Cradle CFD

Machinery and Heavy Industry

SC/Tetra

Improving Fan Efficiency with CFD Application

Case Study for Panasonic Ecology Systems Co., Ltd.

Verifying static pressure efficiency and fan characteristics by using SC/Tetra

Utilization of CFD Software in Fan Design

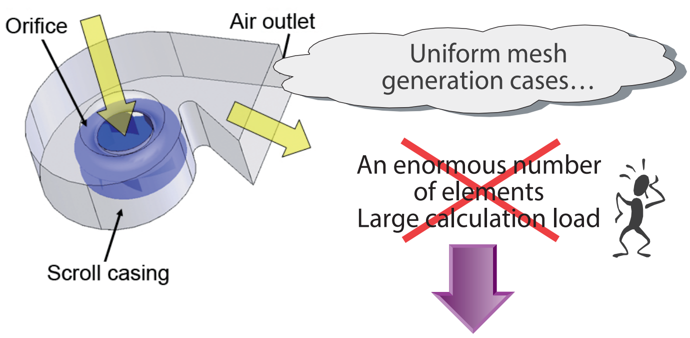
CFD software can be a highly effective tool to improve fan efficiency. In CFD simulations, engineers can see the generation of vortices by visualizing the flow field around fan blades, and estimate the efficiency of the fan from the calculation results on static pressure and torque.

Simulation model

Key point: To create a fan model with enough accuracy required for performance evaluation within a practical range of computational load.

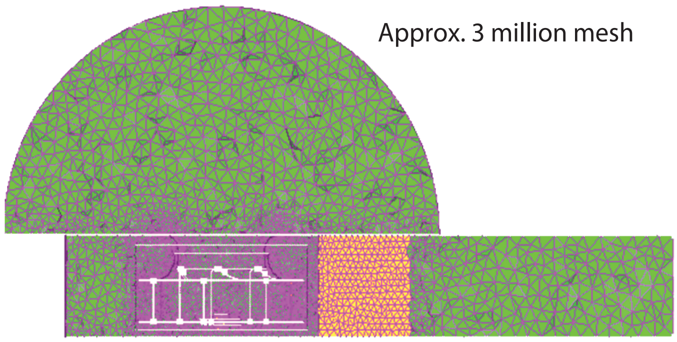
1. With a scroll casing:

A non-axisymmetric model. The entire perimeter of impellers is required for its analysis range transient analysis because performance of the fan is significantly influenced by geometry of the scroll casing and by positions of the blades.



Heavily affected regions — Fine mesh
Less affected regions — Coarse mesh

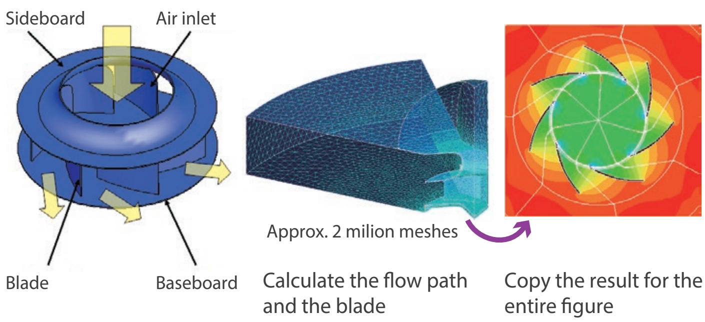
Approx. 3 million mesh



Decrease computation time & maintain accuracy

2. Without a scroll casing:

A flow path between each axisymmetric blade is assumed to be same, and the flow circulation is periodically repeated.



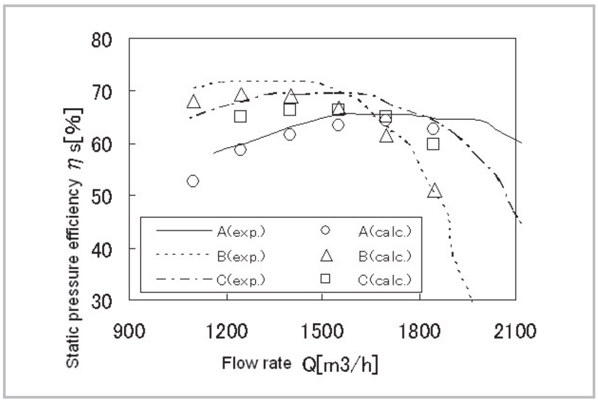
Comparison of results

[Without a casing]

Three types of centrifugal fans with different dimensions are compared.

	Model (A)	Model (B)	Model (C)
Fan diameter	229 mm	229 mm	229 mm
Blade inner diameter	142 mm	156 mm	156 mm
Blade thickness	62 mm	62 mm	82 mm
Orifice height	29.5 mm	39.5 mm	39.5 mm

Approx. 2 million mesh, k-eps model



Maximum Static Pressure Efficiency (B) > (C) > (A)

Air Capacity at Best Efficiency Point (A) > (C) > (B)

Showing simulation curves consistent with experiment

The consistency is also confirmed for the casing types

Customer Comments

A suitable CFD simulation model of the fan is used to identify effective parameters for improving fan's efficiency. Reduction in power consumption of fan has always been one of major challenges.

While high performance computing is growing fast, developers of high-efficiency fans count on CFD applications for complicated analyses.

SC/Tetra

Influence of Rotating Stall on Aerodynamic Characteristics and Noise of a Scirocco Fan

Case Study for Nagasaki University

Evaluate the influence of aerodynamic noise and rotating stall by comparing the internal flows between different geometries of scirocco fans using SC/Tetra

Improving Efficiency and Reducing Noise

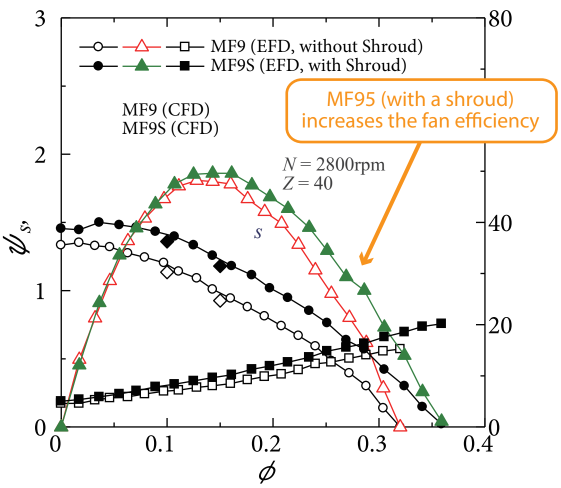
Noise reduction is a key issue in scirocco fans used for residential ventilation equipment.
A fan shroud assists in improving aerodynamic characteristics of the fan by attaching the shroud to the front side of the blade wheel even though a large part of its fluid dynamic mechanism is still unknown. The influence of a shroud on the aerodynamic characteristics and the noise is evaluated based on performance test. The influence of rotating stall on a broadband noise is discussed based on the numerical simulation of internal flow.

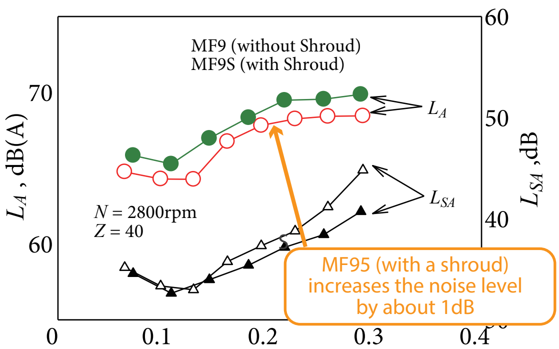
Experimental blade wheels

MF9 (without a shroud)

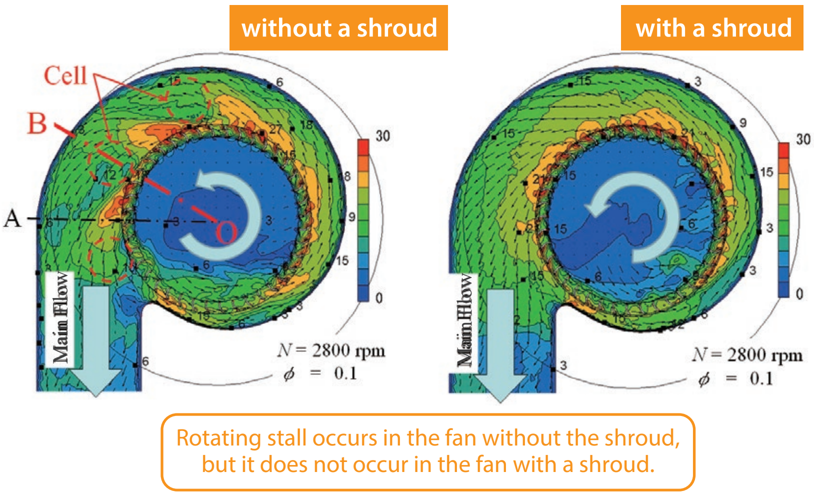
MF95 (with a shroud)

Experimental results

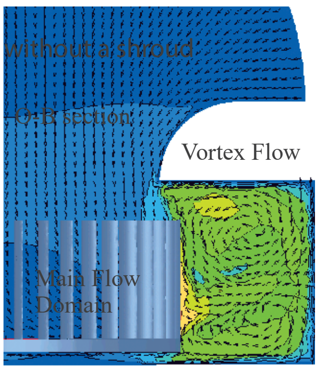




Simulation results



Influence of rotating stall on broadband noise



MF9S (with a shroud) delays rotating stall which, in turn, causes the low frequency noise to increase compared to that of MF9 near stalling speed. The blade wheel of MF9 does not fully increase the static pressure between the blades and results in a relatively low static pressure.

Customer Comments

SC/Tetra demonstrated the air flow through the fan and evaluated the influence of the rotating stall on a broadband noise of two fans. Aerodynamic characteristics and broadband noise contradict each other, but the specific noise level of MF95 is lower than that of MF9 in the area which has higher flow rate than the maximum efficiency point. It was determined with SC/Tetra that the shroud improved the overall performance of the scirocco fans.

SC/Tetra

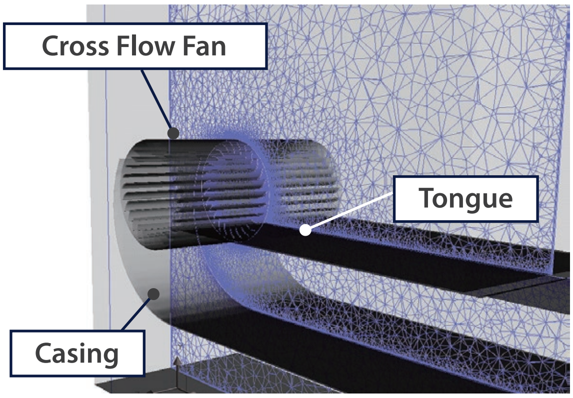
Internal Flow Mechanism of a Cross Flow Fan

Case Study for Osaka-Electro-Communication University

The internal flow of the impeller end captured by SC/Tetra

Cross Flow Fan

The internal flow of the impeller involves an eccentric vortex, and it may affect the performance of the fan and the level of the noise. Recently, as the "small size" and "low level noise" fans are in high demand, 3D calculation and experiment are performed to study the flow mechanism around the impeller.

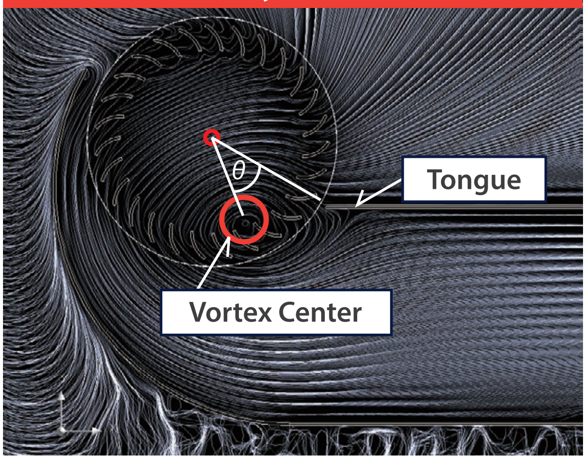


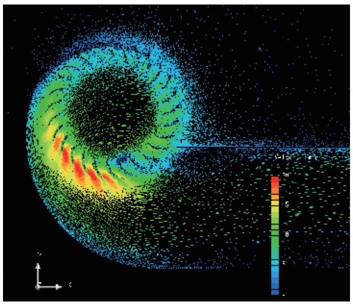
Analysis model

The impeller internal flow

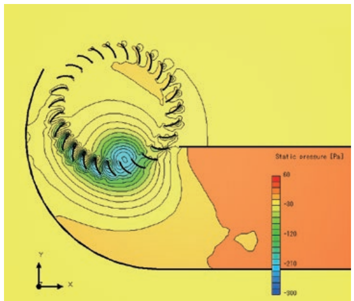
The location of eccentric vortex center is analyzed in order to clarify the influence of the casing side plate.

Result of 3D Analysis

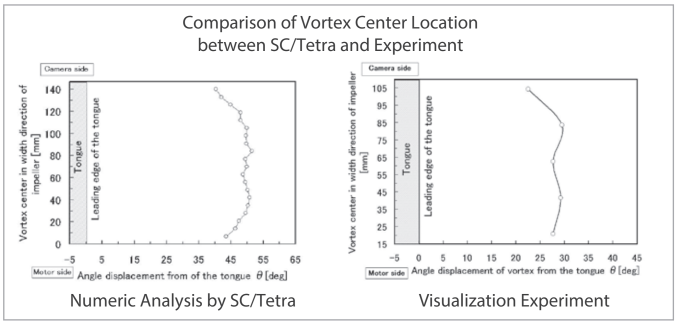




Velocity Distribution

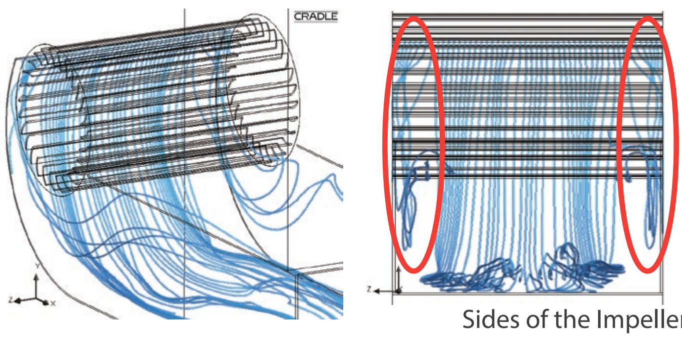


Static Pressure Distribution



The angle θ between vortex center and the tongue becomes smaller toward the edge of the impeller.

3D flow between the casing and the impeller



Sides of the Impeller

- Center of the impeller: flow along the casing
- Near the side boards: flow not along the casing

It affects the location of the vortex center inside the impeller

Customer Comments

The result of simulation of the cross flow fan captured a similar pattern to the visualized experimental result. Furthermore, it captures the flow between the center of the casing and the impeller that is difficult to be captured in physical testing.

SC/Tetra

CFD Application for Fan Performance Improvement

Case Study for Teral Inc. and Teral Kurita Inc.

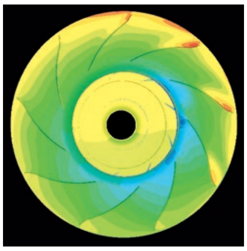
SC/Tetra for more efficient fan product development

Turbofan

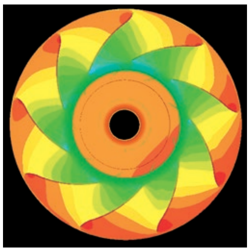
Used for:

- Drying machine
- Dust collector
- Separator

To reduce environmental burden, the company has been developing more energy efficient fans with less noise and lower vibration. They used CFD analysis and other simulation tools, such as structural analysis software, during fan development to assess and evaluate product capability.



10 m³/min air-flow
39.2% total efficiency

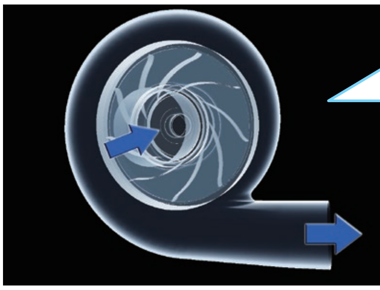


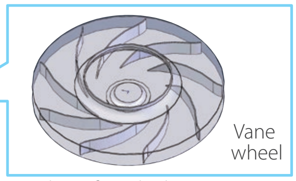
40 m³/min air-flow
78.2% total efficiency

		Analysis Results for Fan Total Efficiency							Test Results	
		1	2	3	4	5	6	7	3	4
Air flow	Q (m³/min)	0	10	20	30	40	50	60	29	45
Total pressure	P_t (Pa)	4393	4158	3981	3965	3680	3143	2306	4064	3159
Static pressure	P_s (Pa)	4393	4125	3848	3669	3141	2312	1126	3825	2580
Shaft power	L_0 (kW)	1.27	1.77	2.02	2.56	3.14	3.69	4.09	2.54	3.26
Total efficiency	η (%)	0.0	39.2	65.6	77.3	78.2	71.0	56.4	76.8	72.4

Analysis objective and model

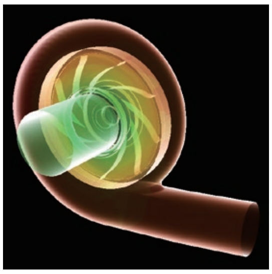
The analysis investigated fan performance and identified the optimal fan models by visualizing the fan's inner flow and pressure distribution and evaluating velocity, pressure and efficiency.





Number of Mesh Elements:
Approx. 7.8 million mesh elements
Analysis Condition:
- Steady-state analysis
- ALE (rotating boundary), rotating/stationary regions

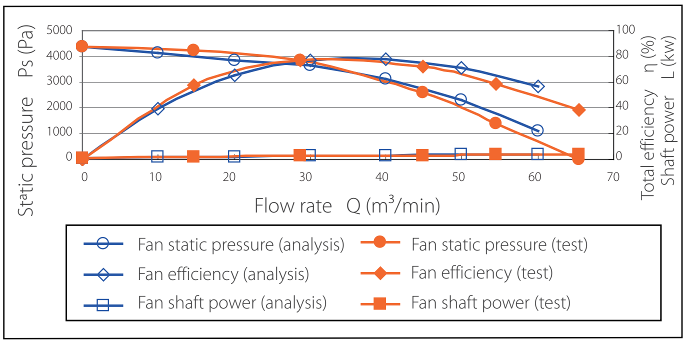
Comparison between analysis and test results



*power source frequency is 60 Hz

Visualizing pressure distribution

Comparison between flow rate of 10 m³/min and 40 m³/min



As indicated in the graph above, the analysis and experimental results are in excellent agreement.

Customer Comments

Applying SC/Tetra enabled us to evaluate fan performance and study optimal modeling at the same time. It also helped us to examine performance at various stages in the design and development phases by visualizing the movement of a vane wheel, inner flow within the fan casing, and pressure distribution. We achieved 86.5% total efficiency (at 50Hz) by applying a newly designed vane wheel, demonstrating the significant role SC/Tetra played in the design of the optimal fan that keeps noise at a minimum. Overall, we have successfully reduced the number of prototypes and production tests by applying fluid analysis. This has contributed to reducing cost and development time.

SC/Tetra

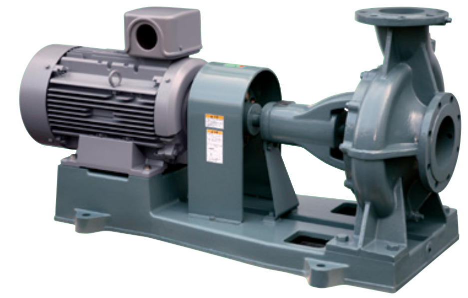
Centrifugal Pump Development

Case Study for Teral Inc.

SC/Tetra contributes to more efficient development process

Centrifugal Pump

Fluid analysis combined with structural analysis, 3D CAD, 3D measurement and 3D printer lead to the development of high-value-added products.



For Hot & Cold Water Circulation

For General Supply

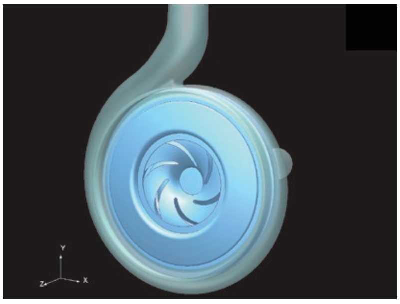
For Supplying Industrial & Sewage Water

•Advanced Research, Designing & Development

•Responding to Sophisticated Needs

•Maintaining Quality Products

Simulation model



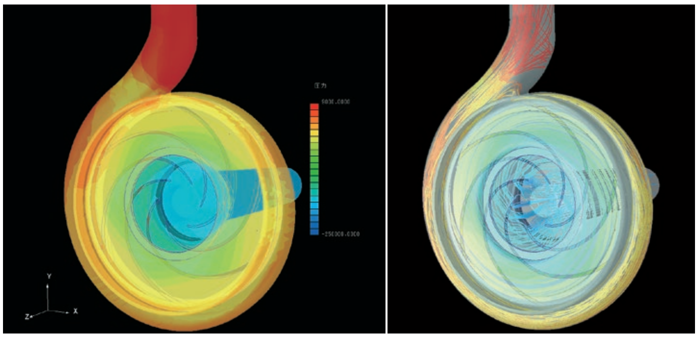
Approx. 7.74 million elements

Approx. 2.2 million nodes

Steady-state analysis

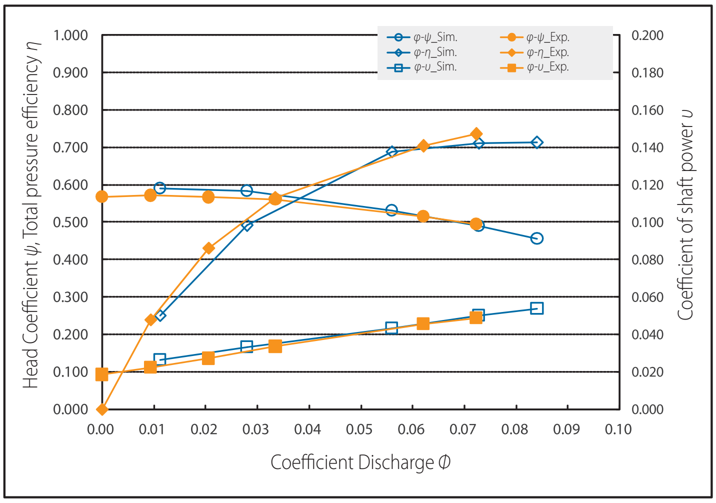
ALE for rotating and stationary regions

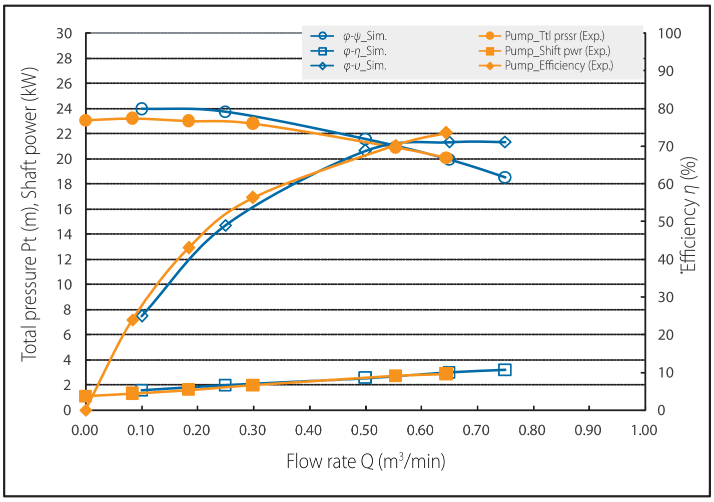
Simulation results



Flow patterns and pressure distribution can be examined visually for better understanding. Also, comparison of simulation results can assist in determining better designs before prototyping.

Result comparisons





Showing close agreements between simulation results & experimental results

Customer Comments

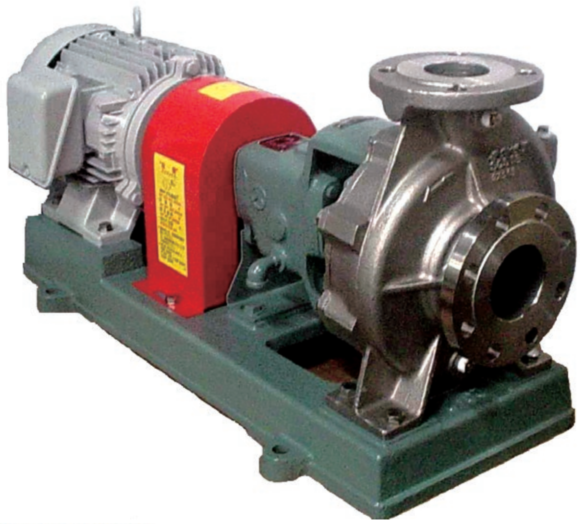
SC/Tetra accelerates the pump development processes in areas such as performance evaluation and visualization of internal flow. This results in a reduction in development time (25-65 %) and cost. In addition, engineers can be actively involved in the evaluation of the simulation results regardless of experience or skill level.

SC/Tetra

Analysis of Centrifugal Pumps

Case Study for MALHATY PUMP MFG. CO., LTD.

Application of SC/Tetra to Simulate and Validate the Performance of a Centrifugal Pump



MALHATY PUMP

MALHATY PUMP MFG. CO., LTD.

Pictured: FIC type pump

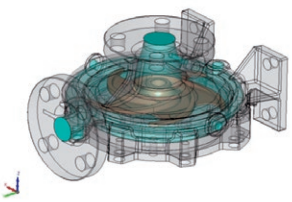
Uses of Centrifugal Pumps

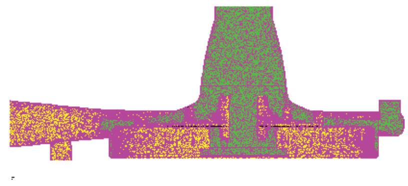
- General plant use
- Effluent treatment plants
- Environmental pollution control systems
- Water supply and sewage treatment systems
- Solvent transfer and handling of unique fluids

Characteristics

- Main rotor and impeller can be inspected without removing pipes because the pump is integrated with the base back pullout section
- Meets Japan Society of Industrial Machinery Manufacturers Standards (JIMS) for 16-bar-class products

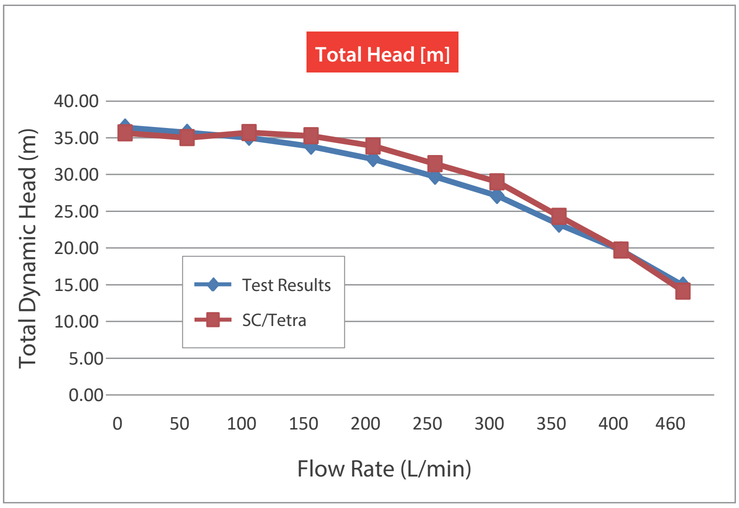
Analysis model





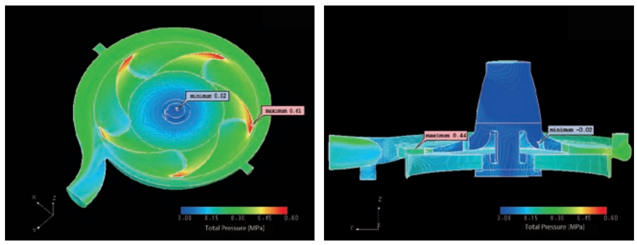
Number of mesh elements: 2,534,983

Comparison between analysis and test results*

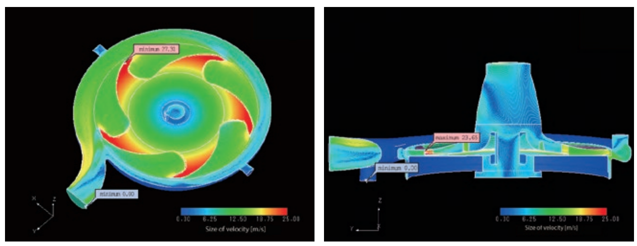


* SC/Tetra was used to conduct the analysis above

Analysis results



Total pressure distribution (400 L/min)



Velocity distribution (400 L/min)

Customer Comments

It is difficult to experimentally evaluate the complex flow phenomena occurring inside a high-speed rotating pump. This is where CFD analysis can be very effective for providing insight about the flow's behavior inside the pump. The pump's calculated pressure versus flow curve agrees well with the experimental values although some values were estimated slightly larger. This has helped us understand the overall tendency of the pump performance.

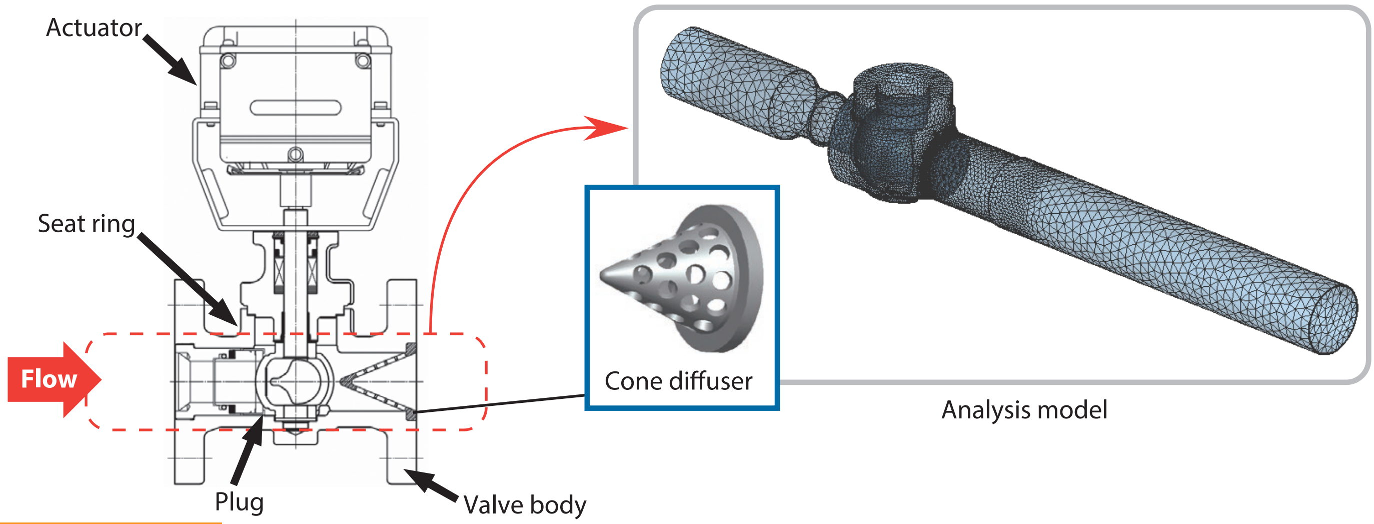
SC/Tetra

Development of a Cavitation Resistant Rotary Control Valve

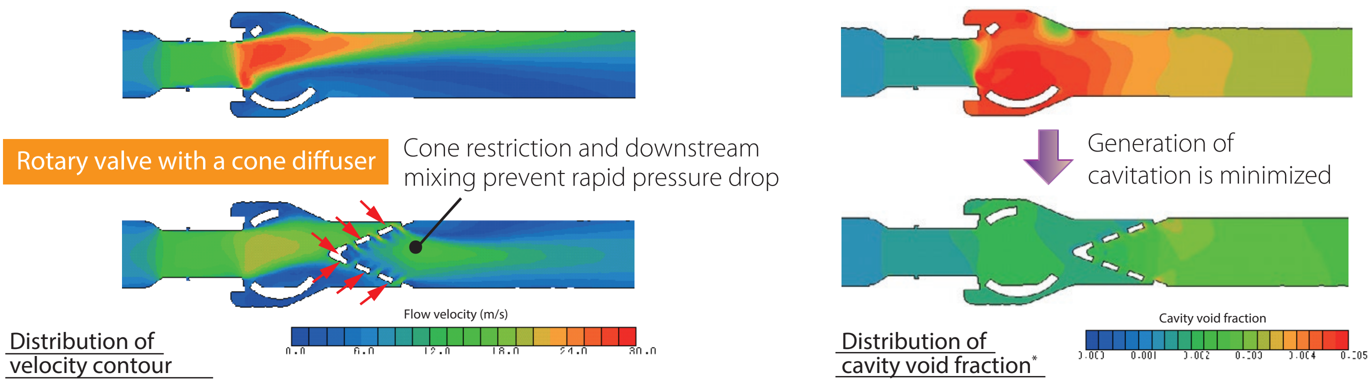
Case Study for Azbil Corporation

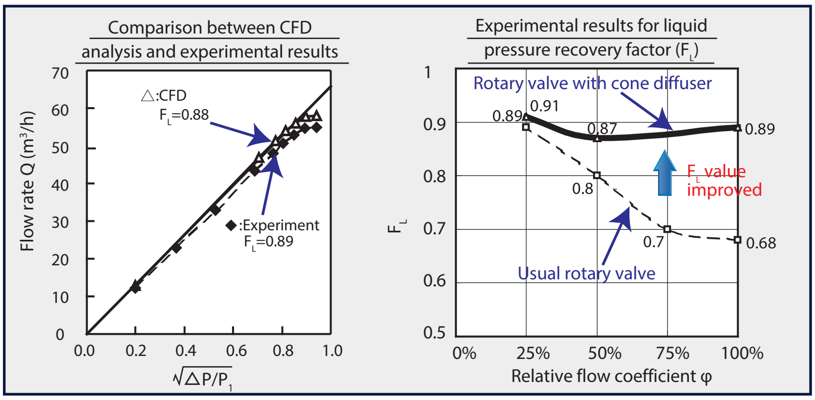
SC/Tetra was used to develop a rotary control valve that has an improved cavitation resistance

Predicting the Effects of Cone Diffuser by Visualizing the Flow inside Rotary Valve



Usual rotary valve ⇨ Valve damage is caused by the cavitation generated due to the rapid pressure drop





*Void fraction is the ratio of vapor volume.

The cone diffuser reduced the cavitation risk and minimized the overall size and weight

Customer Comments

Using the CFD cavitation model enabled us to achieve the following;

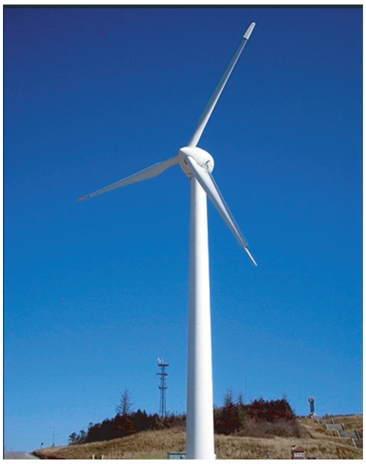
- Improving physical understanding and revalidating the effects by visualizing the flow velocity and void fraction
- Optimizing valve design to minimize cavitation, balance flow capacity, and maximize liquid pressure recovery factor (F_L)

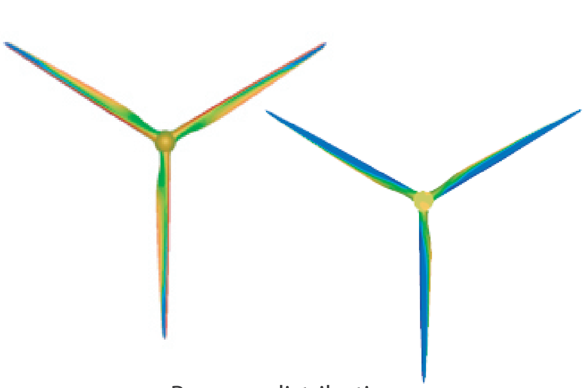
SC/Tetra

Renewable Energy and Simulation

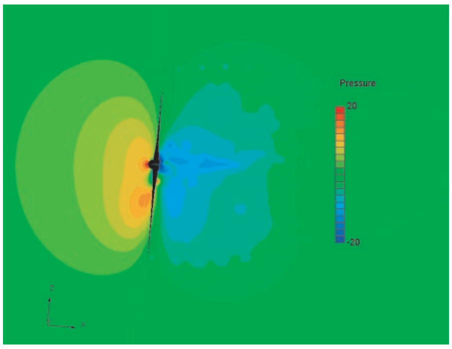
Power Generation

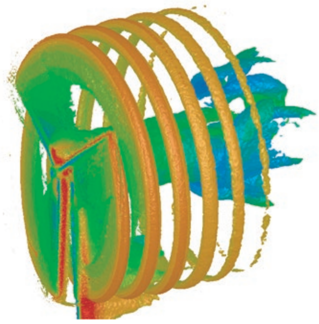
SC/Tetra





Pressure distribution
(Left: Front & back view / Right: Side view)





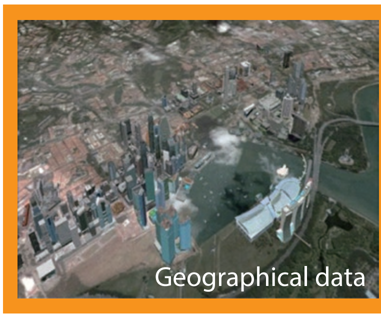
Isosurface of vorticity

Power generation

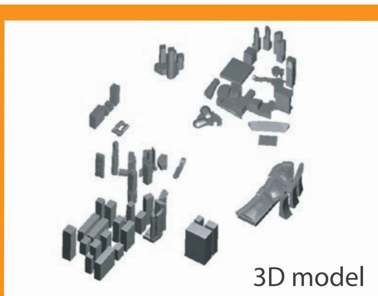
Renewable energy is attracting attention as next-generation energy sources, and wind power generation is one of them. CFD simulation can be used to predict wind environment around turbines, as well as pressure force and torque acting on turbine blades.

Wind Environment and Urban Planning

scSTREAM

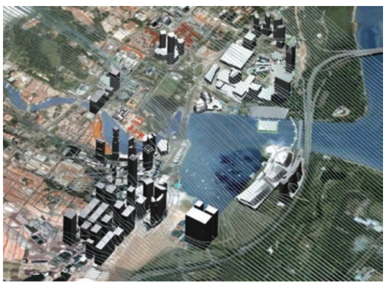


Geographical data

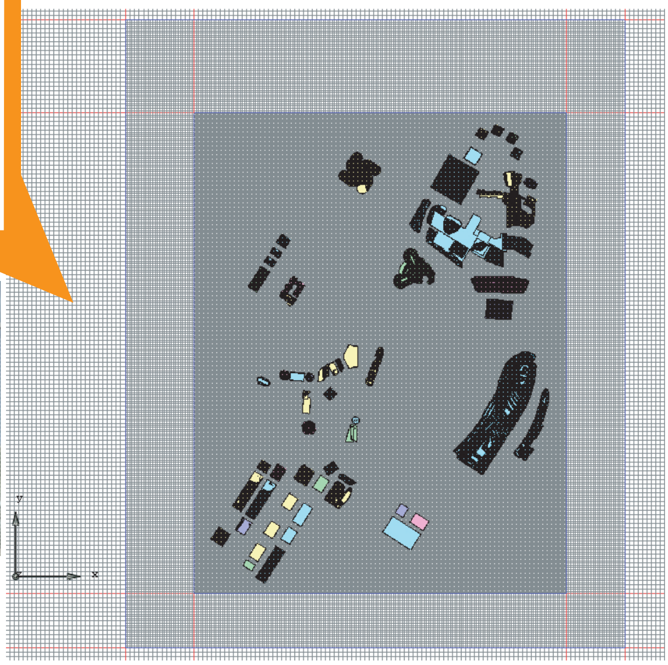


3D model

Velocity around buildings



Wind flow around buildings



Mesh of scSTREAM

Wind environment

Wind environment assessment is becoming highly important in urban planning and architectural design due to the increase in high-rise buildings in a congested area, making the wind flow very complicated. CFD simulation is very useful for predicting the wind effect and for taking measures against strong wind blowing through buildings. Even a wide area can be easily simulated using electronic geographical data.

Contact us here

Hexagon is a global leader in digital reality solutions, combining sensor, software and autonomous technologies. We are putting data to work to boost efficiency, productivity, quality and safety across industrial, manufacturing, infrastructure, public sector, and mobility applications.

Our technologies are shaping production and people-related ecosystems to become increasingly connected and autonomous – ensuring a scalable, sustainable future.

Hexagon's Manufacturing Intelligence division provides solutions that use data from design and engineering, production and metrology to make manufacturing smarter. For more information, visit hexagonmi.com.

Learn more about Hexagon (Nasdaq Stockholm: HEXA B) at hexagon.com and follow us @HexagonAB.