Platform Technology for Computational Fluid Dynamics Supporting Design of System Products

—from Power Plants and Industrial Machinery to Home Appliances

Shigehisa Funabashi
Taku Iwase, Dr. Eng., P.E.Jp (Mechanical)
Kiyotaka Hiradate
Masashi Fukaya, Dr. Eng.

OVERVIEW: Pumps, fans, and other fluid devices are used in a wide range of system products, from power plants and industrial machinery to home appliances. Although CFD has been used in the design of these devices for some time, growing demands to reduce the load on the environment, to improve reliability and other aspects of product performance, and to cut costs are driving the need for analyses that cover a broad scope and achieve a high level of accuracy. On its own and through collaborations between industry and academia, Hitachi is responding to this need by working on large-scale analyses using fluid dynamics to study phenomena that could not previously be tackled. Hitachi is also applying techniques such as multi-objective design optimization in situations where trade-offs exist between different objectives, and special cavitation analysis techniques.

INTRODUCTION

THE pursuit of greater energy efficiency is an important consideration in the development of many different products, from home appliances to power plants and industrial machinery. Competition on energy efficiency in the field of home appliances is becoming progressively more intense with each passing year, and development is ongoing with attention given to even the smallest of losses. As small differences in efficiency have a significant impact on power plants and industrial machinery, Hitachi recognizes the importance of developing products with superior efficiency to those of its competitors. Pumps, fans, and other fluid devices are used in many of these systems, and computational fluid dynamics (CFD) has for some time been successfully used as a core technology for their design.

However, as CFD comes to be used in routine design work, the development of distinctive products requires that further work be undertaken in areas such as techniques for applying CFD in design and in the study of phenomena that are not amenable to analysis by conventional techniques.

This article describes examples of the use by Hitachi of core CFD technologies, including the use of CFD by high performance computing for the visualization of noise sources and the study of unstable head curves in pumps, the use of multi-objective design optimization techniques for control of unstable phenomena and high efficiency in blowers, and the use of cavitation analysis to predict where erosion will occur on pump impellers and to determine how to improve residual stresses in the internal structure of nuclear reactors.

TECHNIQUES FOR CFD BY HIGH PERFORMANCE COMPUTING

The prediction of phenomena such as fan noise or the unstable head curves of pumps at low flow rates requires that detailed analyses be performed that treat the flow as an unsteady phenomenon. To predict these phenomena, Hitachi is working with The University of Tokyo on the development of technology for CFD by high performance computing. The following section describes analyses performed using free software called “FrontFlow/blue (FFB).” FFB is primarily

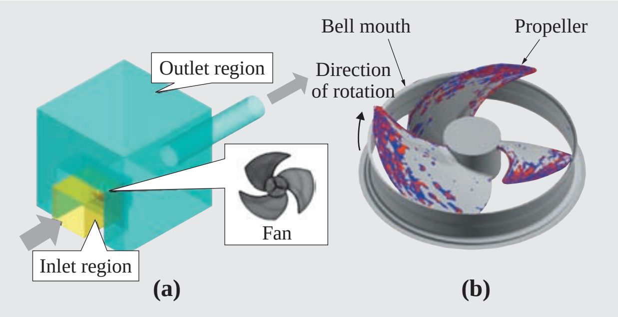


Fig. 1—Visualization of Fan Noise Source.
Diagram (a) shows the computational model, and diagram (b) shows a Powell noise source map. The computation was performed for a total of 16 million mesh elements, of which 2.8 million corresponded to the blade-to-blade region.

developed by The University of Tokyo, and is based on incompressible large eddy simulation (LES). FFB is therefore suitable for high performance computing.

Visualization of Fan Noise Sources

Reducing the aerodynamic noise produced by the fans used in products such as chillers and air conditioners, home appliances, and information technology (IT) equipment is an important consideration when seeking to improve comfort in the home, workplace, and other environments. The development of low-noise fans requires techniques for visualizing noise sources.

Fan noise is caused by phenomena such as flow separation and lift fluctuation due to vortices. Accordingly, it is necessary to capture the fluctuation of tip vortex and flow separation on the blade surface for the visualization of a noise source. The large-scale unsteady fluid analyses required to evaluate these phenomena can be performed by using FFB and super computer.

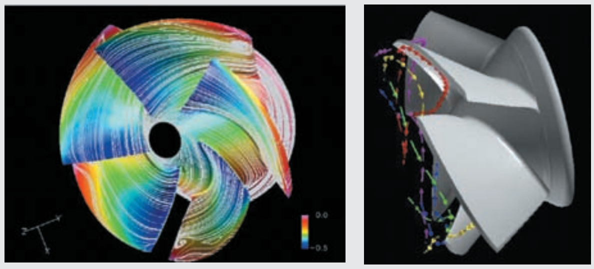
Fan noise source visualization[1] based on analysis results was performed to investigate location where aerodynamic noise generates (see Fig. 1). The Powell noise source referred to in the figure is an index that represents the noise source due to deformation of vortices [div($\omega \times v$), where ω is vorticity and v is velocity][2]. The figure shows how the noise sources are concentrated at the tips and around the leading edges, indicating that the tips and leading edges are influencing the noise. In this way, use of analysis to identify noise sources helps find ways to reduce fan noise.

Study of Unstable Head Curves in Pumps

When operating in their partial capacity range, pumps sometimes become unstable, resulting in pulsations in their flow rate and pressure. For this reason, it is necessary to develop pump design techniques (operating range enlargement) that will avoid unstable head curves and produce pumps capable of stable operation over a wide range of flow rates.

Causes of an unstable head curve include flow separation from the blade. Accordingly, the prediction of unstable head curves requires the execution of large-scale unsteady fluid analyses by using LES, a technique capable of predicting the behavior of unsteady and complex flows.

Working with The University of Tokyo, Hitachi undertook a study of the causes of unstable head curves using CFD by high performance computing with 78 million mesh elements[3] (see Fig. 2). The figure shows how localized pressure increases occurred along the trailing edges of the blades, and that these pressure increases caused flow separation to occur. The results demonstrated that the influence of this flow separation extended to the adjacent blade and that these phenomena were the cause of the unstable head curve. Hitachi is studying complex flow phenomena through detailed analyses such as this to devise new design method.



(a) (b)

Fig. 2—Results of Analyzing Pump Operating in Partial Capacity Range.
The figures show the distribution of pressure on the blade surface (a) and the stream lines on the suction side of the blades (b). Computational fluid dynamics (CFD) by high performance computing can be used to perform detailed analyses.

MULTI-OBJECTIVE DESIGN OPTIMIZATION TECHNIQUES

Multi-objective optimization is a technique for evaluating the trade-offs between several product characteristics (objective functions). The multi-objective genetic algorithm (MOGA) is one of the multi-objective optimization techniques, and Hitachi has developed an optimization method that combines MOGA with CFD and applied it to the development of products such as vacuum cleaner fans[4]. The following is an example that achieved higher efficiency and wider operating range in centrifugal blowers for sewage aeration using this method.

Definition of Multi-objective Optimization Problem for Centrifugal Blower Impeller

Centrifugal blowers are widely used in products that require liquid or gas to have relatively high pressure and a low flow-rate. Centrifugal impellers are one of the components of centrifugal blowers and require higher design-point efficiency and, like the pump described above, wider operating range. In general, design-point efficiency and operating range are in a trade-off relationship. Therefore, adjustment of this trade-off is important in impeller design.

Performing multi-objective optimization using CFD results at the design and some low-flow-rate points is a simple method for evaluating the impeller operating range. However, this approach is very time-consuming. As an alternative, Hitachi developed a technique for evaluating the impeller operating range only from the CFD result at the design point.

Senoo et al.[5] reported that an increase of the relative velocity deceleration ratio from the inlet to the throat of an impeller causes an increase in the boundary layer thickness near the throat and flow separation. Here, the throat of the impeller is defined as the section where the sectional area of the blade-to-blade passage becomes narrowest. In our preliminary investigation, the flow separation near the throat occurred in several impellers analyzed by CFD. Moreover, in these impellers, the larger the throat deceleration ratio ($W_1/W_{th,s}$) at the design point, the higher the flow-rate at which the flow separation occurred [see Fig. 3 (a)]. In other words, a reduction of $W_1/W_{th,s}$ at the design point decreases the flow-rate at which flow separation and impeller stall occurs and improves the operating range.

Accordingly, an impeller shape optimization using the adiabatic efficiency (η_{imp}) and $W_1/W_{th,s}$ derived from CFD at the design point as the objective functions was performed[6].

Optimization Results

The distribution of solutions obtained from the optimization result shows the trade-off relationship between these two objective functions clearly [see Fig. 3 (b)]. The solution that showed the best balance between η_{imp} and $W_1/W_{th,s}$ was selected as the optimized shape. The results of a detailed analysis indicated that the optimized shape achieved 1.8% higher efficiency and 5% wider operating range than the conventional one [see Fig. 3 (c)].

In this way, it was confirmed that selecting appropriate evaluation indices (objective functions) could expand the scope of application of multi-objective optimization. Hitachi considers that the applications of design optimization can be further expanded by finding new evaluation indices using CFD by high-performance computing as described earlier in this article.

CAVITATION ANALYSIS TECHNIQUE

Cavitation is a phenomenon of bubble generation in low-pressure regions in a liquid. When the bubbles collapse, a high level of impact energy is released to the surrounding area under certain conditions. While this impact energy applied to the material can cause erosion, it can also be utilized for improving residual stress. Hitachi, therefore, has developed an analysis technique that considers the pressure interaction between flow and bubbles at a scale factor of 10^6. This analysis technique can be used both to predict the behavior of bubbles that expand and contract on the order of a microsecond and to estimate the impact energy[7]. The sections below describe the application of this cavitation analysis technique to a large industrial pump and to a preventive maintenance technology for nuclear power plants.

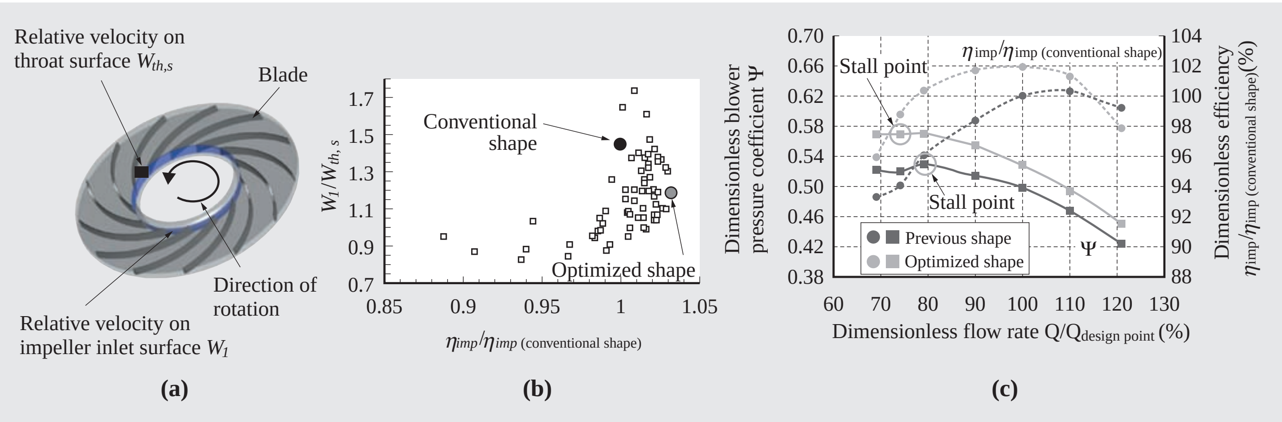


Fig. 3—Multi-objective Optimization of Centrifugal Blower Design.
Diagram (a) shows the impeller. The operating range of the centrifugal impeller can be estimated from the throat deceleration ratio at the design point. Graph (b) shows the results of design optimization. The vertical axis represents the throat deceleration ratio and the horizontal axis represents the dimensionless efficiency, such that points at the bottom right corner of the graph indicate superior performance. The detailed analysis results in graph (c) show how the selected optimum shape has improved efficiency and shifted the point at which stalling occurs to a lower flow rate, indicating a wider operating range.

Prediction of Erosion Location on Pump Impeller

Pumps are a form of fluid machinery that is used extensively in social infrastructure such as industrial plants, water treatment and sewage plants, and vehicles. Ensuring the reliability of pumps is an important mission. If it becomes possible to predict whether and where cavitation erosion occurs in a pump, it is possible to take measures to prevent it at the design stage.

In the past, the detection of the erosion location in a large pump was done by making a small model pump for experimental tests. After the impeller was painted and the pump was operated for a certain period of time, the location of erosion could be identified by noting where paint peeling occurred, which indicates where the impact energy was high. By using the new analysis technique, Hitachi achieved a world-first prediction of the expansion and contraction of large numbers of bubbles in a rotational flow field, and the distribution of bubble nuclei. The predicted bubble behavior was then used to estimate the distribution of impact energy(7). This allowed the design to be completed in a short period of time taking account of reliability as well as performance (see Fig. 4).

Prediction of Residual Stress Improvement on Structure Surface in Nuclear Reactor

Tensile residual stress occurs on weld surfaces of the internal structure of nuclear reactors at nuclear power plants. The tensile residual stress can be a cause of stress corrosion cracking when combined with sensitization of materials and a corrosive environment.

Water jet peening (WJP) is a preventive maintenance technology for mitigating stress corrosion cracking. It reduces the risk of stress corrosion cracking by applying a cavitating jet from an underwater nozzle to welds so that the impact energy converts the residual stress from tensile to compressive.

In the past, the nozzle locations were selected by operating WJP on mockups of the internal structure and measuring the distribution of the residual stress after WJP. However, by quantifying the strong correlation between the impact energy and the residual stress, the developed analysis is able to estimate the residual stress from the impact energy(8). As a result, it is now possible to investigate the operating conditions of WJP in detail at low cost (see Fig. 5).

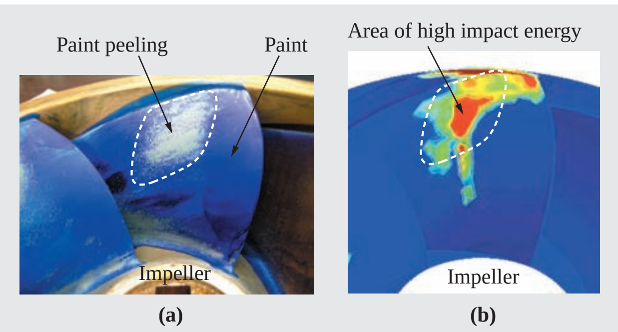


Fig. 4—Location of Cavitation Erosion on Pump Impeller.
Diagram (a) shows the result of a paint test on a small model pump, and diagram (b) shows the impact energy distribution determined by the analysis. The results show a good agreement between the area where the cavitation caused the paint peeling and the area of high impact energy.

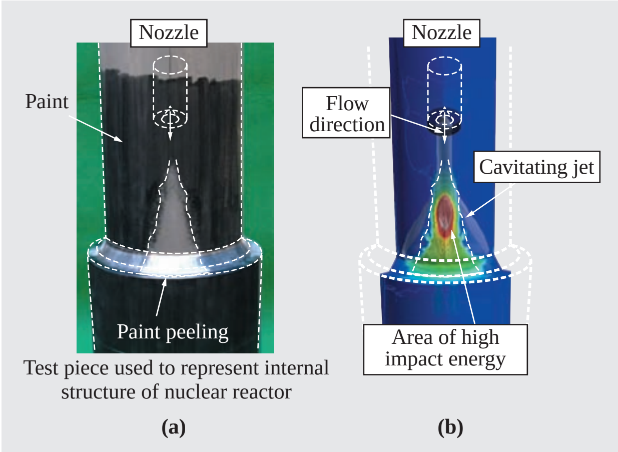


Fig. 5—Residual Stress Improvement on Internal Structure Surface in Nuclear Reactor.
Diagram (a) shows the results of a paint test using a mockup of the internal structure in a nuclear reactor. High compressive residual stress was measured in the region where the paint was peeled due to cavitation. Diagram (b), meanwhile, shows the impact energy distribution determined by the analysis. The results show a good agreement between the area where the paint was peeled and the area of high impact energy.

CONCLUSIONS

This article has described examples of the use by Hitachi of core CFD technologies for product design.

While CFD already has a long history of use in product design, the development of new analysis techniques and other improvements such as in computer performance mean that routine design work can now take advantage of more advanced types of analysis. These practical applications in turn bring about new challenges and requirements. In response to these challenges, and to take advantage of technical innovations in product development, Hitachi believes that continuing to seek innovations in core CFD technologies is important, and that this represents a foundational technology for Hitachi.

REFERENCES

(1) N. Shikazono et al., “Latest Developments of Highly Effective Refrigeration, Air-Conditioning and Hot Water Supply Technologies,” CMC Publishing (Nov. 2011) in Japanese.

(2) C. Kato et al., “Large Eddy Simulation of Internal Flow of a Propeller Fan and Visualization of Its Aeroacoustics Source,” Journal of Visualization **23**, No. 91, pp. 192–198 (Oct. 2003) in Japanese.

(3) Y. Yamade et al., “Large Eddy Simulation of Internal Flow of a Mixed-Flow Pump,” Proceedings of the ASME 2009 Fluids Engineering Division Summer Meeting, FEDSM2009–78416 (Aug. 2009).

(4) K. Sugimura et al., “Multi-Objective Robust Design Optimization and Knowledge Mining of a Centrifugal Fan That Takes Dimensional Uncertainty Into Account,” ASME Paper No. GT2008–51301 (Jun. 2008).

(5) Y. Senoo et al., “Experimental Study on Flow in a Supersonic Centrifugal Impeller: 1st Report, Characteristics of Impeller and Its Operating Range,” Transactions of the Japan Society of Mechanical Engineers B **45**, No. 390, pp. 156–163 (Mar. 1979) in Japanese.

(6) K. Hiradate et al., “Improvement in Efficiency and Stall Margin of Centrifugal Blower Impeller by Multi-Objective Optimization,” ASME Paper No. GT2011–45351 (Jun. 2011).

(7) M. Fukaya et al., “Prediction of Cavitation Intensity and Erosion Area in Centrifugal Pump by Using Cavitating Flow Simulation with Bubble Flow Model (Fluids Engineering),” Transactions of the Japan Society of Mechanical Engineers B **74**, No. 746, pp. 2126–2123 (Oct. 2008) in Japanese.

(8) M. Fukaya et al., “Mitigation of Stress Corrosion Cracking Based on Residual Stress Improvement by Water Jet Peening (WJP),” Proc. of ASME 2011 Pressure Vessels & Piping Division Conference, PVP2011–57493 (Jul. 2011).

ABOUT THE AUTHORS

Shigehisa Funabashi

Joined Hitachi, Ltd. in 1993, and now works at the Department of Applied Energy Systems Research, Energy and Environment Research Center, Hitachi Research Laboratory. He is currently engaged in the research and development of wind turbines. Mr. Funabashi is a member of The Japan Society of Mechanical Engineers (JSME).

Taku Iwase, Dr. Eng., P.E.Jp (Mechanical)

Joined Hitachi, Ltd. in 1995, and now works at the Department of Advanced Simulation Research, Mechanical Engineering Research Center, Hitachi Research Laboratory. He is currently engaged in the research and development of highly efficient and silent turbo machinery. Dr. Iwase is a member of the JSME, the Turbomachinery Society of Japan (TSJ), and The Institution of Professional Engineers, Japan (IPEJ).

Kiyotaka Hiradate

Joined Hitachi, Ltd. in 2005, and now works at the Department of Advanced Simulation Research, Mechanical Engineering Research Center, Hitachi Research Laboratory. He is currently engaged in the research and development of highly efficient fluid machinery with a wide operating range. Mr. Hiradate is a member of the JSME.

Masashi Fukaya, Dr. Eng.

Joined Hitachi, Ltd. in 1997, and now works at the Department of Advanced Simulation Research, Mechanical Engineering Research Center, Hitachi Research Laboratory. He is currently engaged in the research and development of prediction and evaluation systems for cavitation in nuclear power plants and fluid machinery. Dr. Fukaya is a member of the JSME, Atomic Energy Society of Japan (AESJ), and TSJ.