Steaming Ahead with Turbomachinery Simulation

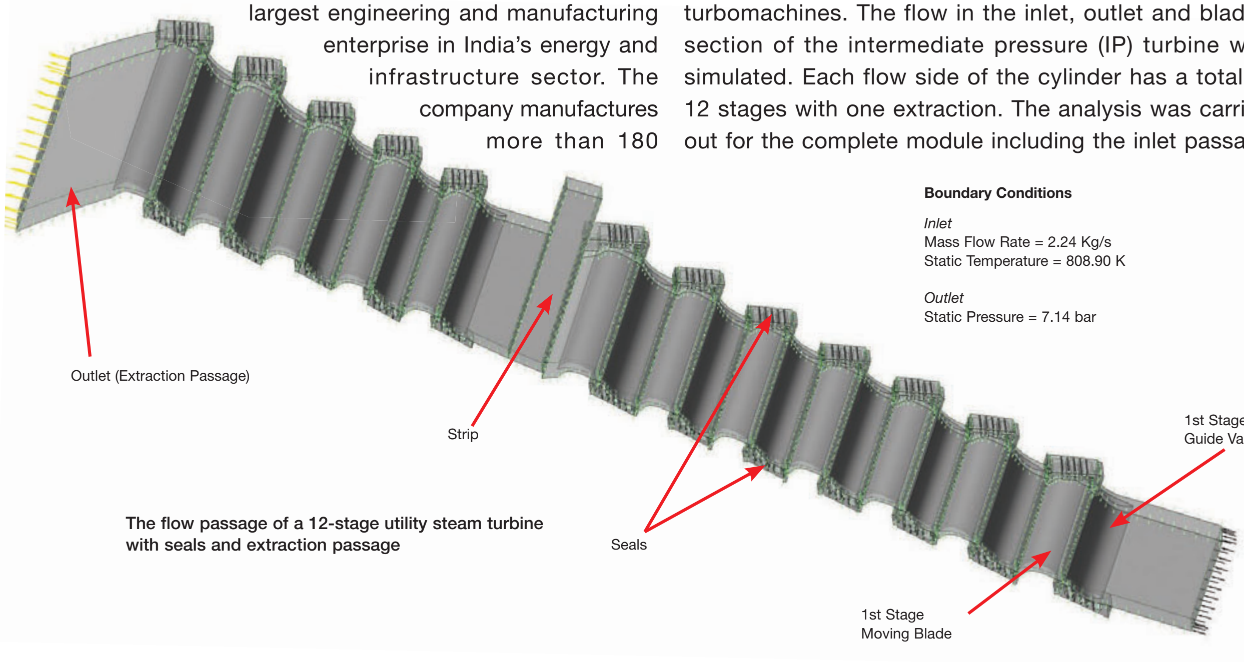
Engineers gain valuable insight into design of a complex steam turbine by employing simulation.

By G.V. Subbarao, Senior Deputy General Manager
and Prabhat Kumar Hensh, Engineer, Corporate Research and Development,
Bharat Heavy Electricals Limited, Hyderabad, India

Turbomachinery design is highly complex and demanding. Small improvements in the performance of these complicated machines can translate to greater electrical output and ultimately increased profit for the operating utility. However, understanding what is happening within a turbine as it operates is extremely difficult. Experiment and testing can show improved performance but often cannot detect exactly why the improvement has occurred. Furthermore, testing requires building a prototype, which is usually cost prohibitive. Often, engineering simulation provides the insight needed to pinpoint areas for improvement, especially when performed well before the machine is built — ultimately resulting in far more output.

Bharat Heavy Electricals Limited (BHEL) is the largest engineering and manufacturing enterprise in India's energy and infrastructure sector. The company manufactures more than 180 products under 30 major product groups. BHEL develops products to a high level of quality and reliability. This is the result of its emphasis on design, engineering and manufacturing to international standards. To accomplish this, the organization acquires and adapts some of the best technologies from leading companies in the world and develops technology in its own research and development centers.

Software from ANSYS has helped engineers at BHEL to gain important insight into the performance of one of its utility system steam turbines. The research and development group chose ANSYS CFX software to analyze the flow path for the turbine because this fluid dynamics tool is well known for its ability to analyze turbomachines. The flow in the inlet, outlet and bladed section of the intermediate pressure (IP) turbine was simulated. Each flow side of the cylinder has a total of 12 stages with one extraction. The analysis was carried out for the complete module including the inlet passage



Boundary Conditions

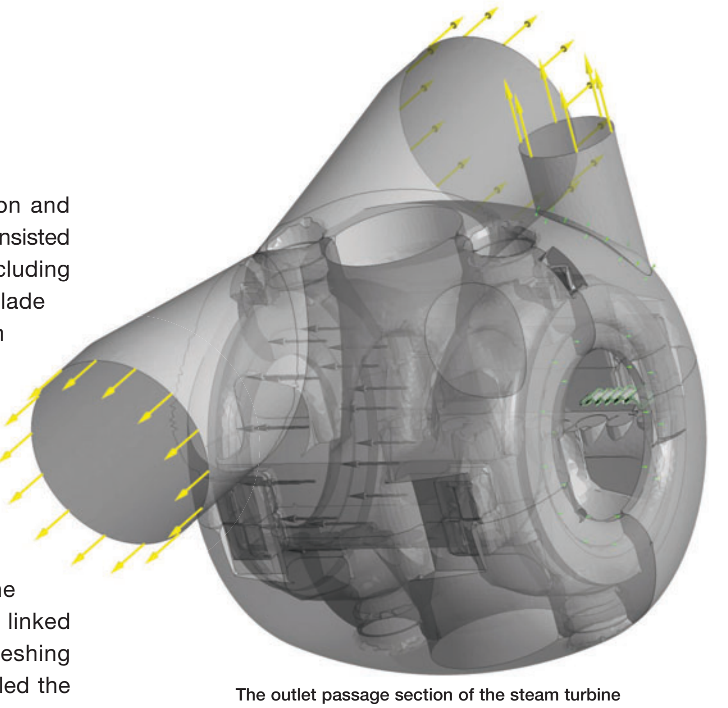
Inlet
Mass Flow Rate = 2.24 Kg/s
Static Temperature = 808.90 K

Outlet
Static Pressure = 7.14 bar

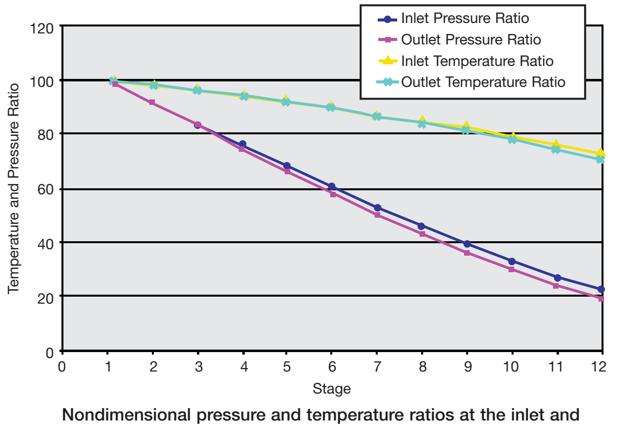
The flow passage of a 12-stage utility steam turbine with seals and extraction passage

section, bladed path with seals, steam extraction and outlet passage section of the turbine. The model consisted of a single blade passage for each blade row including the seal and interface regions for a total of 24 blade rows and 50 components. The engineering team generated blade row meshes using the ANSYS TurboGrid tool and employed ANSYS ICEM CFD software to generate the mesh for the seals, inlet and outlet passage sections.

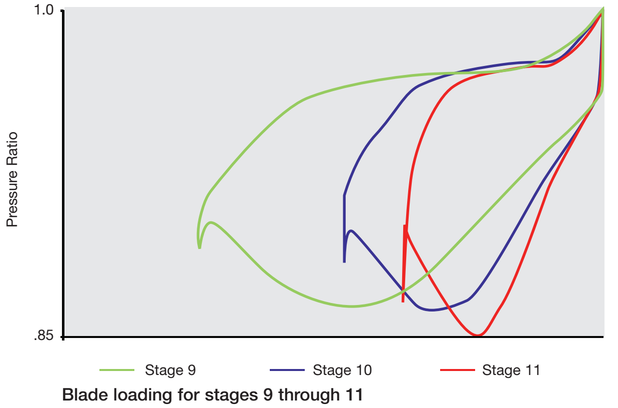
Engineers modeled and meshed the blade and seal regions separately. The hub, shroud and seal regions were attached to the blade region by a general grid interface (GGI), and the software's multiple frame of reference capability linked adjacent components together. In addition to meshing the blade flow path, the team meshed and modeled the inlet and outlet passages of the turbine. The mass flow rate and static temperature at inlet and static pressure at outlet were used as boundary conditions for the simulation. The total simulation of the complete flow passage had approximately 6 million nodes consisting of hexa and tetra elements with many GGI and stage interfaces to connect the 50 components to form a large and complex model from many simple parts. Due to the large size of the model, a high-performance compute cluster was required to run the simulation.



The outlet passage section of the steam turbine



Nondimensional pressure and temperature ratios at the inlet and outlet of each of the 12 stages



Blade loading for stages 9 through 11

BHEL's team computed the steam parameters such as pressure, temperature, enthalpy, power per stage, stage isentropic efficiency and leakage flow for each stage using CFD-Post post-processing capabilities. Using fluid dynamics analysis results, the engineers calculated the total pressure loss coefficient for inlet and outlet passage sections.

Fluid dynamics predictions for the stage-wise thermal parameters are in broad agreement with previously performed two-dimensional program results. The flow in the inlet and outlet passage sections of the IP turbine is smooth, and loss coefficients are within the acceptable limits. Minor recirculation zones are evident in the outlet section; consequently, the loss coefficient for the outlet is more than that for the inlet section.

The analysis carried out using ANSYS CFX fluid flow software for the complete module of the utility turbine has helped BHEL to acquire insight into the flow parameters for each stage and into loss coefficients for the inlet and outlet passage sections.

BHEL plans to use software from ANSYS to analyze the design of similar turbines to gain a better understanding of the flow mechanisms and, thereby, improve performance. ■