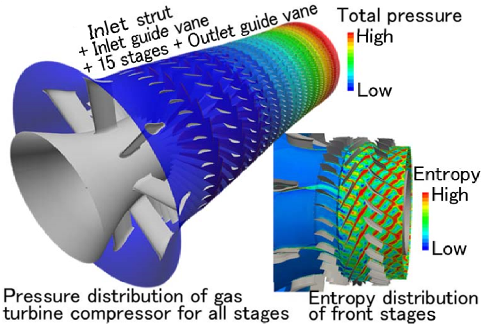
Computational Fluid Dynamics Technology Applied to High Performance, Reliable Axial Compressors for Power Generation Gas Turbines



RYOSUKE MITO[*1] XUELIN GAO[*2]

THOMAS WALKER[*2] YASURO SAKAMOTO[*3]

HIROYUKI HAMANA[*4]

When developing high-performance, reliable gas turbine compressors, repeated cycles of experimentation, verification, and analysis are used to eventually determine the optimum design. Currently, thanks to advances in computational fluid dynamics (CFD) technology, full-stage analysis results can be obtained within a practical amount of time. This allows for the loading distribution and internal flow phenomena for all stages to be computed accurately and understood in a quantitative fashion during the design phase. This is not only for the rated load condition, but also for the partial load condition meaning that using these results as part of the design improvement process can reduce the required experimental verification cases. Given that the role of CFD analysis is expanding, Mitsubishi Heavy Industries, Ltd. (MHI) has made efforts to improve the analysis accuracy. This paper discusses recent improvements in CFD methods and illustrates some applications for the improved technology.

1. Introduction

In recent years, power generation gas turbines are beginning to be required to extend operability through a larger part load operation range as well as improve efficiency at the rated load condition. Therefore, high-performance compressor development now requires a more precise, quantitative understanding of the flow conditions. Specifically, some stages under partial load conditions have a flow field that is nearly stalled. This implies that the prediction of the compressor's operating range requires evaluating the entire compressor using three-dimensional CFD. So far, however, due to time restrictions, three-dimensional CFD has only been applied to some of the operating range or some of the compressor stages and thus it could not be used to accurately predict all of the relevant compressor flow behavior.

Given this, instead of using a conventional CPU (central processing unit) for CFD, through open innovation MHI has introduced a new ultra-high-speed parallel computing technology which utilizes a GPU (graphics processing unit) and a new analysis code designed to take advantage of GPU technology. This advancement in simulation technology was applied to our turbomachinery well ahead of any other organization in the world. With this technology, coupled with the continued development of GPUs, CFD calculation time was reduced to one twentieth of that of existing CPUs. Furthermore, the prediction accuracy was also improved via modifying some of the flow models used in the CFD code. These modifications where verified by showing that the resulting flow field was close to the actual internal flow phenomena obtained from detailed experimental data. As a result, CFD is now being applied to many more aspects of turbomachinery design and a gas turbine multistage compressor can be evaluated in detail from its inlet through to its outlet.

*1 Chief Staff Manager, Takasago Research & Development Center, Technology & Innovation Headquarters

*2 Takasago Research & Development Center, Technology & Innovation Headquarters

*3 Manager Takasago Research & Development Center, Technology & Innovation Headquarters

*4 Mitsubishi Hitachi Power Systems, Ltd.

2. Advanced CFD Technology for Turbomachinery

2.1 Ultra-high-speed parallel computing technology using GPUs

For a given set of commands, GPUs have the ability to process a much larger amount of data in parallel compared to the conventional CPU. The new parallel computing technology works by combining the command (and logic) functionality of the CPU with the large amount data processing capacity of the GPU (**Figure 1**). Furthermore, MHI has followed the introduction of this new technology with improvements to the CFD analysis code to be used with the GPUs. This, combined with the continuing development of GPU technology, has led to reduced calculation time (about one twentieth of conventional CPU computation). This allows the application of full-stage CFD analysis at a practical stage in the compressor design process. In addition, it is now possible to predict the stall point of a multistage compressor within a practical amount of time through unsteady CFD analysis.

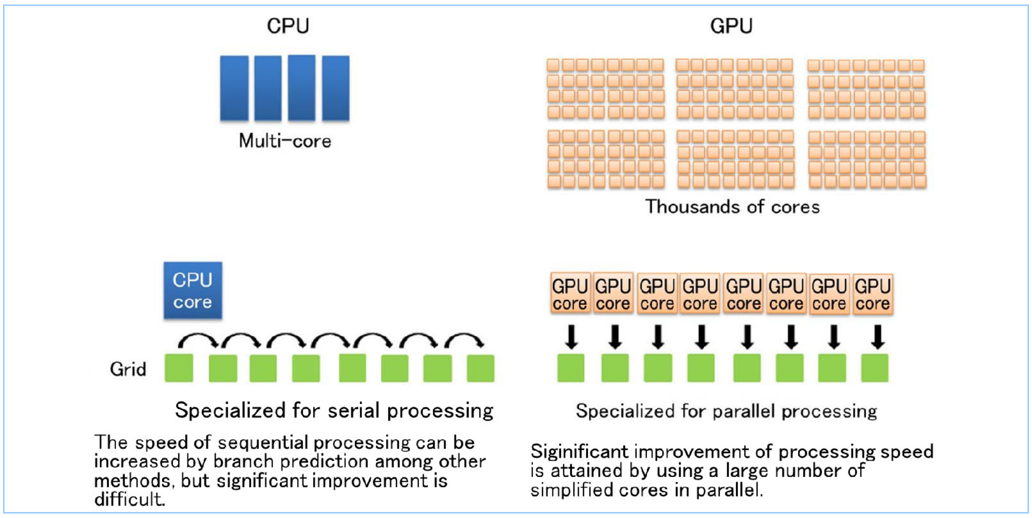


Figure 1 GPU parallel computing technology (20 times faster than conventional CPU computing technology)

2.2 Detailed simulation of the compressor's real geometry

Due to the reduction of calculation time it is now possible to run CFD on a large calculation mesh. This allows for geometries that tended to be simplified in the past, such as cavities and clearances, to be simulated in detail (**Figure 2**).

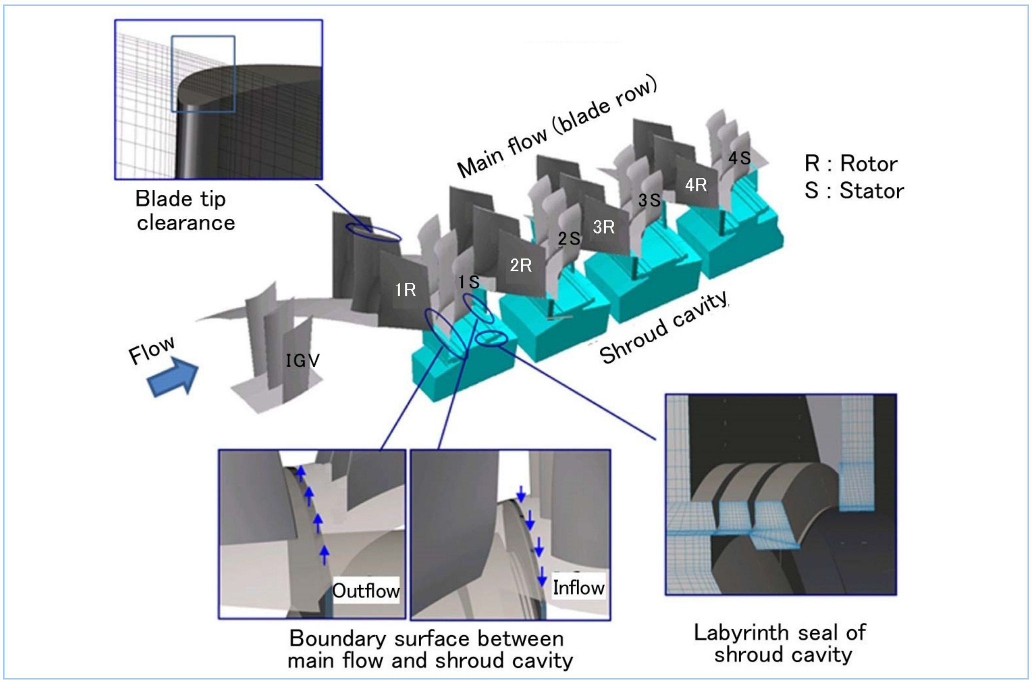


Figure 2 Compressor CFD model(main flow + clearance)

2.3 Acquisition of detailed measurement data and improvement of CFD methods

The internal flow conditions in a power generation gas turbine compressor change as the flow moves from the front stages to the rear stages. In the front stages, the flow is transonic, meaning that prediction of shockwaves generated by the rotor is important when computing compressor performance characteristics. In the rear stages, the flow is subsonic, meaning that the secondary flow near the endwall and the rotor tip clearance flow are the most important flow features. In addition, the pressure ratio of each stage increases or decreases compared to the design point depending on the power output requirement. Since the compressor is operated at a constant speed, the power requirement changes based on air temperature, season, and time of day. The performance characteristic as well as the internal flow of the compressor is interdependent on each stage and thus, accurate data acquisition for each stage is necessary for the prediction of the full-stage performance. For this reason, MHI uses several test rigs to reproduce the front stage as well as the middle/rear stage flow in order to acquire performance characteristic and internal flow data. This is not only under rated load conditions, but also at high pressure conditions, low speed conditions, and conditions with enlarged rotor tip clearances. The improved CFD method (shown in **Figure 3**) was validated against this data.

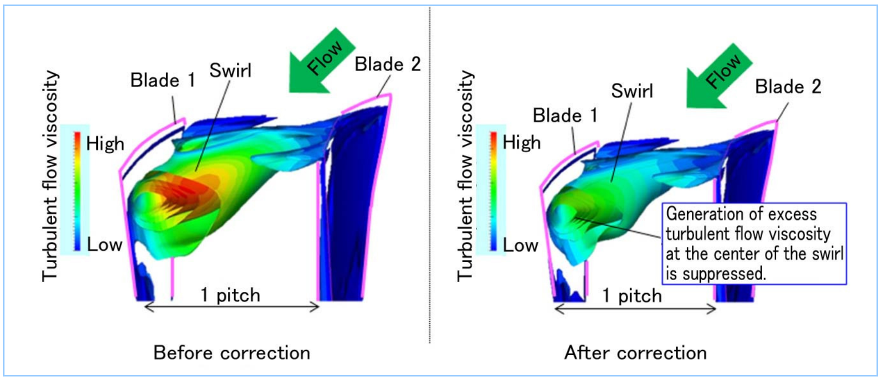


Figure 3 Example of improvements to the turbulence model

3. Verification and improvement of accuracy through comparison with measured data

3.1 Comparison with rig test data

Figure 4 compares the compressor characteristic obtained with a subsonic four-stage compressor test rig and the CFD result. **Figure 5** compares the total pressure loss coefficient distribution from a total pressure measurement probe inserted into the flow path with the CFD result. The improvement of the CFD method increased the prediction accuracy of the characteristic line as well as the total pressure loss coefficient distribution in the span-wise direction.

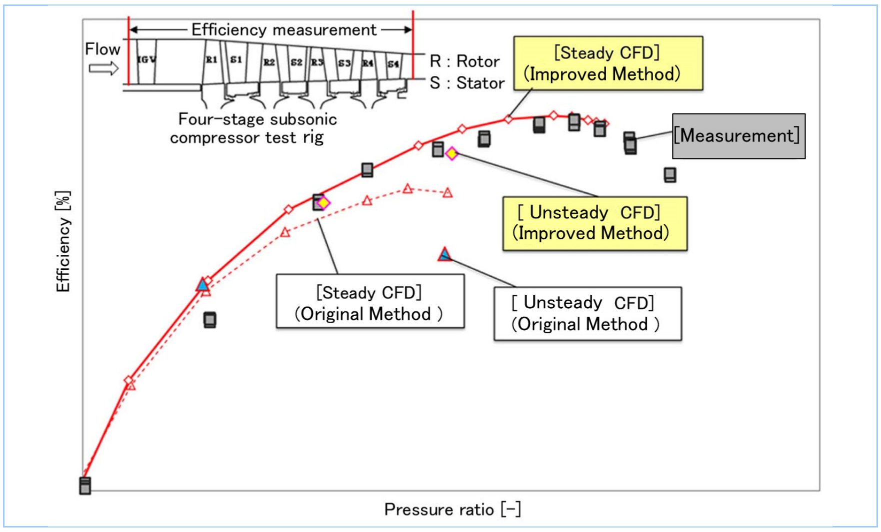


Figure 4 Pressure ratio vs efficiency for a four-stage subsonic compressor (Measurement vs. CFD)

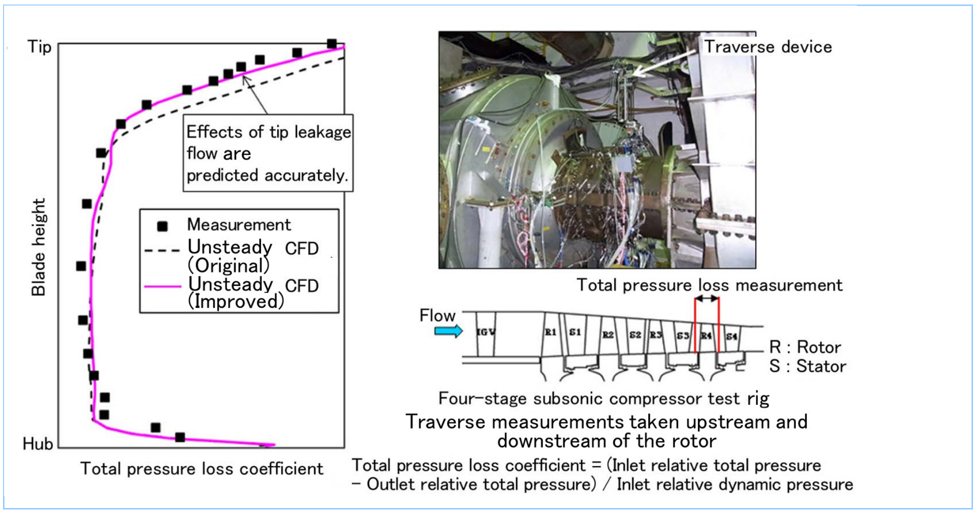


Figure 5 Total pressure loss coefficient for the 4th rotor (measurement vs. CFD)

3.2 Comparison with gas turbine verification facility (T-point) measurement data

Figure 6 shows a picture of the gas turbine combined cycle power plant verification facility (T-point), which is a part of Mitsubishi Hitachi Power Systems, Ltd.'s Takasago Works. T-point is a combined cycle power plant verification facility consisting of a gas turbine, a steam turbine, and a heat recovery steam generator (HRSG). It is used to acquire various component data through special measurements taken during trial operation of prototype machines.

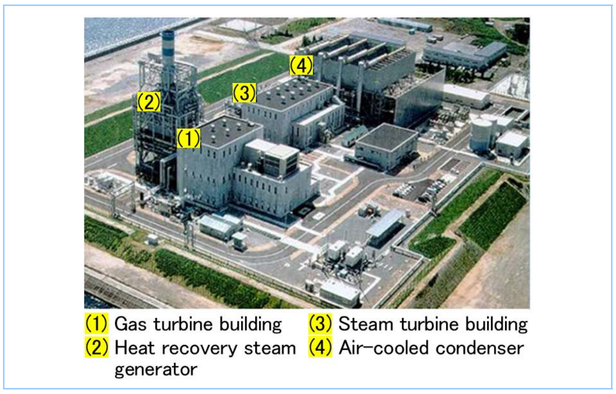


Figure 6 Combined cycle power plant verification facility in Takasago (T-Point)

Figures 7 and **8** show the full-stage CFD results for the actual compressor. Comparing these to the T-point data, the CFD prediction accuracy for the casing surface pressure distribution and the total pressure distribution in the flow path can be validated.

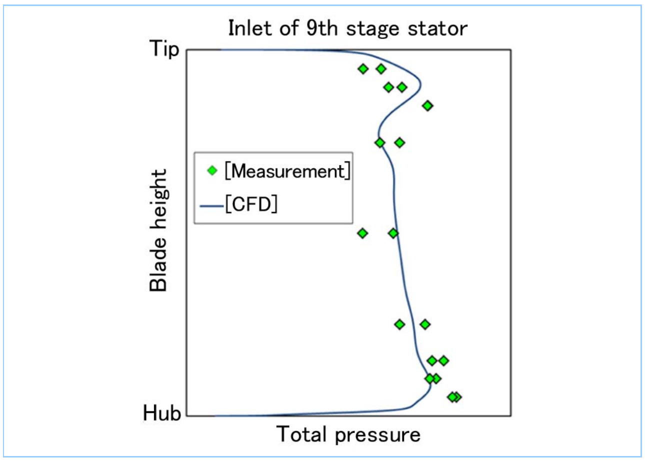


Figure 7 Total pressure distribution for the middle stages of the actual compressor (measurement vs. CFD)

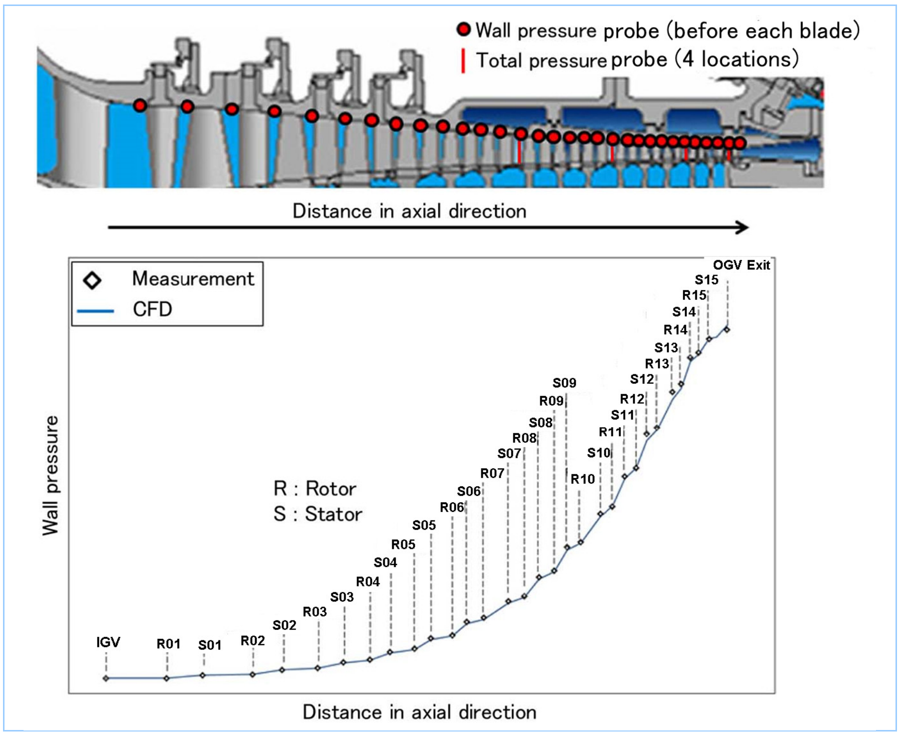


Figure 8 Casing pressure distribution for the full-stage compressor before each blade and OGV exit (measurement vs. CFD)

4. Application to power generation gas turbine compressor design

4.1 Compressor aerodynamic performance improvements through detailed understanding of internal flow phenomena

By making use of CFD results, for which the prediction accuracy has been validated against measurement data acquired through rig testing as well as verification testing on the actual machine, MHI is working on improving the understanding of internal flow phenomena as well as improving aerodynamic performance. For example, when the rotor tip clearance is relatively large, such as in the middle and rear stages, the loss increases near the rotor tip because of the effect of the leakage jet generated by the flow passing through the tip clearance as shown in **Figure 9**. Therefore, in order to increase performance it is important to control the leakage flow from the tip. MHI is trying to reduce the amount of time required to improve the aerodynamic performance in the actual compressor by quickly evaluating proposed blade geometry improvements using rig testing and selecting the best designs before moving to verification tests on the actual machine.

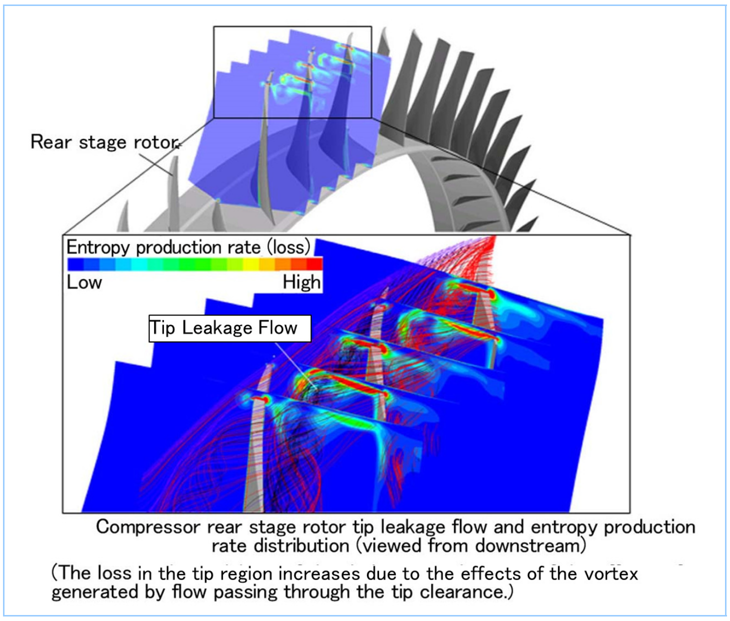


Figure 9 Detailed flow in the rotor tip for the rear stages

4.2 Prediction of compressor pressure fluctuation during gas turbine startup

Figure 10 shows an example of unsteady CFD of a compressor during gas turbine startup. When the gas turbine is started, the flow rate is low which in turn causes rotating stall to occur in the front stages. This is a very unstable flow field; however, advancements in CFD technology now allows for phenomena such as this to be predicted. The predicted value of the pressure fluctuation agrees favorably with the measured value at the casing surface. In this analysis, there were four stall cells, which also agreed with the actual measurement results.

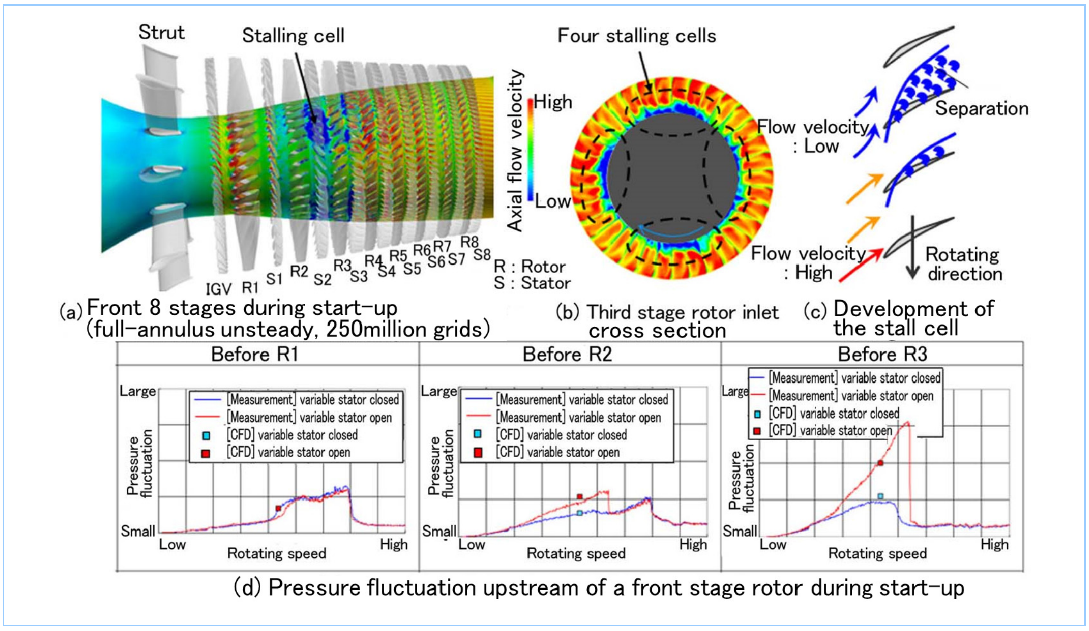


Figure 10 Prediction of pressure fluctuation during start-up of compressor

4.3 Evaluation of the effect of compressor bleed on the main flow

Compressors have bleed pipes, a bleed chamber, and a bleed slot to ensure stability during startup and to supply the turbine with cooling air. Since a number of pipes, say four, are placed at discreet circumferential locations, a non-uniform flow develops around the annulus. The effect this non-uniformity has on the stalling behavior has been evaluated. **Figure 11** shows the calculation model and CFD results for changes in (1) the number of the bleed pipes, (2) the size of the bleed chamber, and (3) the bleed slot geometry.

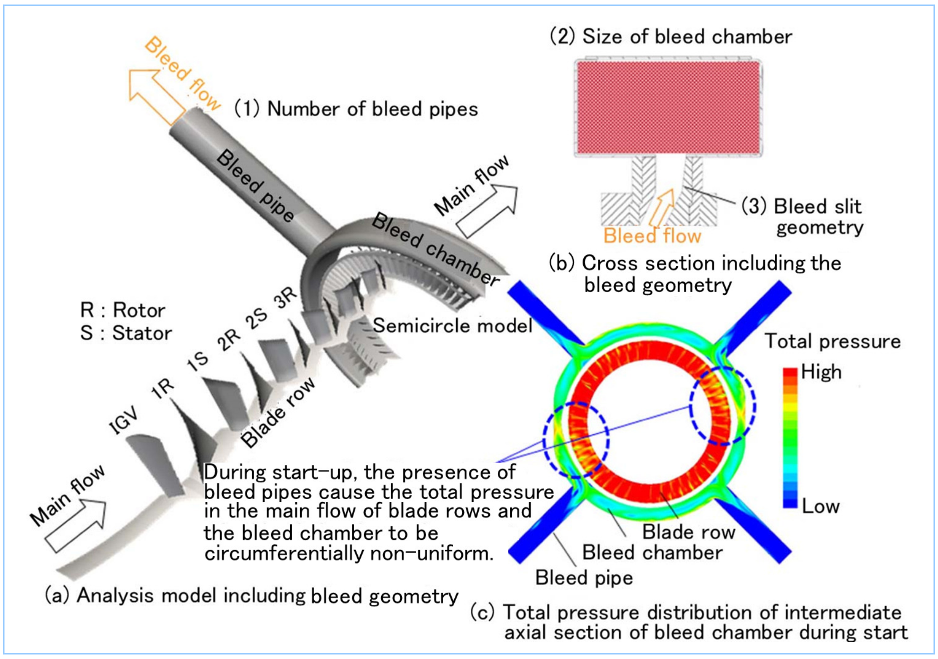


Figure 11 Evaluation of the effect of bleed geometry on the main flow

4.4 Evaluation of the effect of the inlet duct on the compressor front stages

A single suction duct is adopted for the intake of the compressor. The effect of the curved section of the duct on the upstream flow and the effect on the mounting angle of the inlet guide vane cause non-uniform flow to form in the circumferential direction. The effect of the non-uniform flow on compressor aerodynamic performance was evaluated (**Figure 12**).

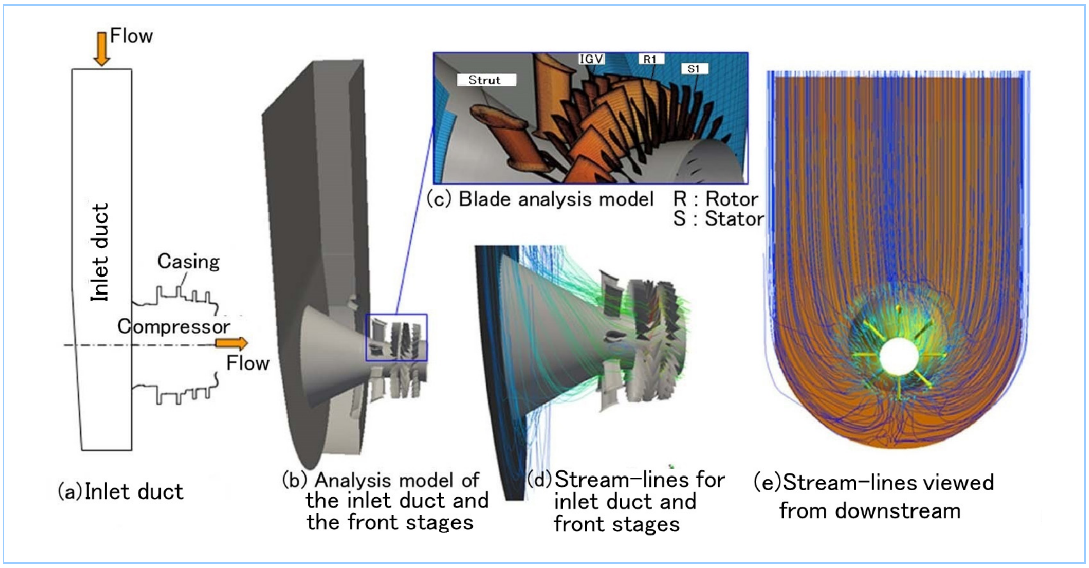


Figure 12 Evaluation of the effect of the inlet duct shape on the compressor front stages

5. Conclusion

This paper presented improvements in our CFD methods and some applications of our new ultra-high-speed parallel computing GPU technology to detailed calculation models designed to help improve the performance of power generation gas turbine compressors. The role of CFD analysis is expanding as computation technology advances and it is expected that high fidelity analysis will be applied in even more aspects of compressor design in the future. However, to facilitate this movement toward CFD analysis, reliable and detailed experimental data is necessary to verify the accuracy and understand the phenomena seen in the CFD results. Although not mentioned in this paper, MHI has also worked on developing high precision measuring technology and advanced experimental equipment in order to accurately acquire the necessary data. With a precise, detailed understanding of the physical phenomena through a combination of these technologies, as well as efforts put into shortening the new proposal, experimental verification, and product development cycle, MHI endeavors to deliver superior products to the market faster.

Reference

1. Hada, S. et al., Test Results of the World's First1,600C J-series Gas Turbine, Mitsubishi Heavy Industries Technical Review Vol. 49 No. 1 (2012) pp.19～24
2. Tobias Brandvik and Graham Pullan, An Accelerated 3D Navier-Stokes Solver for Flows in Turbomachines, ASME paper, GT2009-60052
3. Ito, E., CFD and Optimization of Gas Turbine Components including Inlet and Exhaust Ducts, Journal of the Gas Turbine Society of Japan Vol. 40 No. 6 (2012. 11)