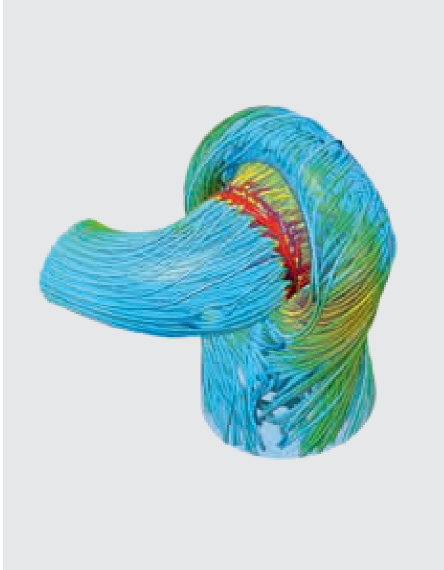
In general, there was very good agreement between the test measurements and the CFD simulations of the turbine-stage characteristic quantities ➔ 6. Because the CFD simulations are verified and validated against test measurements for a number of configurations, good insight into the flow conditions in the turbine stage from the inlet flange to the outlet flange can be gained ➔ 7.

As an example, a radial intake manifold and a 90° turn in the exhaust opening in the exhaust manifold can be simulated. With this configuration, there is a high potential for undesirable flow separation and recirculation in the diffusor region.

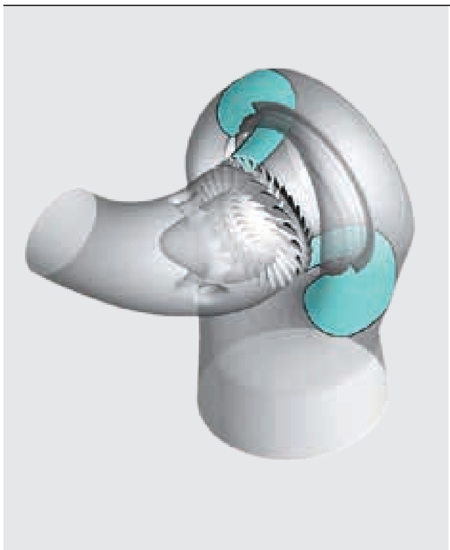
6 Comparison of selected characteristic quantities from test measurements (blue) and CFD simulation (grey)



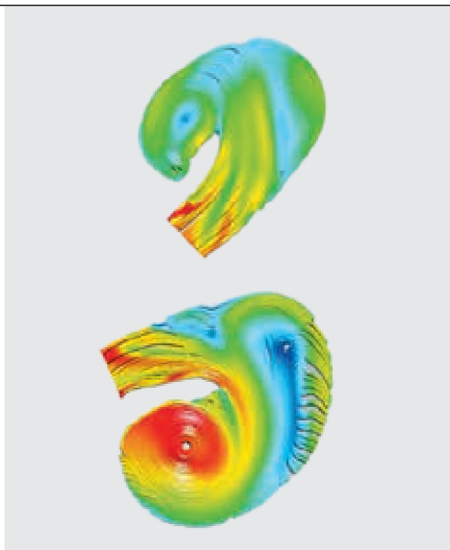
7 Streamlines from inlet flange to outlet flange. The streamline color indicates flow velocity.



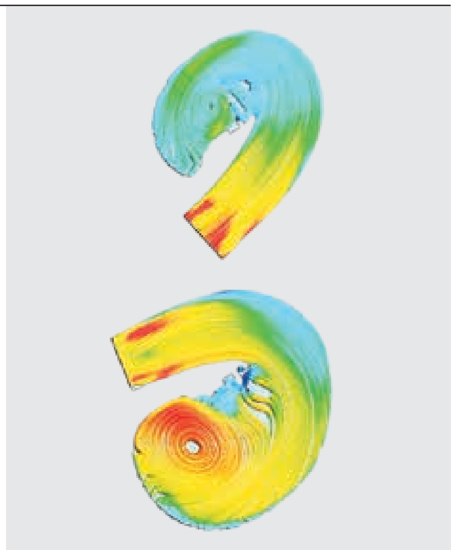
8 Streamlines for different exhaust manifold designs



8a Planes of interest



8b Undesired flow separation and recirculation in diffusor



8c Desired expansion in diffusor

As was revealed in test measurements, the performance of the turbine stage deteriorates under certain operating conditions due to nonsymmetrical inflow conditions. The drop in turbine efficiency can be captured and reproduced by CFD simulations, and the flow conditions inside the diffusor can be further analyzed by studying the CFD simulations ➔ 8. As an illustration, the plane susceptible to flow separation and recirculation inside the diffusor is highlighted in ➔ 8a. Exhaust manifolds with different diffusor designs can be simulated to identify those that avoid flow separation inside the diffusor region ➔ 8b–8c. In this way, turbine performance can be maintained even under the restrictions imposed by the radial intake and exhaust manifolds, and the adverse inflow and outflow conditions.

Outlook beyond limit

At ABB, CFD is being used to advance understanding in new areas of turbocharging technology – for example, heat transfer simulation and thermal analysis for cooling design and material selection, as well as acoustic simulation for noise reduction. In parallel, the ABB development facility – CFD simulation tools, test center and HPC cluster – are continuously being improved. Through these efforts, the next generation of turbochargers will be created.

Kwok-Kai So
Bent Phillipsen
Magnus Fischer
ABB Process Automation, Turbocharging
Baden, Switzerland
kwok-kai.so@ch.abb.com
bent.phillipsen@ch.abb.com
magnus.fischer@ch.abb.com