Fugaku Showcase

Future Manufacturing

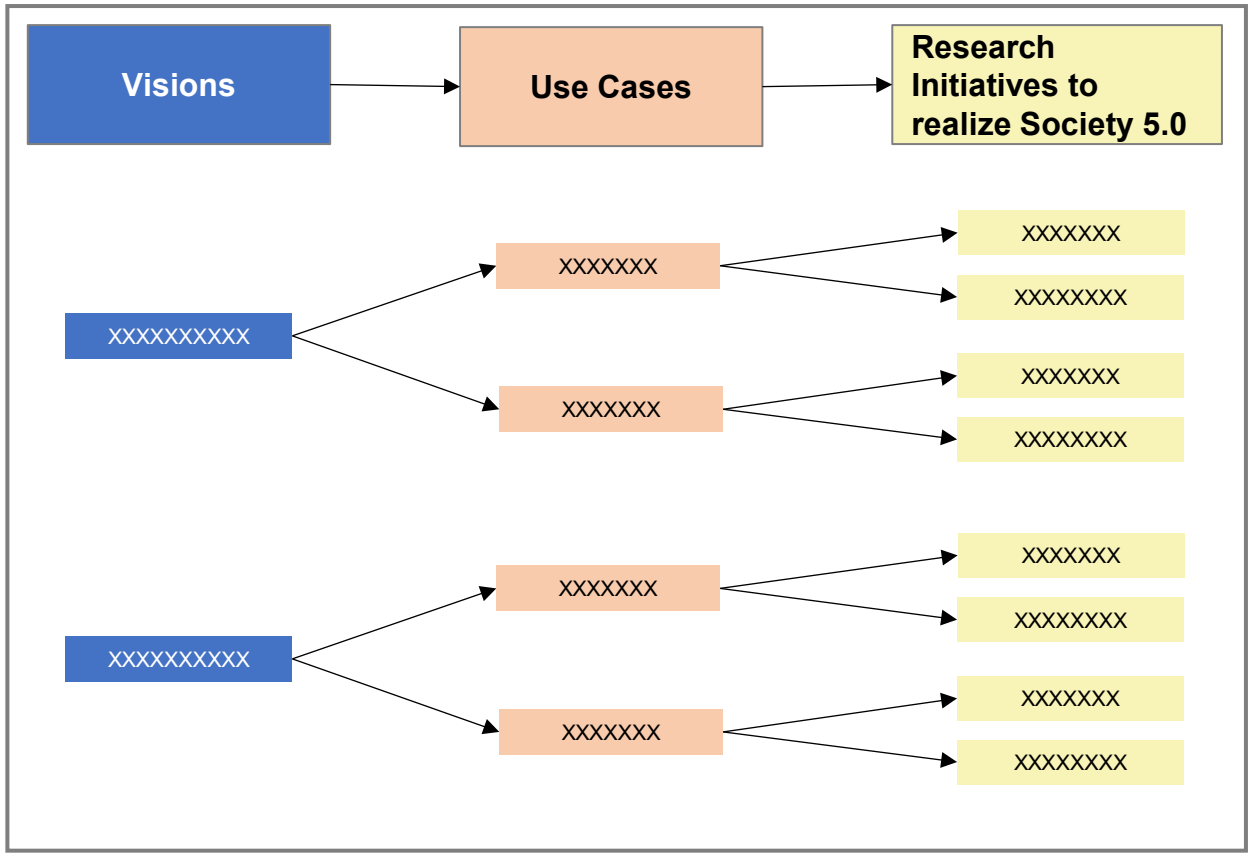
5/11/2026

RIKEN Center for Computational Science
Office of the Fugaku Society 5.0 Initiative

What is Fugaku Showcase?

The Fugaku Showcase is a collection of contents that show how the vision of Society 5.0 can be realized through research results and social implementation examples that utilize Fugaku for the several fields along with introductions to the applications. The contents present its story by backcasting: starting with the Visions which are images of the society we wish to create around 2030 through technology and business under the Society 5.0 concept, then moving to Use Cases that show how value appears in specific scenes and finally listing examples of Research Initiatives and technologies that can contribute to realizing Society 5.0.

The contents in this showcase were prepared at Office of the Fugaku Society 5.0 Initiative based on publicly accessible materials as well as materials provided by industry partners. The organization names, affiliations, and case study contents presented on the slides reflect the information available at the time of their preparation.



Visions

Images of the society we aim to achieve by about 2030 through the development of technologies and businesses, grounded in the worldview of Society 5.0.

Use Cases

Representative scenes from the Visions (assumed circa 2030) that anticipate how value will emerge through the integration of AI and simulation as further R&D advances.

Research Initiatives to realize Society 5.0

Examples of research and technologies that will contribute when considering concrete collaborative research themes aimed at realizing Society 5.0.

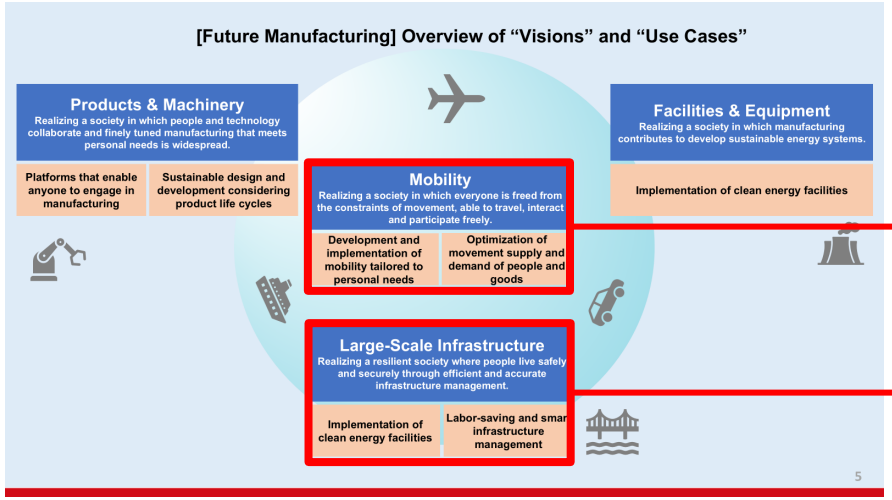
Structure of the Material

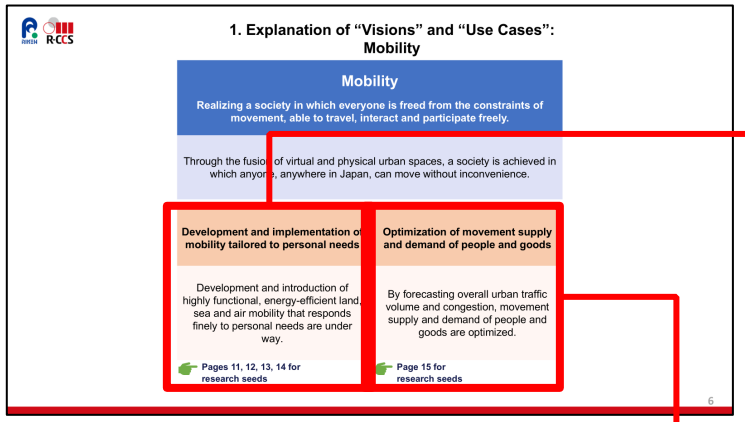
First page of each domain: An overview of the visions and use cases in the relevant domain is presented.
Second and subsequent pages of each domain: Detailed explanations of the visions and use cases in the relevant domain are provided.
Research initiatives contributing to the realization of Society 5.0 (S5): Research initiatives related to the use cases are presented.

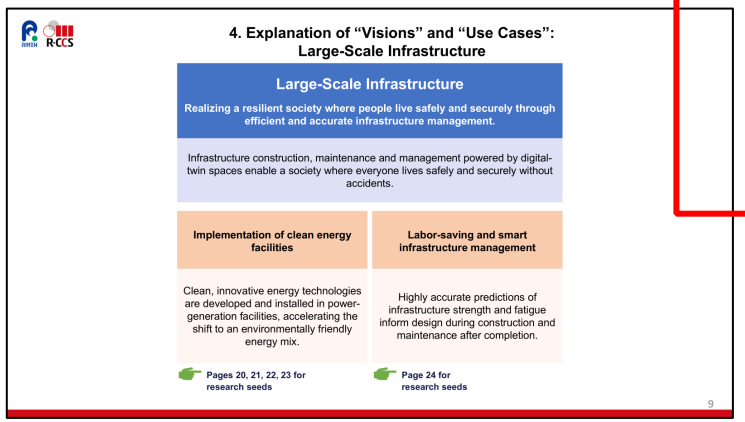
<First page>
An overview of the visions and use cases in the relevant domain is presented.

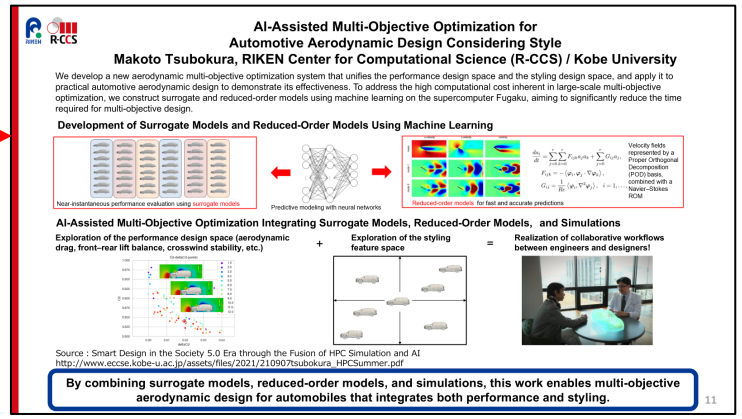
<Second and subsequent pages>
Detailed explanations of the visions and use cases are provided.

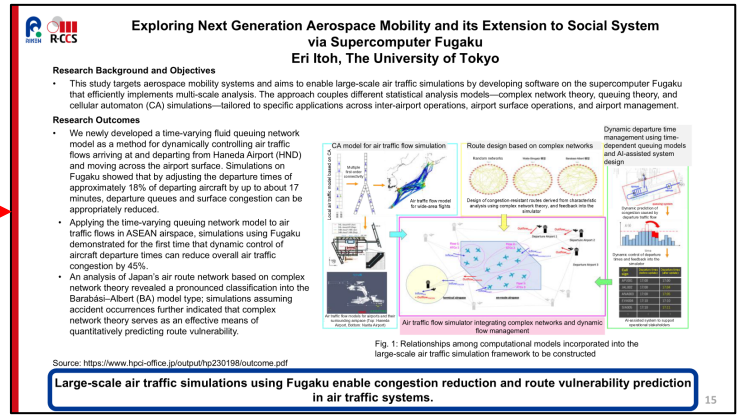
<Research initiatives contributing to the realization of Society 5.0 (S5)>
Research initiatives related to the use cases are presented.











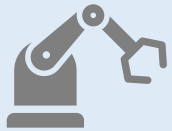
Future Manufacturing Visions and Use Cases

[Future Manufacturing] Overview of “Visions” and “Use Cases”

Products & Machinery

Realizing a society in which people and technology collaborate and finely tuned manufacturing that meets personal needs is widespread.

- Platforms that enable anyone to engage in manufacturing
- Sustainable design and development considering product life cycles



Mobility

Realizing a society in which everyone is freed from the constraints of movement, able to travel, interact and participate freely.

- Development and implementation of mobility tailored to personal needs
- Optimization of movement supply and demand of people and goods

Large-Scale Infrastructure

Realizing a resilient society where people live safely and securely through efficient and accurate infrastructure management.

- Implementation of clean energy facilities
- Labor-saving and smart infrastructure management

Facilities & Equipment

Realizing a society in which manufacturing contributes to develop sustainable energy systems.

- Implementation of clean energy facilities

1. Explanation of “Visions” and “Use Cases”: Mobility

Mobility

Realizing a society in which everyone is freed from the constraints of movement, able to travel, interact and participate freely.

Through the fusion of virtual and physical urban spaces, a society is achieved in which anyone, anywhere in Japan, can move without inconvenience.

Development and implementation of mobility tailored to personal needs	Optimization of movement supply and demand of people and goods
Development and introduction of highly functional, energy-efficient land, sea and air mobility that responds finely to personal needs are under way.	By forecasting overall urban traffic volume and congestion, movement supply and demand of people and goods are optimized.
Pages 11, 12, 13, 14 for research seeds	Page 15 for research seeds

2. Explanation of “Visions” and “Use Cases”: Products & Machinery

Products & Machinery

Realizing a society in which people and technology collaborate and finely tuned manufacturing that meets personal needs is widespread.

Demand forecasts link and coordinate the entire manufacturing process for overall optimization, enabling manufacturing that responds precisely to personal needs.

Platforms that enable anyone to engage in manufacturing	**Sustainable design and development considering product life cycles**
By accessing and using manufacturing know-how consolidated on a unified platform, anyone can participate in manufacturing.	Guided by analyses of environmental impact, material selection and production processes strike a balance between quality assurance and efficient resource circulation.

Pages 16, 17, 18, 19 for research seeds

3. Explanation of “Visions” and “Use Cases”: Facilities & Equipment

Facilities & Equipment

Realizing a society in which manufacturing contributes to develop sustainable energy systems.

The global environment is preserved, and everyone can use energy freely and sustainably.

Implementation of clean energy facilities

Clean, innovative energy technologies are developed and installed in power-generation facilities, accelerating the shift to an environmentally friendly energy mix.

Pages 20, 21, 22, 23 for research seeds

4. Explanation of “Visions” and “Use Cases”: Large-Scale Infrastructure

Large-Scale Infrastructure

Realizing a resilient society where people live safely and securely through efficient and accurate infrastructure management.

Infrastructure construction, maintenance and management powered by digital-twin spaces enable a society where everyone lives safely and securely without accidents.

Implementation of clean energy facilities	Labor-saving and smart infrastructure management
Clean, innovative energy technologies are developed and installed in power-generation facilities, accelerating the shift to an environmentally friendly energy mix.	Highly accurate predictions of infrastructure strength and fatigue inform design during construction and maintenance after completion.

Pages 20, 21, 22, 23 for research seeds

Page 24 for research seeds

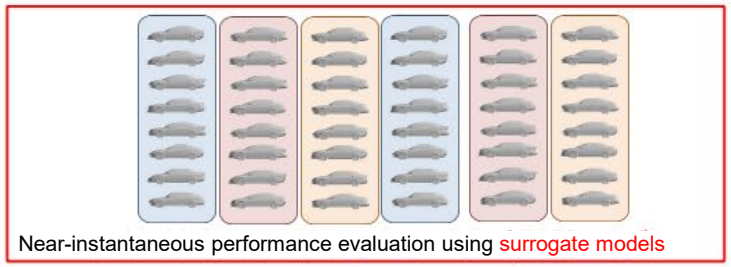
Future Manufacturing
～Research initiatives that contribute to the realization of Society 5.0～

AI-Assisted Multi-Objective Optimization for Automotive Aerodynamic Design Considering Style

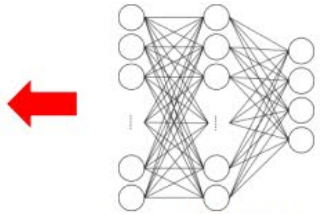
Makoto Tsubokura, RIKEN Center for Computational Science (R-CCS) / Kobe University

We develop a new aerodynamic multi-objective optimization system that unifies the performance design space and the styling design space, and apply it to practical automotive aerodynamic design to demonstrate its effectiveness. To address the high computational cost inherent in large-scale multi-objective optimization, we construct surrogate and reduced-order models using machine learning on the supercomputer Fugaku, aiming to significantly reduce the time required for multi-objective design.

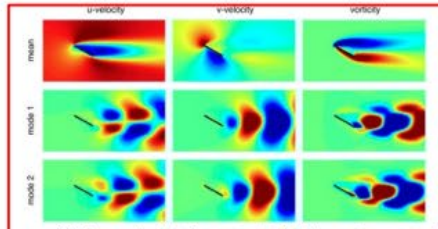
Development of Surrogate Models and Reduced-Order Models Using Machine Learning



Near-instantaneous performance evaluation using surrogate models



Predictive modeling with neural networks



$$\frac{da_i}{dt} = \sum_{j=0}^{r}\sum_{k=0}^{r} F_{ijk}a_j a_k + \sum_{j=0}^{r} G_{ij}a_j,$$

$$F_{ijk} = -\langle \varphi_i, \varphi_j \cdot \nabla\varphi_k \rangle,$$

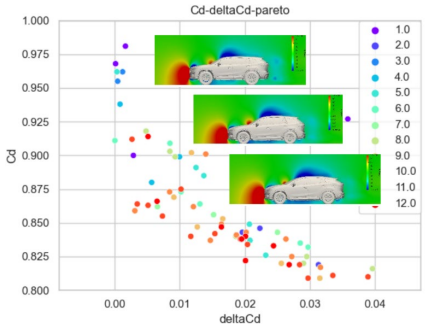
$$G_{ij} = \frac{1}{Re}\langle \varphi_i, \nabla^2\varphi_j \rangle, \quad i = 1, \ldots,$$

Reduced-order models for fast and accurate predictions

Velocity fields represented by a Proper Orthogonal Decomposition (POD) basis, combined with a Navier–Stokes ROM

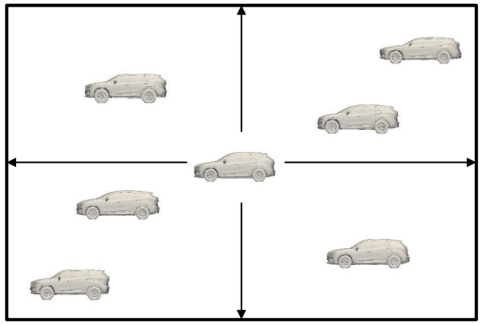
AI-Assisted Multi-Objective Optimization Integrating Surrogate Models, Reduced-Order Models, and Simulations

Exploration of the performance design space (aerodynamic drag, front–rear lift balance, crosswind stability, etc.)



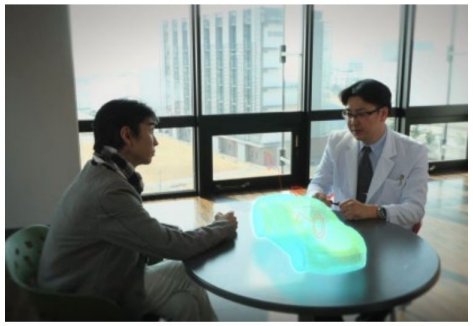
+

Exploration of the styling feature space



=

Realization of collaborative workflows between engineers and designers!



Source : Smart Design in the Society 5.0 Era through the Fusion of HPC Simulation and AI
http://www.eccse.kobe-u.ac.jp/assets/files/2021/210907tsubokura_HPCSummer.pdf

By combining surrogate models, reduced-order models, and simulations, this work enables multi-objective aerodynamic design for automobiles that integrates both performance and styling.

Contributing to Fuel Efficiency Improvement and Shorter Vehicle Development Cycles via Accelerated In-Cylinder Combustion Analysis: A Research of Ultra-highly Efficient Engine for Vehicle by HINOCA The Research Association of Automotive Internal Combustion Engines

1. Objective

- In-cylinder phenomena of automotive internal combustion engines involve complex interactions among physical and chemical processes with widely differing temporal and spatial scales. Accurately reproducing these phenomena to derive guidelines for high-efficiency combustion requires enormous computational resources.
- This study performed large-scale engine combustion simulations on the supercomputer Fugaku, targeting four research topics related to gasoline engines, one topic for diesel engines, and one topic common to both engine types.

2. Overview

- Leveraging Fugaku's computational power, the study conducted detailed simulations that are generally difficult for companies or universities. Six sub-research topics were pursued to support the development of ultra-high-efficiency automotive internal combustion engines aimed at achieving carbon neutrality.

3. Achievements

- Cycle-to-cycle variation simulations were conducted for multiple intake port shapes achievable only on Fugaku, allowing the selection of ports with high turbulence intensity and low fluctuation.
- The potential for high-efficiency gasoline combustion using ultra-lean rapid combustion combining secondary-chamber combustion and autoignition combustion was demonstrated.
- Effective in-cylinder water injection timing and direction were identified.
- Engine combustion simulations using a single type of heat-shield coating feasible at the time were conducted, revealing potential thermal efficiency improvements.
- Combustion simulations for large-truck diesel engines were performed to support further efficiency improvement studies.
- Computation of particulate matter highlighted that near-wall conservation strongly affects results, identifying a key future research challenge.

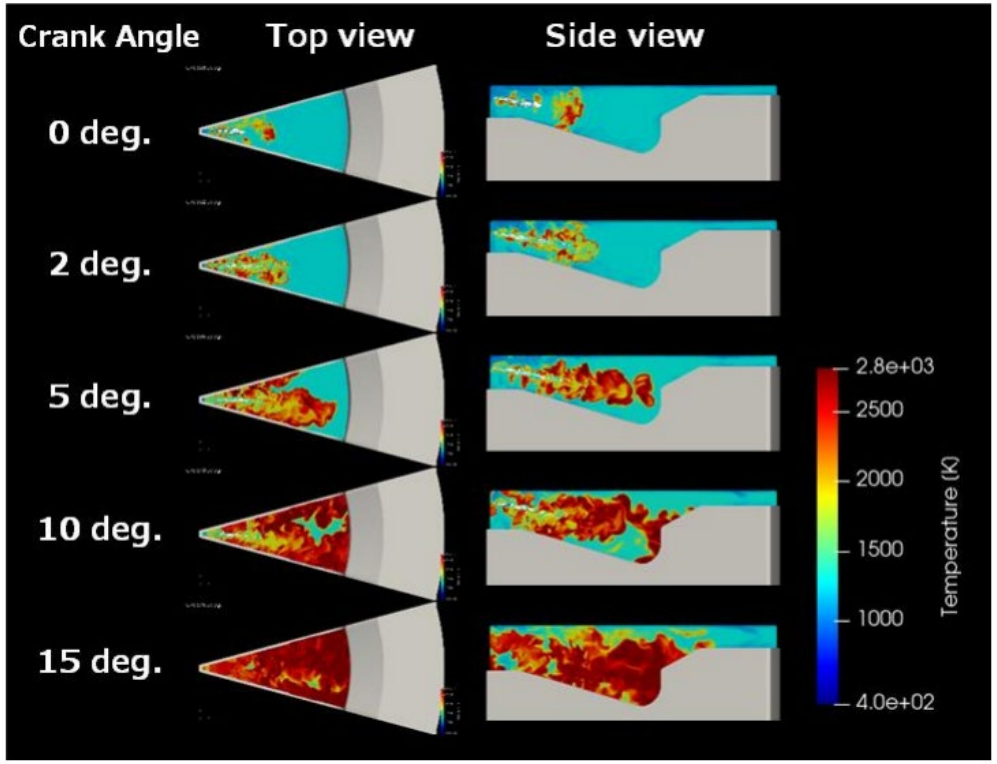


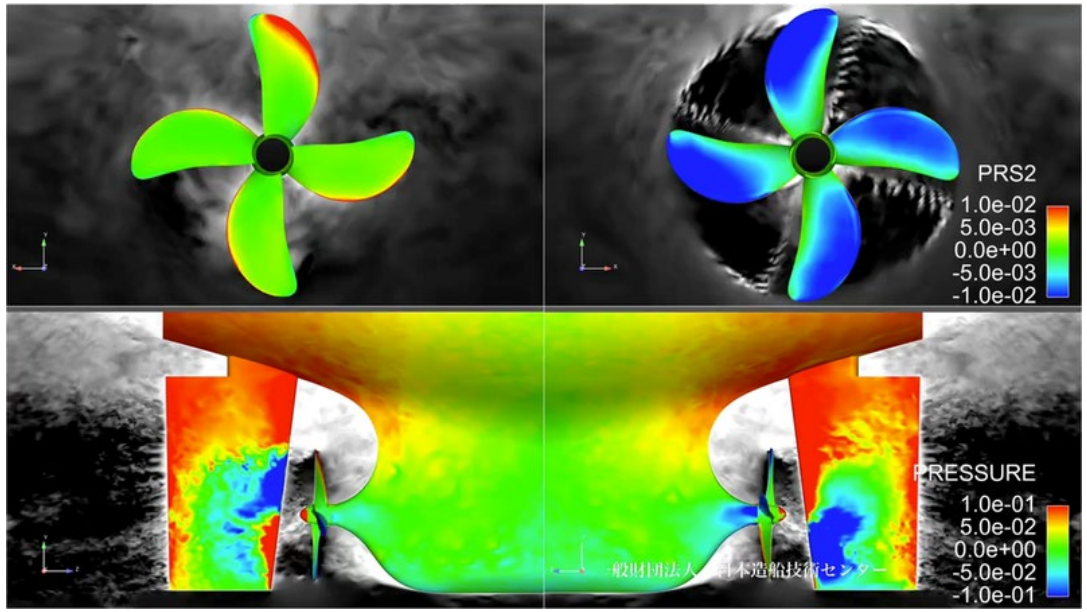
Fig. 1: Visualization example of large diesel engine combustion simulation (temperature distribution)

Source: https://www.hpci-office.jp/output/hp230269/outcome.pdf

Enabling a Numerical Towing Tank and Improving Propulsion Efficiency through Energy-Saving Devices

Chisachi Kato[1], Tatsuo Nishikawa[2],
[1]The University of Tokyo, [2]Shipbuilding Research Centre of Japan

Wall-resolved large eddy simulations (LES) with up to 120 billion elements enabled full resolution of the turbulent boundary layer around ships, contributing to the development of more efficient energy-saving devices. Large-scale numerical simulations using applications tuned to fully exploit the performance of the supercomputer Fugaku demonstrated that such simulations can potentially substitute for conventional ship performance tests.



(Image Source: Shipbuilding Research Centre of Japan)

Simulation of Interference Effects Between Propeller and Surrounding Flow (left figure)

The water flow around the hull interacts with propeller rotation, affecting ship propulsion performance. Self-propulsion trials are typically conducted to measure these interference effects; however, it is difficult to obtain detailed flow data around the hull and propeller in such experiments. Simulations can provide detailed flow information, thus enabling more advanced ship design. Pressure distribution is shown in color, while velocity distribution is shown in grayscale.

Sources: https://www.fugaku-pj.iis.u-tokyo.ac.jp/2020/theme_1.html,
https://www.hpci-office.jp/output/hp220172/outcome.pdf, https://fugaku100kei.jp/mag/03/

Large-scale numerical simulations using Fugaku demonstrated the potential to substitute for ship performance tests, contributing to advanced ship design.

High-Fidelity Full-Aircraft Simulations Partially Replacing Physical Wind Tunnel Tests: Research Toward DX in Aircraft Development Led by Digital Flight

Soshi Kawai, Tohoku University

Overview

- Leveraging high-fidelity Computational Fluid Dynamics (CFD) and the supercomputer Fugaku, we demonstrated the world's first predictive evaluation of full-aircraft aerodynamics under real-flight, high-Reynolds-number conditions using compressible Large Eddy Simulation (LES).
- In collaboration with the Mitsubishi Heavy Industries Group, we aim to establish next-generation development processes in which high-fidelity LES substitutes physical flight tests.
- The FFVHC-ACE compressible LES solver developed in this project will be broadly released as a foundational HPC solver for aerospace in industry and academia.

Research Results

- For complex full-aircraft geometries including high-lift devices, we automated the generation of hierarchical, uniformly spaced Cartesian grids directly from geometry input, enabling robust, high-fidelity wall-modeled LES.
- Wall-modeled LES around realistic high-Re aircraft and LES around a baseline wing required tens of billions of grid points; these simulations were realized only with FFVHC-ACE on Fugaku, producing unprecedented insights into aerodynamic performance.

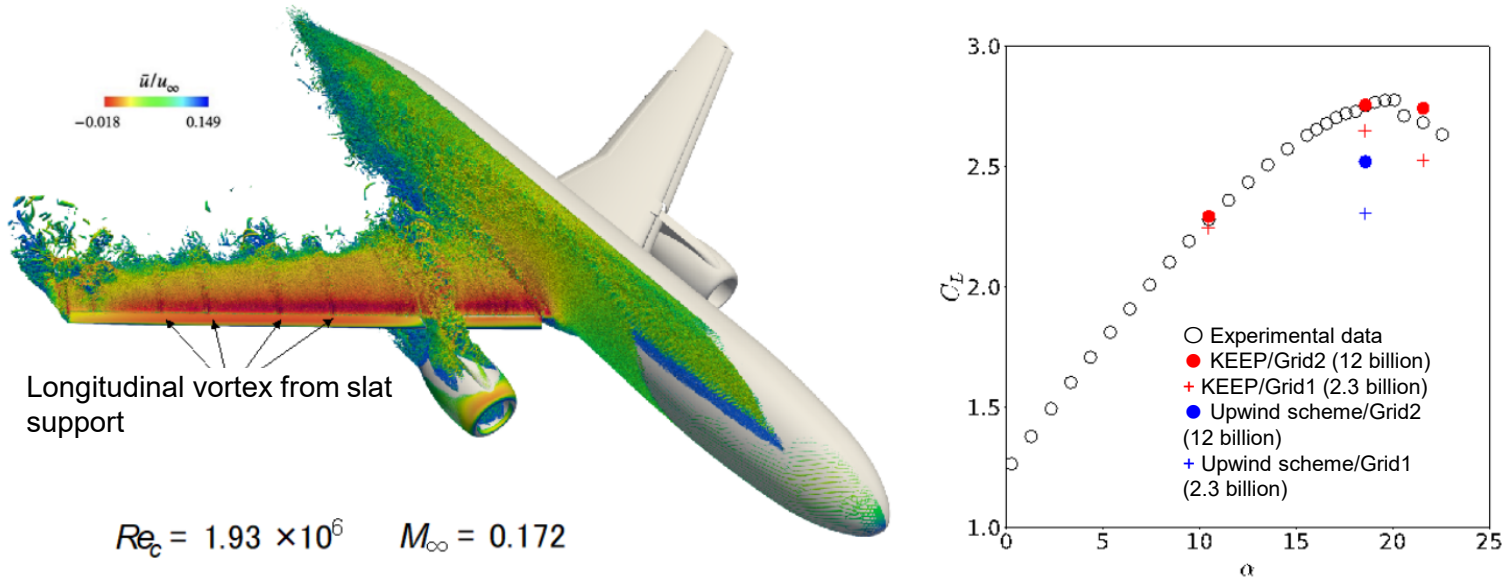


Fig. 1: Wall-Modeled LES of a Full Aircraft (JAXA Standard Model, JSM) Using FFVHC-ACE
Left: Iso-surfaces of Q-value Right: Lift coefficient versus angle of attack

Source: https://www.klab.mech.tohoku.ac.jp/fugaku/index.html, https://www.hpci-office.jp/output/hp210168/outcome.pdf

Exploring Next Generation Aerospace Mobility and its Extension to Social System via Supercomputer Fugaku

Eri Itoh, The University of Tokyo

Research Background and Objectives

- This study targets aerospace mobility systems and aims to enable large-scale air traffic simulations by developing software on the supercomputer Fugaku that efficiently implements multi-scale analysis. The approach couples different statistical analysis models—complex network theory, queuing theory, and cellular automaton (CA) simulations—tailored to specific applications across inter-airport operations, airport surface operations, and airport management.

Research Outcomes

- We newly developed a time-varying fluid queuing network model as a method for dynamically controlling air traffic flows arriving at and departing from Haneda Airport (HND) and moving across the airport surface. Simulations on Fugaku showed that by adjusting the departure times of approximately 18% of departing aircraft by up to about 17 minutes, departure queues and surface congestion can be appropriately reduced.
- Applying the time-varying queuing network model to air traffic flows in ASEAN airspace, simulations using Fugaku demonstrated for the first time that dynamic control of aircraft departure times can reduce overall air traffic congestion by 45%.
- An analysis of Japan's air route network based on complex network theory revealed a pronounced classification into the Barabási–Albert (BA) model type; simulations assuming accident occurrences further indicated that complex network theory serves as an effective means of quantitatively predicting route vulnerability.

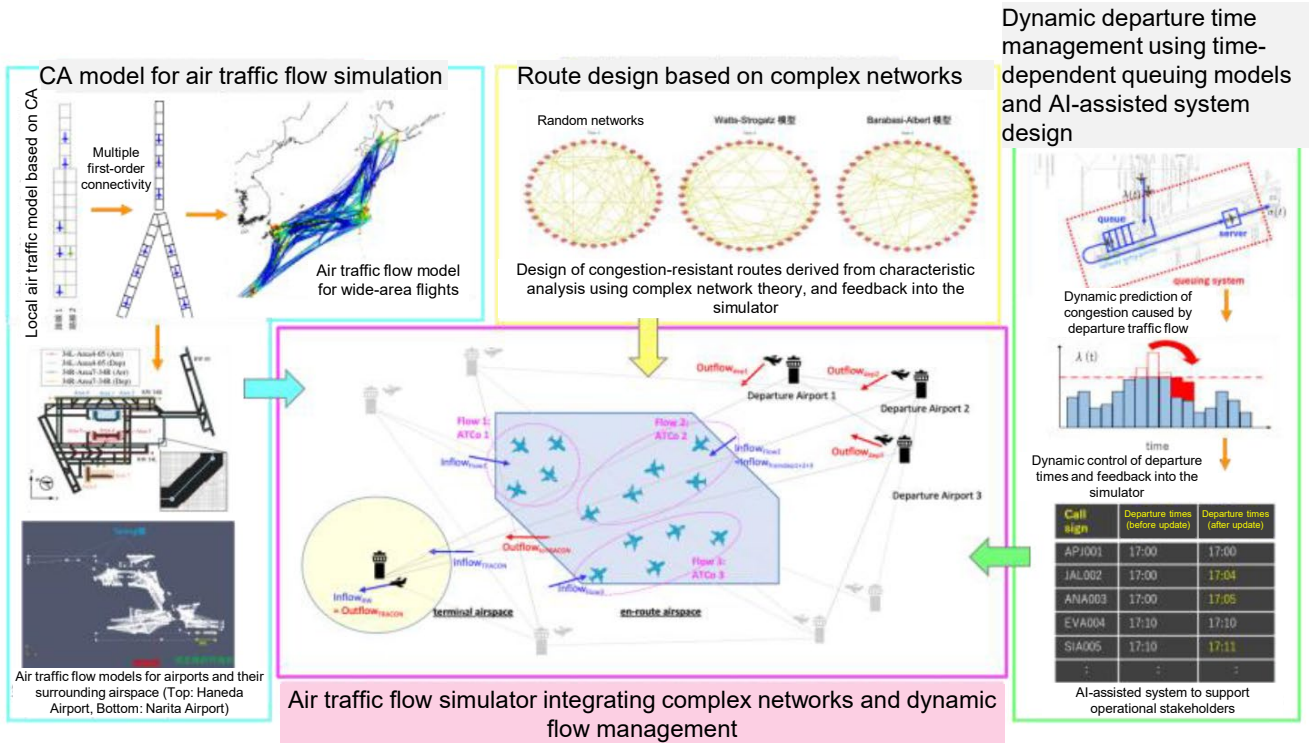


Fig. 1: Relationships among computational models incorporated into the large-scale air traffic simulation framework to be constructed

Source: https://www.hpci-office.jp/output/hp230198/outcome.pdf

Large-scale air traffic simulations using Fugaku enable congestion reduction and route vulnerability prediction in air traffic systems.

Optimization Design of Ammonia Burner for CO_2-free Combustion

Yukihiko Okumura, Kagawa University

1. Objective

- Ammonia is gaining attention as a hydrogen carrier for efficient transport and storage. In industrial applications, using ammonia directly as a fuel, in addition to hydrogen production, is desirable. Despite its high calorific value (383 kJ/mol), ammonia's low flame propagation speed limits its fuel usability, and forced combustion produces significant NOx. Achieving CO_2-free combustion requires optimal ammonia burner design and a detailed understanding of NOx formation and reduction mechanisms.
- This study investigated reliable ammonia reaction mechanisms to enable optimized design of CO_2-free ammonia burners.

2. Overview

- Three ammonia reaction mechanisms were tested for optimizing CO_2-free ammonia burners. Computational results were compared with experimental data (flame structure and NOx emissions) to select the most accurate and optimal scheme for NH_3/H_2 fuel.

3. Results

- Analyses using the CRECK-Mech scheme showed good agreement with experimental flame structures and NOx emission levels. Under H_2/NH_3 fuel conditions, GRI-Mech3.0 and Okafor-Mech reaction progress are underestimated.
- CRECK-Mech clarified NOx formation and reduction pathways: turbulence-supplied OH and H radicals induce NOx formation, while simultaneous mixing and reduction with NH_3 convert NO to N_2.
- NOx reduction is strongest near formation zones, particularly where NH_3 is rich, preventing NOx accumulation. Therefore, actively positioning reduction zones where NOx can fully react with N, NH, and NH_2 is essential in burner design.
- A hydrogen-stabilized ammonia turbulent combustion burner was successfully developed.

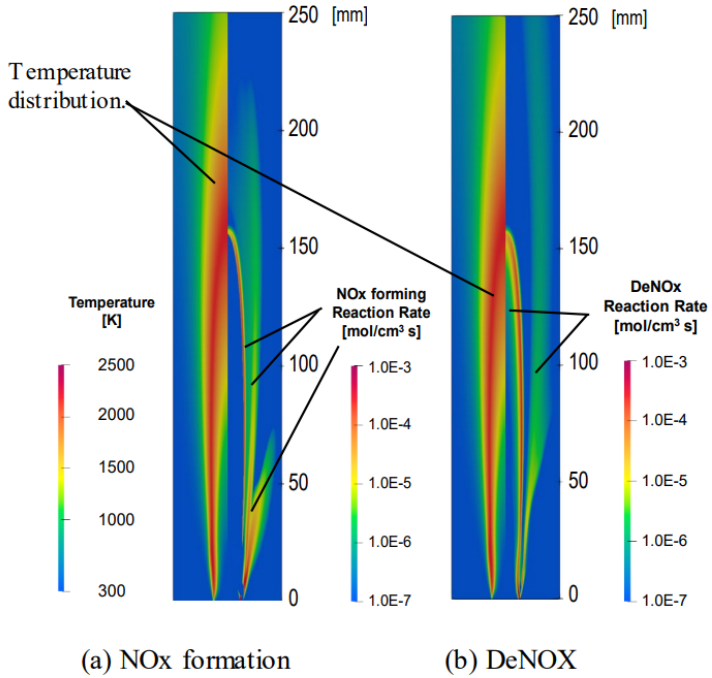


Fig. 1: Temperature distribution (left) and reaction rate distributions of (a) NOx formation and (b) NOx destruction (CRECK-Mech).

Sources: https://www.hpci-office.jp/output/hp200176/outcome.pdf,
https://www.hpci-office.jp/output/hp210108/outcome.pdf

Numerical simulations of flame speed and structure, validated against experiments, enabled optimized design of ammonia burners for CO2-free combustion.

Simulation of Fan Turbulent Flow Field by Large-Scale LES Analysis and Examination of Air Conditioning and Cooling Technologies
DENSO CORPORATION

1. Objective

- As vehicle electrification progresses toward a carbon-neutral society, the miniaturization and energy-saving of in-vehicle Heating Ventilation and Air Conditioning (HVAC) and cooling systems become increasingly critical. Miniaturization places fans close to heat exchangers, causing disturbed fan outflow to enter the heat exchanger without smoothing. For further efficiency improvement, optimizing heat exchangers and fans while considering turbulence is essential.
- This study aims to clarify the effects and mechanisms of fan outflow turbulence on heat transfer at heat exchanger surfaces.

2. Overview

- A model representing the louver fin rows of an in-vehicle radiator heat exchanger was analyzed. Large-scale LES (Large Eddy Simulation) was performed using the unstructured-grid CFD solver NuFD/FrontFlowRed to quantify how turbulent inflow affects the heat transfer performance of the heat exchanger surfaces.

3. Results

- The influence of fan outflow turbulence on heat exchanger surfaces was clarified:
  - (1) The area-averaged heat transfer coefficient of fins in turbulent flow increased by approximately 10% compared to uniform flow.
  - (2) Enhancement was most significant at the 1st and 3rd louvers from the front, where turbulence refined vortical structures on the fin surfaces and promoted heat exchange with the main flow.
- Large-scale LES on Fugaku visualized fine vortical structures around the louvers, enabling detailed discussion of heat transfer enhancement mechanisms. Future experimental evaluation will verify the hypotheses derived from these LES insights.

Source: https://www.hpci-office.jp/output/hp220122/outcome.pdf

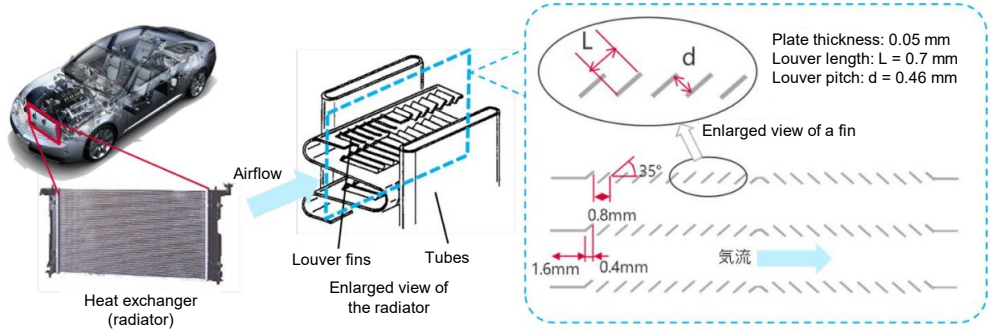


Fig. 1: Model of louver fin rows

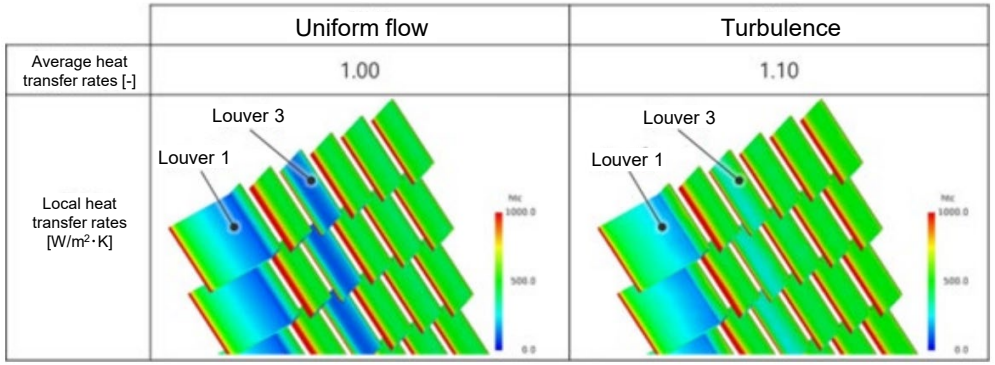


Fig. 2: Heat transfer performance and local heat transfer coefficient distribution

Large-scale LES on Fugaku clarified the impact of turbulence on in-vehicle heat exchangers, contributing to the design of energy-efficient, high-performance HVAC systems.

Performance Improvement of Heat Pump Fans

Chisachi Kato, Yosuke Hasegawa, The University of Tokyo

Objective

- Surrogate models were utilized to explore a wide range of design parameters previously unexamined, enabling the design of a propeller fan that significantly surpasses conventional performance.

Overview

- A fundamental review and redesign of heat pump propeller fan parameters was conducted.
- Fluid analyses were performed using RANS (Reynolds-Averaged Navier-Stokes Simulation) and LES (Large Eddy Simulation) to predict fan performance and flow fields.
- The solver FrontFlow/blue (FFB), optimized for high-speed computation on Fugaku CPU nodes, was employed.

Results

- An environment for design and modeling based on 3D measurements of actual fans was established.
- The design parameter range for heat pump propeller fans was greatly expanded, enabling concept design of improved thin-profile fans equipped with diffusers.
- Although detailed optimization of individual components has not yet been performed, predictions indicate that the improved fan outperforms the current fan both as a standalone unit and when integrated into its housing, confirming the effectiveness of the concept design.

Source: https://www.rist.or.jp/sc/report/r05/f321_r05.pdf

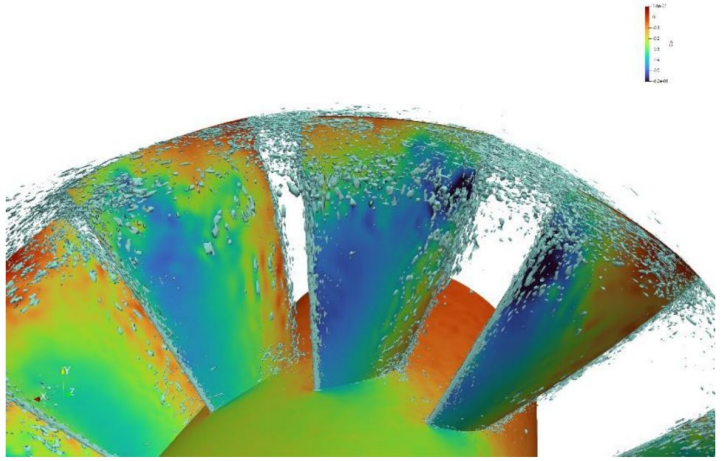


Fig. 1: Instantaneous distribution of static pressure coefficients obtained from LES using FFB

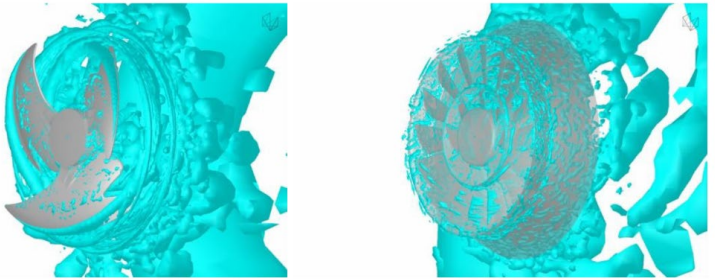


Fig. 2: Comparison of vortex structures around blades for the current fan (left) and improved fan (right)

High-speed fluid analysis and surrogate modeling enabled broad parameter exploration, resulting in a propeller fan that surpasses conventional performance.

Mist Flow Simulation of Numerous Very Small Droplets for Plumbing Products
TOTO LTD.

1. Objective

- To develop “sustainable products” for water-related spaces that combine eco-friendliness, comfort, and cleanliness, a simulation framework was previously established using the supercomputer Fugaku to calculate gas-liquid multiphase flows of 1 mm-scale microdroplets and thin-film flows in showers and bathrooms for product performance evaluation.
- In unstructured-grid fluid simulations, multiphase flow with 150 million mesh points in the fine internal structure of showers became feasible within practical computation time, demonstrating that multiphase flow simulations, previously limited to sanitary ceramic products, can also be applied to faucets, showers, bathrooms, and other components.
- This research aims to establish simulation technology for 0.1 mm-scale mist microdroplet dispersion and product performance evaluation, supporting future mist water product development.。

2. Overview

- A numerical model was developed to calculate gas-liquid multiphase flows, such as microdroplets and thin-film flows in showers and bathrooms, establishing simulation technology for 0.1 mm-scale mist microdroplet dispersion and product performance evaluation.

3. Results

- Using the simulation framework for shower water ejection, dispersion, and impingement, a hollow-cone-type mist generation simulation was conducted, confirming that thin water films break up to form mist.
- To evaluate microdroplet dispersion, air resistance was modeled and strongly coupled with air flow on an orthogonal grid using the SPH particle method. This allowed, for the first time in-house, practical calculation of microdroplets dispersing while decelerated by air resistance. By modeling skin grooves and dirt, it was shown that mist showers have higher cleaning performance than conventional showers.
- Utilizing these fluid simulations, a technology was established to pre-evaluate comfort, hygiene, and cleaning performance of various products involving mist water, not limited to showers.

Sources: https://www.hpci-office.jp/output/hp210013/outcome.pdf, https://www.hpci-office.jp/output/hp230239/outcome.pdf

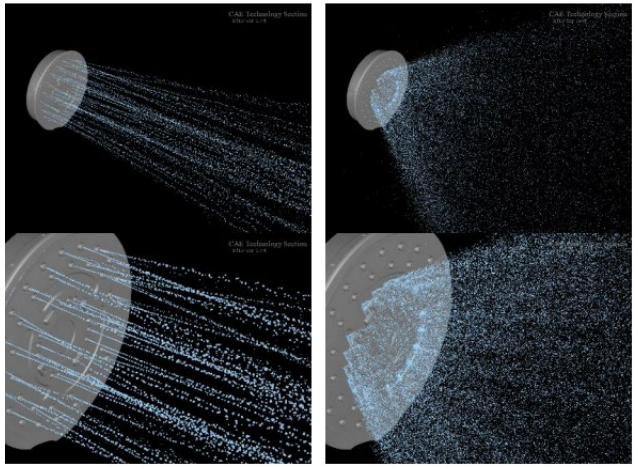


Fig. 1: Simulation results of conventional shower (left) and mist shower (right) water ejection

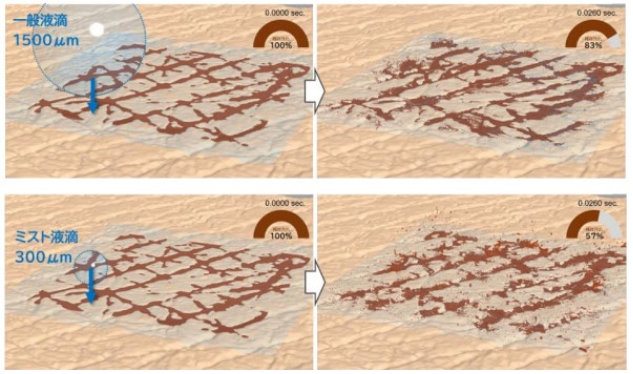


Fig. 2: Simulation of droplets impacting skin and removing dirt: conventional shower (top), mist shower (bottom)

Gas-liquid multiphase flow analysis of microdroplets and thin-film flows enables pre-evaluation of comfort, hygiene, and cleaning performance of products such as mist showers.

Construction and Utilization of a Digital Twin for Supercritical CO_2 Turbines

Shinobu Yoshimura, The University of Tokyo

Research Background and Objective

The objective of this study is to construct a digital twin of next-generation thermal power generation systems, such as supercritical CO_2 turbines, which are suitable for the application of CCS (Carbon dioxide Capture and Storage) technology and will serve as a key clean energy system supporting Society 5.0.

Results

- Using FFR-Comb with a Flamelet/Artificial Neural Network (ANN) combustion model, 20,000 parallel calculations were performed on the supercomputer Fugaku. Large-scale LES (Large Eddy Simulation) combustion analysis of the entire combustor was completed in about three days, demonstrating the effectiveness of this approach for accurately predicting supercritical combustion under actual operating pressures.
- High-fidelity super-simulations of a pilot-scale supercritical CO_2 turbine (Toshiba ESS at 28.5 MPa were conducted on Fugaku, providing insights for the development and optimization of full-scale turbines.
- Comparisons of combustor outlet temperature distributions with measured and assumed values confirmed the high accuracy of the simulations.
- A high-precision, high-efficiency surrogate combustion model was developed by combining the Flamelet method with AI (machine learning) and implemented in FFR-Comb, establishing a digital twin of the supercritical CO_2 turbine. This approach significantly reduced the database size needed for the Flamelet method and the memory usage of the simulations while maintaining predictive accuracy.

Source: https://www.hpci-office.jp/output/hp220169/outcome.pdf

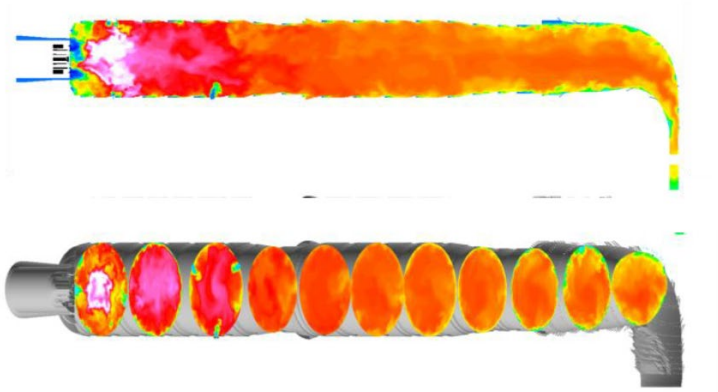


Fig. 1: Instantaneous temperature distribution at the combustor's center and vertical cross-sections.

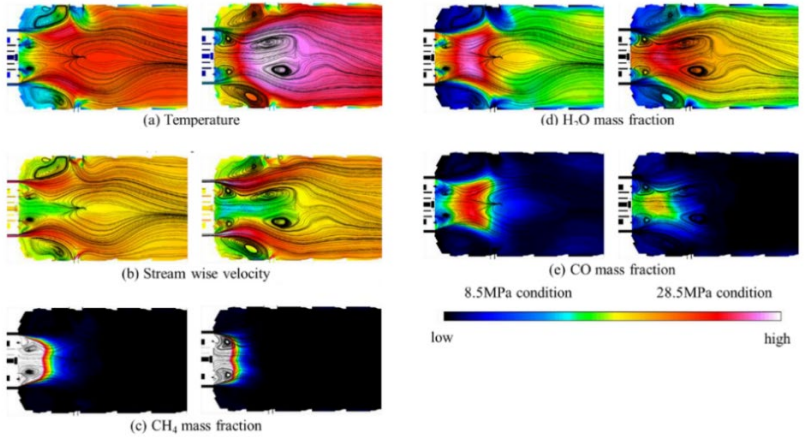


Fig. 2: Time-averaged temperature, velocity, and species mass fractions (CH_4, H_2O, CO) near the burner in the central cross-section.

High-fidelity prediction of supercritical combustion under actual operating pressures of supercritical CO_2 turbines, contributing to next-generation power systems supporting Society 5.0

Construction and Application of a Digital Twin for Large Offshore Wind Turbines in Tandem Arrangement

Shinobu Yoshimura, The University of Tokyo

Research Background and Purpose

- To support Society 5.0, this project aims to construct a digital twin of offshore wind farms (e.g., tandem-arranged large offshore wind turbines), suitable for application in clean energy systems integrated with Carbon dioxide Capture and Storage (CCS).

Results

- Three turbines arranged in tandem at 7.5 rotor diameters—a spacing representative of practical layouts—were simulated using Large Eddy Simulation with FrontFlow/blue (FFB-LES), clarifying wake-recovery characteristics. Such large-scale LES became feasible on the supercomputer Fugaku.
- Using the 5-MW reference offshore wind turbine model published by the National Renewable Energy Laboratory (NREL), we performed detailed FFB-LES on 400 Fugaku nodes; the resulting near- and far-wake flow fields enabled construction of a wake distribution model capable of accurate near- and far-wake predictions.
- We established a computational environment for cumulative fatigue-damage analysis on Fugaku, enabling integrated fluid–structure–fatigue simulations for tandem turbine models.

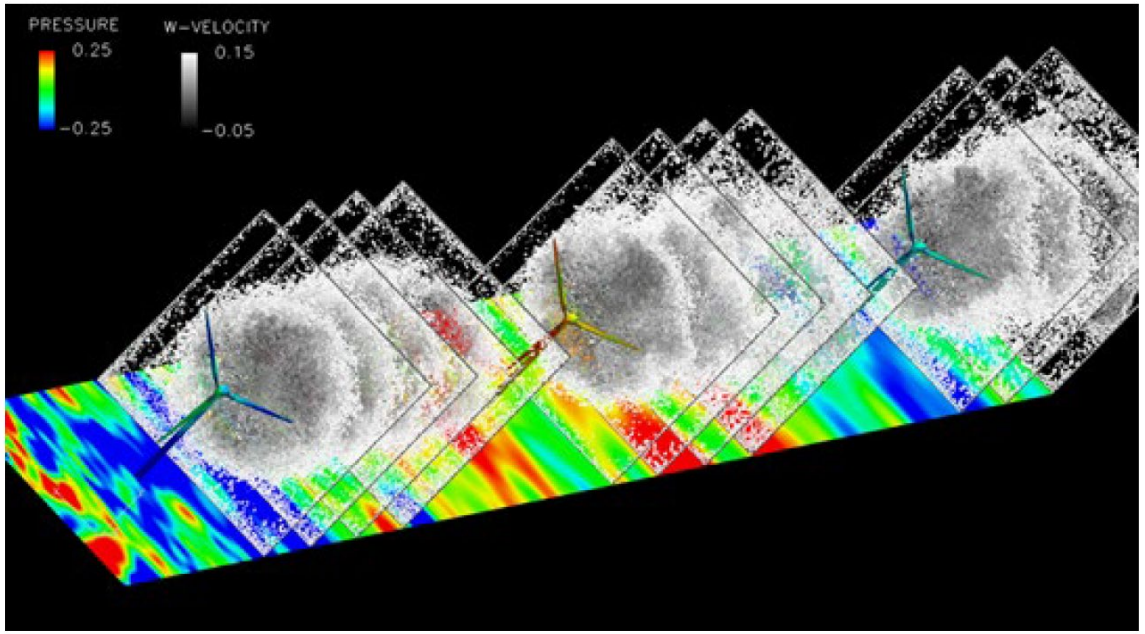


Fig. 1. Bird's-Eye View of Flow Fields and Free-Surface Pressure Around a Wind Farm

Sources: https://www.hpci-office.jp/output/hp210175/outcome.pdf
https://www.hpci-office.jp/output/hp220169/outcome.pdf

Constructed a digital twin of offshore wind farms, enabling sequential execution of fluid dynamics, structural, and fatigue analyses on the supercomputer Fugaku.

Exploration of Burning Plasma Confinement Physics
Tomo-Hiko Watanabe, Nagoya University

Research Background and Objective

To achieve scientific demonstration of magnetic confinement fusion, it is critical to understand how plasma density and temperature distributions are formed inside devices such as ITER, and how the plasma responds to spontaneously generated perturbations accompanied by electromagnetic field fluctuations. This study employs large-scale first-principles (ab initio) simulations on the supercomputer Fugaku to investigate particle, momentum, and heat transport induced by turbulent magnetic confinement fusion plasmas, as well as the transport mechanisms of high-energy particles.

Multiscale Turbulence Simulation of Nuclear Fusion Plasmas

Using the GKV code optimized for Fugaku, we conducted multiscale turbulence simulations of fuel–ash mixed plasmas and high-electron-temperature regions representative of nuclear fusion plasmas. We newly discovered that interactions between different turbulence scales can reduce the electron heat flux.

Optimization of Global Turbulent Transport Analysis Code on Fugaku

By utilizing a communication-reducing matrix solver with variable-precision arithmetic, which became available on Fugaku for the first time, we demonstrated that ITER-scale numerical experiments can be executed using thousands of Fugaku nodes within practical computation times.

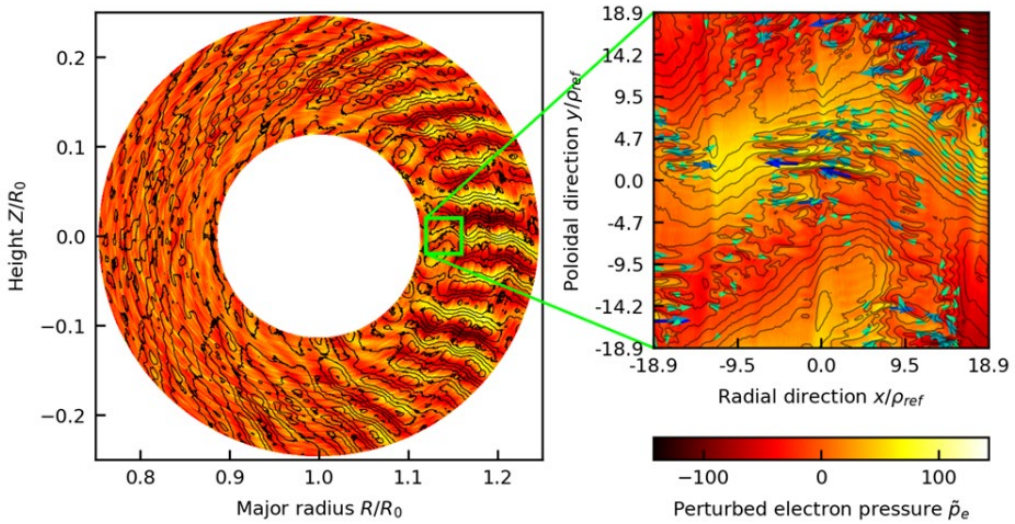


Fig. 1: electron pressure fluctuation $\tilde{p}_e$ color map and poloidal cross-section of turbulent fluctuation streamlines (black lines)

The left panel shows relatively large-scale TEM fluctuations, with streamlines extended radially; high-temperature regions are transported outward, and low-temperature regions inward. The right panel shows coexistence of finer ETG mode turbulence fluctuations. This suggests that electron-scale turbulence modulates the electron pressure fluctuations carried by TEM radial flows, inhibiting TEM turbulence development and affecting net heat flux.

Source: https://www.hpci-office.jp/output/hp200127/outcome.pdf

Large-scale nuclear fusion plasma simulations on Fugaku enable ITER-scale numerical experiments.

Evaluation of Multi-physics and Multi-scale Transient Response of the Commercial-scale Gasifier of IGCC Power Plant Mitsubishi Heavy Industries, Co., Ltd.

1. Research Background and Objective

- To reduce CO_2 emissions in power generation, short- to medium-term strategies include improving the efficiency of fossil fuel power generation and integrating CO_2 capture and storage (CCS) technologies. In the medium- to long-term, power generation using blue hydrogen produced from lignite gasification is anticipated. Gasifiers are the core equipment for these technologies, and the development of high-efficiency, stable gasifiers is strongly required.
- This study aims to conduct Large Eddy Simulation (LES) of the thermal–fluid–reaction fields inside a full-scale IGCC gasifier and to capture phenomena such as the development and detachment of molten slag layers on the gasifier walls under transient response conditions through one-way coupled structural analysis.

2. Overview

- A gas–solid–liquid three-phase LES is applied to a full-scale IGCC gasifier by coupling the Volume of Fluid (VOF) method for molten slag flows with conventional Eulerian–Lagrangian two-phase LES of gas–solid reactive flows.
- Non-steady two-layer gas–solid reactive flow analysis within the IGCC gasifier is performed, and one-way coupled simulations are conducted with heat transfer and structural/solid–liquid slag layer models. Unit-cell models considering solid–liquid separation in the slag layer are used for heat transfer and structural analysis.

3. Results

- The three-phase LES coupled with VOF produced results comparable to conventional LES for the reactive flow fields in the full-scale gasifier.
- The formation of molten slag layers from ash particle deposition on the gasifier wall was simulated. Although qualitatively consistent with observed phenomena, the very large time constants of molten slag flow indicate the need for future computational acceleration.
- One-way coupled heat transfer and structural/solid–liquid slag layer simulations were performed. From the stress distribution in the slag layer during rapid load changes and heating, potential failure regions of the solid slag layer were successfully evaluated.

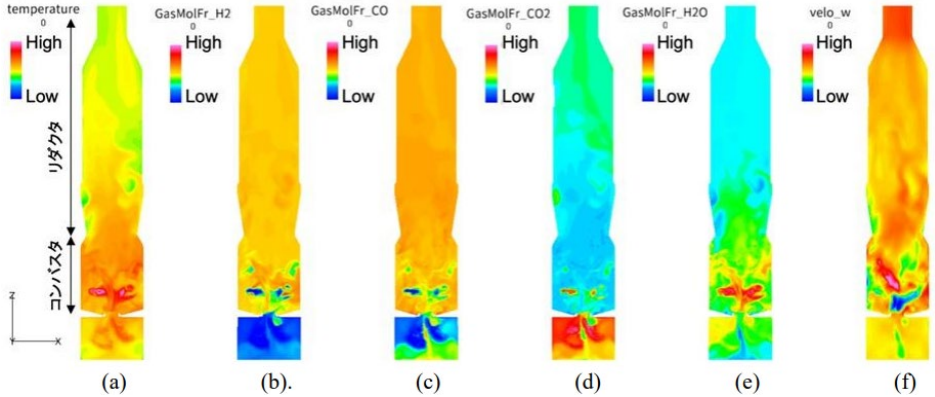


Fig. 1: Instantaneous distributions of physical quantities inside the gasifier: (a) Gas temperature, (b) H_2 mole fraction, (c) CO mole fraction, (d) CO_2 mole fraction, (e) H_2O mole fraction, (f) Axial velocity (no slag condition).

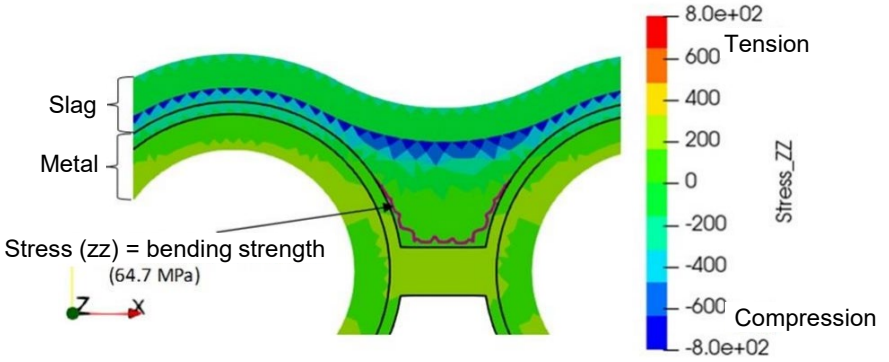


Fig. 2: Stress distribution within the slag layer. Although the model does not incorporate solid slag fracture, the red line indicates where fracture might occur if included.

Source: https://www.hpci-office.jp/output/hp230091/outcome.pdf

Gas–solid–liquid three-phase LES of a full-scale IGCC gasifier enables the development of high-efficiency, stable gasifiers for CO_2 emission reduction.

Wind-Resistant Design of an Entire Long-Span Cable-Stayed Bridge Using Fluid-Structure Interaction Analysis
Hanshin Expressway Company Limited

1. Objective of R&D

- As part of the wind-resistant design of one of the world's largest multi-span continuous cable-stayed bridges (total length: approximately 2,700 m), this study aims to verify the aerodynamic safety of the bridge by conducting fluid–structure interaction (FSI) analysis targeting the entire bridge structure.

2. Overview

- Vibration phenomena previously confirmed through wind tunnel tests of a standalone pylon (main tower) model and a main-girder cross-sectional model were computed using FSI analysis. Their reproducibility was examined, and analyses modeling the entire bridge were conducted to identify challenges toward application in practical wind-resistant design.
- The computations performed in this study involve an extremely large number of elements, and a long total simulation time is required to derive statistical quantities of aerodynamic vibration. In particular, the full-bridge analysis employed a computational grid of approximately 200 million elements with a total simulation time of 600 seconds. Such simulations are infeasible on conventional computers and represent cases that are realistically computable only on the supercomputer Fugaku within the HPCI environment.

3. Achievements

- Through FSI analyses of the standalone pylon and the main-girder cross section, as well as flow analyses of the entire bridge, the validity of the proposed approach was confirmed. In addition, the overall flow field around the entire bridge was visualized, providing valuable insights for wind-resistant bridge design.
- For the standalone pylon case, the results enabled visualization of wind flow around the pylon and the sequence of vortex shedding induced by structural vibration, thereby clarifying the mechanism of aeroelastic instability. For the main-girder cross-sectional case, the results demonstrated the potential of aerodynamic appendages to control aeroelastic instability of the main girder.
- In computations using the full-bridge model, it was shown that the mean drag coefficient and mean lift coefficient vary depending on the location along the main girder. This characteristic was demonstrated for the first time by a study conducting computations based on an entire-bridge model.
- In future work, based on these results, FSI analysis using the full-bridge model will be conducted to investigate the characteristics of aeroelastic instability of the entire bridge structure.

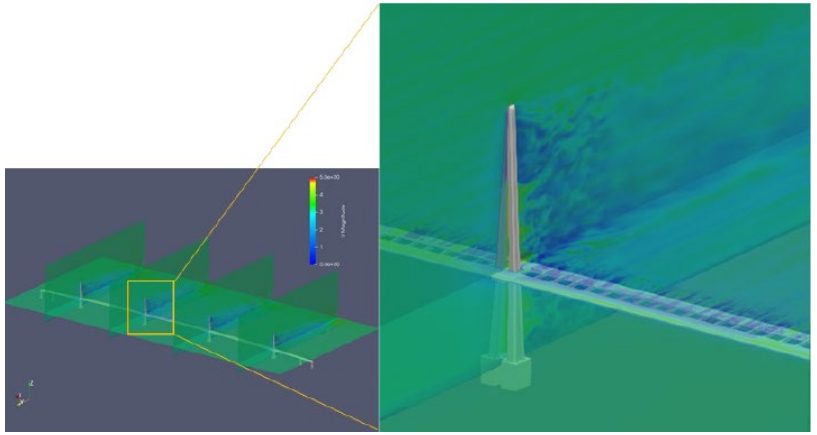


Fig. 1: Computational results for the full-bridge model (wind speed contour [m/s])

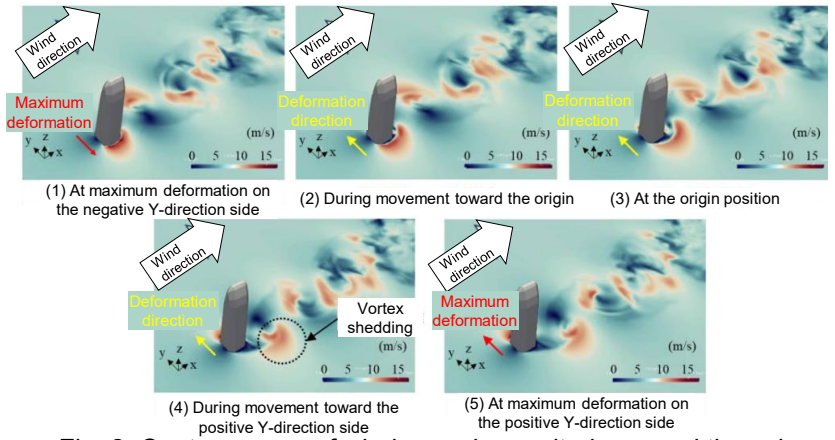


Fig. 2: Contour maps of wind-speed magnitude around the pylon

Sources: https://www.hpci-office.jp/output/hp220123/outcome.pdf, https://www.hpci-office.jp/output/hp230162/outcome.pdf

Large-scale aerodynamic flow analysis was conducted to support wind-resistant bridge design, targeting an entire 2,700 m–long bridge and employing a computational grid of approximately 200 million elements.

[Future Manufacturing] List of Social Implementation Cases (1/5)

To introduce a wide range of social implementation cases in the manufacturing fields utilizing Fugaku, relevant usage projects were extracted from user reports on the High-Performance Computing Infrastructure (HPCI).
Among the projects in which Fugaku has been utilized by industry, those whose research area is classified as "Engineering and manufacturing" are listed.

Periods of Use	Project Name	Organization Name
FY2020	Large-scale Unsteady CFD Simulation of High-speed Train	Kawasaki Heavy Industries, Ltd.
FY2020	Verification and Performance Evaluation of Commercial CAE Applications	Fujitsu Limited
FY2020	Try and evaluation of leading-edge CAE for automobiles with Fugaku supercomputer	Japan Automobile Manufacturers Association, Inc.
FY2020	validation of WHEEL	Longtail software LLC
FY2020	Trial Calculation for optimizing a structure by using Fugaku	Fuji Electric Co., Ltd.
FY2020	Simultaneous Optimization of Rotor and Stator Shape for Aeroengine Fan	IHI Corporation
FY2020	Porting of parallel GPU multiphase fluid simulation code to A64fx parallel supercomputer for developing household equipment	TOTO LTD.
FY2020	A Preliminary Study on Numerical Algorithms based on the Performance Evaluation for Practical CAE Applications	AdvanceSoft Corporation
FY2020	Development of digital twin machining simulation technology for machine tools	DMG MORI CO.,LTD
FY2020	Multiscale Modeling and Simulation of Rubber for Tyres	Sumitomo Rubber Industries, Ltd.
FY2020	Verification and Performance Evaluation of Thermo-Fluid Simulation Application scFLOW	Fujitsu Limited
FY2020	Verification and Performance Evaluation of Electromagnetic Simulation Application JMAG	Fujitsu Limited

Source: https://www.hpci-office.jp/achievements/user_report (Retrieved November 18, 2025)

[Future Manufacturing] List of Social Implementation Cases (2/5)

Periods of Use	Project Name	Organization Name
FY2021	Multiphase fluid simulation including small bubbles and droplets for application to household equipment	TOTO LTD.
FY2021	Validation of alternative technology by direct turbulence simulation for towing tank test	Shipbuilding Research Centre of Japan
FY2021	Large-scale LES analysis of flow in a cabin and investigation of the technology to prevent droplet infection	DENSO CORPORATION
FY2021	Development of leading-edge CAE for automobiles with Fugaku supercomputer	Japan Automobile Manufacturers Association, Inc.
FY2021	Research on low vibration and noise reduction of control valves using multi-stage perforated hole	Azbil Corporation
FY2021-2022	Large-scale Unsteady CFD Simulation of Aircraft	Kawasaki Heavy Industries, Ltd.
FY2021-2023	A research of ultra-highly efficient engine for vehicle by HINOCA	The Research Association of Automotive Internal Combustion Engines
FY2021	Development of digital twin test cut technology for machine tools	DMG MORI CO.,LTD
FY2021	Performance Evaluation of Aircraft Engine Combustor by Large-scale Large-Eddy-Simulation with Integrated Turbulent Atomization Model	IHI corporation
FY2021	Operation verification and evaluation of parallelization efficiency of thermal hydraulics analysis code	Mitsubishi FBR systems, Inc.
FY2021	Operation check and applicability study of CFD software	Numerical Flow Designing CO.,LTD.
FY2021	Massively Parallel Performance Improvement of Thermo-Fluid Simulation Application scFLOW	Fujitsu Limited
FY2021	Operation Verification of AVL FIRE on Fugaku	Mitsui E&S Machinery Co.,Ltd.
FY2021	Simultaneous Shape Optimization of Aeroengine Fan Rotor and Stator	IHI Corporation
FY2021	Verification and Performance Evaluation of Thermo-Fluid Simulation Application Ansys Fluent	Fujitsu Limited
FY2021	The software performance test of HELYX on the Fugaku supercomputer	GLM Co.,Ltd.
FY2021	Proof of Concept for Industrial Use of Thermo-Fluid Simulation Application scFLOW	Fujitsu Limited

Source: https://www.hpci-office.jp/achievements/user_report (Retrieved November 18, 2025)

[Future Manufacturing] List of Social Implementation Cases (3/5)

Periods of Use	Project Name	Organization Name
FY2021	The software performance test of HELYX-Adjoint on the Fugaku supercomputer	GLM Co.,Ltd.
FY2021	High speed computation of semiconductor simulation for emerging memory	Kioxia Corporation
FY2022	Verification and Performance Evaluation of Commercial CAE Software Simcenter STAR-CCM+	SIEMENS K.K.
FY2022	Large scale multiphase fluid simulation for design of housing space including plumbing products	TOTO LTD.
FY2022	Simulation of Fan Turbulent Flow Field by Large-Scale LES Analysis and Examination of Air Conditioning and Cooling Technologies	DENSO CORPORATION
FY2022	Wind Resistant Design of Long Span Cable-Stayed Bridges by Fluid-Structure Interaction Analysis for the Full Bridge	Hanshin Expressway Company Limited
FY2022	High reliability management in advanced 5G network to meet the various requirements of different communication services	KDDI Research, Inc.
FY2022	Large-scale computational fluid simulation around ship superstructure by Fugaku	Shipbuilding Centre of Japan
FY2022	Aeroelastic analysis by CFD software, Cradle CFD	MSC Software
FY2022-2024	Proof of concept for industrial use of Thermo-Fluid Simulation Application scFLOW	Software Cradle Co., Ltd.
FY2022	Operation verification and performance evaluation of a large-scale simulation of melt plastic flow using particle method	Kobe Steel, LTD.
FY2022	Acceleration of chain assembly model analysis for utilization of Fugaku.	Tsubakimoto Chain Co.
FY2022	Practical Application of Particle-based Casting CAE Simulator	RYOBI LIMITED
FY2022	Investigation and Verification of Execution Environment at Fugaku for CAE Software Simcenter STAR-CCM+	SIEMENS K.K.

Source: https://www.hpci-office.jp/achievements/user_report (Retrieved November 18, 2025)

[Future Manufacturing] List of Social Implementation Cases (4/5)

Periods of Use	Project Name	Organization Name
FY2023	Evaluation of multi-physics and multi-scale transient response of the commercial-scale gasifier of IGCC power plant	Mitsubishi Heavy Industries, Co., Ltd.
FY2023	Performance Prediction for Centrifugal Turbomachinery by Compressible Wall-resolved LES	Hitachi, Ltd.
FY2023-2024	Study on Wind Resistant Stability of Long Span Cable-Stayed Bridges by Fluid-Structure Interaction Analysis for the Full Bridge	Hanshin Expressway Company Limited
FY2023	Mist flow simulation of numerous very small droplets for plumbing products	TOTO LTD.
FY2023	High-fidelity numerical simulation and multi-objective optimization of gasification of solid carbon resources towards innovative gasifier design	Central Research Institute of Electric Power Industry
FY2023	Applicability Assessment of VOF Model for Gas-Liquid Two-Phase Flow in a Vertical Pipe	MHI Nuclear Development Corporation
FY2023	CFD Simulation of Aircraft Stall towards Airworthiness Certification by Analysis	Kawasaki Heavy Industries, Ltd.
FY2023	Understanding the cost of casting solidification analysis using particle method	RYOBI LIMITED
FY2023	Operation verification and performance evaluation of a large-scale simulation inside plastic mixer using particle method	Kobe Steel, LTD.
FY2023	Operation Verification of OpenFOAM2.4.0 on Fugaku	MHI NS Engineering Co., Ltd.

Source: https://www.hpci-office.jp/achievements/user_report (Retrieved November 18, 2025)

[Future Manufacturing] List of Social Implementation Cases (5/5)

Periods of Use	Project Name	Organization Name
FY2024	Evaluation of transient response of the gas-liquid-solid three-phase reacting flows by a multi-physics and multi-scale simulation on the commercial-scale gasifier of IGCC power plant	Mitsubishi Heavy Industries, Co., Ltd.
FY2024	Performance Evaluation of Fluid Acoustic Analysis Application	Yamaha Corporation
FY2024	Rotating stall simulation using OpenFOAM	NIKKISO CO., LTD.
FY2024	Simulation using HBM	SUBARU CORPORATION
FY2024	Benchmark calculations of the casting simulation for automotive parts	Mazda Motor Corporation
FY2024	Verification of Simcenter STAR-CCM+ Execution on the Fugaku	Siemens K.K.
FY2024	Performance evaluation and accuracy verification of large-scale simulation in plastic mixier by particle method	Kobe Steel, LTD.
FY2024	Shape Optimization Using Adjoint Method in Aeroacoustic Analysis	Yamaha Corporation
FY2024	Proof of concept for industrial use of Thermo-Fluid Simulation Application Cradle CFD(scFLOW,scSTREAM)	Software Cradle Co., Ltd.

Source: https://www.hpci-office.jp/achievements/user_report (Retrieved November 18, 2025)

Contact Information

RIKEN Center for Computational Science
Office of the Fugaku Society 5.0 Initiative

r-ccs-s5support@ml.riken.jp