

Page	Title	Author	Abstract	Keywords
1-9	Computational Investigation of Turbulent Swirling Flows in Gas Turbine Combustors	A. C. Benim, M. P. Escudier, P. J. Stopford, E. Buchanan, K. J. Syed	In the first part of the paper, Computational Fluid Dynamics analysis of the combustor flow within a high-swirl lean premixed gas turbine combustor and over the 1st row nozzle guide vanes is presented. In this analysis, the focus of the investigation is the fluid dynamics at the combustor/turbine interface and its impact on the turbine. The predictions show the existence of a highly-rotating vortex core in the combustor, which is in strong interaction with the turbine nozzle guide vanes. This has been observed to be in agreement with the temperature indicated by thermal paint observations. The results suggest that swirling flow vortex core transition phenomena play a very important role in gas turbine combustors with modern lean-premixed dry low emissions technology. As the predictability of vortex core transition phenomena has not yet been investigated sufficiently, a fundamental validation study has been initiated, with the aim of validating the predictive capability of currently-available modelling procedures for turbulent swirling flows near the sub/supercritical vortex core transition. In the second part of the paper, results are presented which analyse such transitional turbulent swirling flows in two different laboratory water test rigs. It has been observed that turbulent swirling flows of interest are dominated by low-frequency transient motion of coherent structures, which cannot be adequately simulated within the framework of steady-state RANS turbulence modelling approaches. It has been found that useful results can be obtained only by modelling strategies which resolve the three-dimensional, transient motion of coherent structures, and do not assume a scalar turbulent viscosity at all scales. These models include RSM based URANS procedures as well as LES and DES approaches.	Turbulent swirling flows, gas turbine combustors, URANS, RSM, LES, DES
10-23	Flows over Concave Surfaces: Development of Pre-set Wavelength Görtler Vortices	S. H. Winoto, Tandiono, D. A. Shah, H. Mitsudharmadi	The development of pre-set wavelength Görtler vortices are studied in the boundary-layer flows on concave surfaces of 1.0 and 2.0 m radius of curvature. The wavelengths of the vortices were pre-set by thin wires of 0.2 mm diameter placed 10 mm upstream and perpendicular to the concave surface leading edge. Velocity contours were obtained from velocity measurements using a single hot-wire anemometer probe. The most amplified or dominant wavelength is found to be 15 mm for free-stream velocity of 2.1 m/s and 3.0 m/s on the concave surface of R = 1 m and 2 m, respectively. The velocity contours in the cross-sectional planes at several streamwise locations show the growth and breakdown of the vortices. Three different regions can be identified based on the growth rate of the vortices. The occurrence of a secondary instability mode is also shown in the form of mushroom-like structures as a consequence of the non-linear growth of the Görtler vortices. By pre-setting the vortex wavelength to be much larger and much smaller than the most amplified one, the splitting and merging of Görtler vortices can be respectively observed.	Görtler vortices, pre-set wavelength, concave surface, boundary-layer flow
24-32	Control of Shock-Wave/Bound-Layer Interactions by Bleed	T. I-P. Shih	Bleeding away a part of the boundary layer next to the wall is an effective method for controlling boundary-layer distortions from incident shock waves or curvature in geometry. When the boundary-layer flow is supersonic, the physics of bleeding with and without an incident shock wave is more complicated than just the removal of lower momentum fluid next to the wall. This paper reviews CFD studies of shock-wave/boundary-layer interactions on a flat plate with bleed into a plenum through a single hole, three holes in tandem, and four rows of staggered holes in which the simulation resolves not just the flow above the plate, but also the flow through each bleed hole and the plenum. The focus is on understanding the nature of the bleed process.	shock-wave/boundary-layer interaction, inlet bleed
33-37	The Development of Protocols for Equitable Testing and Evaluation in Ocean Energy - A Three-Year Strategy	David M Ingram, Jose Luis Villate, Cyrille Abonnel, Cameron Johnstone	EquiMar (Equitable Testing and Evaluation of Marine Energy Extraction Devices in terms of Performance, Cost and Environmental Impact) is one of the first round of energy projects under the European Commissions 7th Framework Programme (FP7). The three year EquiMar project aims to deliver a suite of protocols for the evaluation of both wave and tidal converters, harmonizing testing and evaluation procedures across the wide range of available devices, accelerating adoption through technology matching and improving the understanding of both environmental and economic impacts associated with the deployment of devices. The EquiMar protocols will cover site selection, initial design, scaling up of designs, the deployment of arrays and environmental impact assessment as well as economic issues. EquiMar will build on existing protocols, e.g. UK DTI Marine Renewables Development Fund (MRDF) protocols for wave and tidal energy, and engage with international standards setting activities, e.g. IEC TC114.	Ocean Energy Conversion, Renewable Energy, Protocol Development
38-46	Cavitation Instabilities of Hydrofoils and Cascades	Yoshinobu Tsujimoto, Satoshi Watanabe, Hironori Horiguchi	Studies on cavitation instabilities of hydrofoils and cascades are reviewed to obtain fundamental understandings of the instabilities observed in turbopump inducers. Most of them are based on the stability analysis of two-dimensional inviscid cavitating flow. The most important finding of the analysis is that the cavitation instabilities depend only on the mean cavity length. For a hydrofoil, the characteristic length is the chord length and partial/transitional cavity oscillation occurs with shorter/longer cavity than 75% of the chord length. For cascades, the characteristic length is the blade spacing and various modes of instabilities are predicted when the mean cavity is longer than 65% of the spacing. In the last part, rotating choke is shown to occur when the cavity becomes longer than the spacing.	Cavitation Instability, Hydrofoil, Cascade, Inducer, Rotating Cavitation, Cavitation Surge
47-56	Study on Flow Fields in Variable Area Nozzles for Radial Turbines	Hideaki Tamaki, Masaru Unno	The flow behind the variable area nozzle which corresponds to the flow at the leading edge of the impeller was measured with a 3-hole yaw probe and calculated with CFD. Two nozzle throat-areas were investigated. One is the smallest and the other is the largest opening for the variable nozzle. Test results agreed with the calculated results qualitatively. The leakage flow through the tip clearance of the nozzle vane significantly affected the flow field downstream of the nozzle vane with the smallest opening. However, the effect on leakage flow on the flow field downstream of the nozzle vane with the largest opening was very weak and the effect of wake is dominant.	Radial turbine, Nozzle vane, Variable geometry turbine, Clearance flow
57-63	Study on Flow Fields in Variable Area Nozzles for Radial Turbines	Masafumi Miyano, Toshiaki Kanemoto, Daisuke Kawashima, Akihiro Wada, Takashi Hara, Kazuyuki Sakoda	To optimize the stationary components in the multistage centrifugal pump, the effects of the return vane profile on the performances of the multistage centrifugal pump were investigated experimentally, taking account of the inlet flow conditions for the next stage impeller. The return vane, whose trailing edge is set at the outer wall position of the annular channel downstream of the vane and which discharges the swirl-less flow, gives better pump performances. By equipping such return vane with the swirl stop set from the trailing edge to the main shaft position, the unstable head characteristics can be also suppressed successfully at the lower discharge. Taking the pump performances and the flow conditions into account, the impeller blade was modified so as to get the shock-free condition where the incidence angle is zero at the inlet.	Return Vane Installed in Multistage Centrifugal Pump
64-75	Study on Flow Fields in Variable Area Nozzles for Radial Turbines	Toshifumi Watanabe, Donghyuk Kang, Angelo Cervone, Yutaka Kawata, Yoshinobu Tsujimoto	During an experimental investigation on a 3-bladed and a 4-bladed axial inducer, a severe surge instability was observed in a range of cavitation number where the blade passage is choked and the inducer head is decreased from non-cavitating value. The surge was stronger for the 4-bladed inducer as compared with a 3-bladed inducer with the same inlet and outlet blade angles. For the 4-bladed inducer, the head decreases suddenly as the cavitation number is decreased. The surge was observed after the sudden drop of head. This head drop was found to be associated with a rapid extension of tip cavity into the blade passage. The cause of surge is attributed to the decrease of the negative slope of the head-flow rate performance curve due to choke. Assuming that the difference between the 3 and 4-bladed inducers is caused by the difference of the blockage effects of the blade, a test was carried out by thickening the blades of the 3-bladed inducer. However, opposite to the expectations, the head drop became smoother and the instability disappeared on the thickened blade inducer.	Cavitation, instabilities, inducers, surge

2008					
76-85	Flow Analysis in Positive Displacement Micro-Hydro Turbine and Development of Low Pulsation Turbine	Junichi Kurokawa, Jun Matsui, Young-Do Choi	In order to extract micro hydropower in the very low specific speed range, a Positive Displacement Turbine (PDT) was proposed and steady performance was determined experimentally. However, the suppression of large pressure pulsation is inevitable for practical application of PDT. The objective of the present study is to reveal the mechanism and the characteristics of pressure pulsation in PDT by use of CFD and to suppress the pressure pulsation. Unsteady CFD analysis has revealed that large pressure pulsation is caused by large variation of rotational speed of the following rotor, while the driving rotor, which is output rotor, keeps constant speed. Here is newly proposed a 4-lobe helical type rotor which can reduce the pressure pulsation drastically and the performance prediction of new PDT is determined.	Micro hydropower, Positive displacement turbine, Helical rotor, Pressure pulsation, Performance improvement	
86-91	Matching Diffuser Vane with Return Vane Installed in Multistage Centrifugal Pump	Daisuke Kawashima, Toshiaki Kanemoto, Kazuyuki Sakoda, Akihiro Wada, Takashi Hara	The effects of the diffuser vane on the performances of the multistage centrifugal pump were investigated experimentally, taking account of the interactions among the diffuser vane, the return vane, and the next stage impeller. It is very important to match well the diffuser vane with the return vane, for improving the hydraulic efficiency of the pump. The efficiency may be more improved by making the cross-sectional area of the channel from the diffuser vane outlet to the return vane inlet larger, as much as possible.	Centrifugal pump, diffuser vane, return vane, multistage pump, performance, efficiency, hydraulic loss	
92-100	Unsteady Flows Arising in a Mixed-Flow Vaneless Diffuser System	Hiromu Tsurusaki	The main objective of this study was to clarify the origin of the unsteady flows arising in a mixed-flow vaneless diffuser system and also the effects of physical components of the system. The testing equipment consists of a straight tube, a swirl generator, and a mixed-flow vaneless diffuser. Pressure fluctuations of the flow through the tube and diffuser were measured by using a semiconductor-type pressure transducer and analyzed by an FFT analyzer. In the experiment, the velocity ratio (axial velocity/peripheral velocity) of the internal flow, and the geometric parameters of the diffuser were varied. Two kinds of unsteady flows were measured according to the combination of the components, and the origin of each unsteady flow was clarified. The fundamental frequencies of unsteady flows arose were examined by two-dimensional small perturbation analysis.	Unsteady flow, Mixed-flow, Vaneless diffuser, Swirling flow, Pressure fluctuation, Rotating stall	
101-108	Reynolds Number Effect on Regenerative Pump Performance in Low Reynolds Number Range	Hironori Horiguchi, Daisuke Yumiba, Yoshinobu Tsujimoto, Masaaki Sakagami, Shigeo Tanaka	The effect of Reynolds number on the performance of a regenerative pump was examined in a low Reynolds number range in experiment. The head of the regenerative pump increased at low flow rates and decreased at high flow rates as the Reynolds number decreased. The computation of the internal flow was made to clarify the cause of the Reynolds number effect. At low flow rates, the head is decreased with increasing the Reynolds number due to the decrease of the shear force exerted by the impeller caused by the increase of leakage and hence local flow rate. At higher flow rates, the head is increased with increasing the Reynolds number with decreased loss at the inlet and outlet as well as the decreased shear stress on the casing wall.	Regenerative Pump, Pump Performance, Reynolds Number Effect, Low Reynolds number	
109-120	Control of Pump Performance with Attaching Flaps on Blade Trailing Edges	Yuji Kanemori, Ying Kang Pan	An innovative method of changing a centrifugal low specific speed pump performance and pressure fluctuation by applying outlet flaps to impeller exit has been investigated. The outlet blade edge section corresponds to the trailing edge of wing on the circular-cascade, which dominates the pump performance and pressure fluctuation. Computational fluid dynamics (CFD) analysis of the entire impeller and volute casing and an experimental investigation are conducted. The pressure fluctuation and the vibration of the shaft are measured simultaneously. Kurtosis is applied as a dimensionless parameter with which the unevenness of velocity distribution at impeller outlet is indicated. The influence of the flaps on the pressure fluctuation is explained by the kurtosis. This paper presents a theoretical method of predicting the pump performance related to the attachment of a flap at impeller outlet.	Pump, Flap, Performance, Pressure fluctuation, Vibration	
121-139	Investigation of Corrosion Fatigue Phenomena in Transient Zone and Preventive Coating and Blade Design against Fouling and Corrosive Environment for Mechanical Drive Turbines	Satoshi Hata, Naoyuki Nagai, Toyooki Yasui, Hiroshi Tsukamoto	For mechanical drive steam turbines, the investigation results of corrosion fatigue phenomena in the transient zone are introduced, including basic phenomena on expansion line and actual design and damage experience. These results were analyzed from the standpoint of stress intensity during the start of cracking. In order to resolve such problems, preventive coating and blade design methods against fouling and corrosive environments are developed. Detailed evaluation test results are given for coating performance using a unique test procedure simulating fouling phenomena and washing conditions. Finally, the results of the successful modification of internals and on-line washing results on site are introduced.	Steam Turbine, Corrosion Fatigue, Wilson Zone, Thermodynamics, Numerical Analysis, ISB, Surface Treatment, Hybrid Coating, Deposits, Corrosion, Erosion, On-line wash	
140-147	Experimental Study of Check Valves in Pumping Systems with Air Entrainment	Thong See Lee, Hong Tong Low, Dinh Tam Nguyen, Wei Rong, Avan Neo	An experiment setup was introduced to study dynamic behaviour of different types of check valves and the effects of air entrainment on the check valve performance under pressure transient condition. The experiment results show that the check valves with low inertia, assisted by springs or small traveling distance/angle gave better performance under pressure transient condition than check valves without these features. Air entrainment was found to affect both wave speed and reverse velocity. With the increase of the initial air void fraction in pipeline, the experiment results show that the wave speed was reduced, the reverse velocity was increased. The first peak pressure increased initially and then decreased with the increase of the initial air void fraction, the pressure surge periods were increased proportionally with air void fraction due to the greatly reduced wave speed. The study can be applied to help choosing suitable check valves for a particular pumping system.	air entrainment, pressure surges, pumping system, check valve	
148-154	Influence of Blade Profiles on Flow around Wells Turbine	Masami Suzuki, Chuichi Arakawa	The Wells turbine rotor consists of several symmetric airfoil blades arranged around a central hub, and the stagger angle is 90 degrees. These characteristics simplify the total construction of OWC type wave energy converters. Although the Wells turbine is simple, the turbine produces a complicated flow field due to the peculiar arrangement of blades, which can rotate in the same direction irrespective of the oscillating airflow. In order to understand these flows, flow visualization is carried out with an oil-film method in the water tunnel. This research aims to analyze the mechanism of the 3-D flows around the turbine with the flow visualization. The flow visualization explained the influence of attack angle, the difference between fan-shaped and rectangular wings, and the sweep angle.	Wells Turbine, Flow Visualization, Oil Film Method, Wave Power Generation, Ocean Engineering	
155-168	Variable Geometry Mixed Flow Turbine for Turbochargers: An Experimental Study	Srithar Rajoo, Ricardo Martinez-Botas	This paper investigates a variable geometry (VG) mixed flow turbine with a novel, purposely designed pivoting nozzle vane ring. The nozzle vane ring was matched to the 3-dimensional aspect of the mixed flow rotor leading edge with lean stacking. It was found that for a nozzle vane ring in a volute, the vane surface pressure is highly affected by the flow in the volute rather than the adjacent vane surface interactions, especially at closer nozzle positions. The performance of the VG mixed flow turbine has been evaluated experimentally in steady and unsteady flow conditions. The VG mixed flow turbine shows higher peak efficiency and swallowing capacity at various vane angle settings compared to an equivalent nozzleless turbine. Comparison with an equivalent straight vane arrangement shows a higher swallowing capacity but similar efficiencies. The VG turbine unsteady performance was found to deviate substantially from the quasi-steady assumption compared to a nozzleless turbine. This is more evident in the higher vane angle settings (smaller nozzle passage), where there are high possibility of choking during a pulse cycle. The presented steady and unsteady results are expected to be beneficial in the design of variable geometry turbochargers, especially the ones with a mixed flow turbine.	variable geometry turbocharger, mixed flow turbine, steady flow and pulsating flow	
169-180	Resonance and Instability of Blade-Shaft Coupled Bending Vibrations with In-plane Blade Vibration	Norihisa Anegawa, Hiroyuki Fujiwara, Akira Okabe, Osami Matsushita	As a major component of a power plant, a turbine generator must have sufficient reliability. Longer blades have lower natural frequency, thereby requiring that the design of the shaft and blade takes into account the coupling of the blade vibration mode, nodal diameter $k=0$ and $k=1$ with vibration of the shaft.	Blade-Shaft coupled vibration, Bending vibration, Resonance, Instability, In-plane vibration, Turbine generator	
	Page	Title	Author	Abstract	Keywords

1-12	Application of Surrogate Modeling to Design of A Compressor Blade to Optimize Stacking and Thickness	Abdus Samad, Kwang-Yong Kim	Surrogate modeling is applied to a compressor blade shape optimization to modify its stacking line and thickness to enhance adiabatic efficiency and total pressure ratio. Six design variables are defined by parametric curves and three objectives; efficiency, total pressure and a combined objective of efficiency and total pressure are considered to enhance the performance of compressor blade. Latin hypercube sampling of design of experiments is used to generate 55 designs within design space constituted by the lower and upper limits of variables. Optimum designs are found by formulating a PRESS (predicted error sum of squares) based averaging (PBA) surrogate model with the help of a gradient based optimization algorithm. The optimum designs using the current variables show that, to optimize the performance of turbomachinery blade, the adiabatic efficiency objective is improved substantially while total pressure ratio objective is increased a very small amount. The multi-objective optimization shows that the efficiency can be increased with the less compensation of total pressure reduction or both objectives can be increased simultaneously.	Compressor Blade, Optimization, Surrogate Modeling, Stacking Line, Thickness of Blade, Efficiency
13-20	Analysis of Aerodynamic Performance in an Annular Compressor Bowed Cascade with Large Camber Angles	Shaowen Chen, Fu Chen	The effects of positively bowed blade on the aerodynamic performance of annular compressor cascades with large camber angle were experimentally investigated under different incidences. The distributions of the exit total pressure loss and secondary flow vectors of compressor cascades were analyzed. The static pressure was measured by tapping on the cascade surfaces, and the ink-trace flow visualizations were conducted. The results show that the value of the optimum bowed angle and optimum bowed height decrease because of the increased losses at the mid-span with the increase of the camber angle. The C-shape static pressure distribution along the radial direction exists on the suction surface of the straight cascade with large r camber angles. When bowed blade is applied, the larger bowed angle and larger bowed height will further enhance the accumulation of the low-energy fluid at the mid-span, thus deteriorate the flow behavior. Under 60° camber angle, flow behavior near the end-wall region of some bowed cascades even deteriorates instead of improving because the blockage of the separated flow near the mid-span keeps the low-energy fluid near the end-walls from moving towards the mid-span region, and as a result, a rapid augmentation of the total loss is easy to take place under large bowed angle. With the increase of camber angle, the choice range of bowed angle corresponding to the best performance in different incidences become narrower.	Cascade experiment, compressor, camber angle
21-30	Prediction of Wear Depth Distribution by Slurry on a Pump Impeller	Kenichi Sugiyama, Hiroshi Nagasaka, Takeshi Enomoto, Shuji Hattori	Slurry wear with sand particles in rivers is a serious problem for pump operation. Therefore, a technique to predict wear volume loss is required for selecting wear resistant materials and determining specifications for the maintenance period. This paper reports a method for predicting the wear depth distribution on the blade of an impeller. Slurry wear tests of an aluminum pump impeller were conducted. Prediction results of wear depth distribution approximately correspond with the results of slurry wear tests. This technique is useful for industrial application.	Pump, Impeller, Numerical analysis, Slurry wear, Life prediction
31-39	A Behavior of the Diffuser Rotating Stall in a Low Specific Speed Mixed-Flow Pump	Masahiro Miyabe, Akinori Furukawa, Hideaki Maeda, Isamu Umeki, Yoshinori Jittani	The flow instability in a low specific speed mixed-flow pump, having a positive slope of head-flow characteristics was investigated. Based on the static pressure measurements, it was found that a rotating stall in the vaned diffuser occurs at about 65% flow rate of best efficiency point (BEP). A dynamic Particle Image Velocimetry (PIV) measurement and the numerical simulations were conducted in order to investigate the flow fields. As a result, the diffuser rotating stall was simulated even by Computational Fluid Dynamics (CFD) and the calculated periodic flow patterns agree well with the measured ones by PIV. It is clarified that a periodical large scaled backflow, generated at the leading edge of the suction surface of the diffuser vane, causes the instability. Furthermore, the growth of the strong vortex at the leading edge of the diffuser vane induces the strong backflow from the diffuser outlet to the inlet. The scale of one stall cell is covered over four-passages in total thirteen vane-passages.	Mixed flow pump, Flow instability, Vaned diffuser, Rotating stall, PIV, CFD
40-54	A Backflow Vortex Cavitation and Its Effects on Cavitation Instabilities	Kazuyoshi Yamamoto, Yoshinobu Tsujimoto	Cavitation instabilities in turbo-machinery such as cavitation surge and rotating cavitation are usually explained by the quasi-steady characteristics of cavitation, mass flow gain factor and cavitation compliance. However, there are certain cases when it is required to take account of unsteady characteristics. As an example of such cases, cavitation surge in industrial centrifugal pump caused by backflow vortex cavitation is presented and the importance of the phase delay of backflow vortex cavitation is clarified. First, fundamental characteristics of backflow vortex structure is shown followed by detailed discussions on the energy transfer under cavitation surge in the centrifugal pump. Then, the dynamics of backflow is discussed to explain a large phase lag observed in the experiments with the centrifugal pump.	Cavitation, Instability, Backflow, Cavitation Surge, Inducer, Centrifugal pump, Vortex
55-60	A Thermal Analysis of a Film Cooling System with Normal Injection Holes Using Experimental Data	Kyung Min Kim, Dong Hyun Lee, Hyung Hee Cho, Moon Young Kim	The present study investigated temperature and thermal stress distributions in a film cooling system with normal injection cooling flow. 3D-numerical simulations using the FEM commercial code ANSYS were conducted to calculate distributions of temperature and thermal stresses. In the simulations, the surface boundary conditions used the surface heat transfer coefficients and adiabatic wall temperature which were converted from the Sherwood numbers and impermeable wall effectiveness obtained from previous mass transfer experiments. As a result, the temperature gradients, in contrast to the adiabatic wall temperature, were generated by conduction between the hot and cold regions in the film cooling system. The gradient magnitudes were about 10°-20K in the y-axis (spanwise) direction and about 50°60K in the x-axis (streamwise) direction. The high thermal stresses resulting from this temperature distribution appeared in the side regions of holes. These locations were similar to those of thermal cracks in actual gas turbines. Thus, this thermal analysis can apply to a thermal design of film cooling holes to prevent or reduce thermal stresses.	Gas turbine heat transfer, Film cooling, Thermal analysis, Thermal stress, Finite element method
61-71	Machine Fault Diagnosis and Prognosis: The State of The Art	Tran Van Tung, Bo-Suk Yang	Machine fault diagnostic and prognostic techniques have been the considerable subjects of condition-based maintenance system in the recent time due to the potential advantages that could be gained from reducing downtime, decreasing maintenance costs, and increasing machine availability. For the past few years, research on machine fault diagnosis and prognosis has been developing rapidly. These publications covered in the wide range of statistical approaches to model-based approaches. With the aim of synthesizing and providing the information of these researches for researcher's community, this paper attempts to summarize and classify the recent published techniques in diagnosis and prognosis of rotating machinery. Furthermore, it also discusses the opportunities as well as the challenges for conducting advance research in the field of machine prognosis.	Rotating Machinery, Fault Diagnosis, Prognosis
72-79	Effect of Internal Flow in Symmetric and Asymmetric Micro Regenerative Pump Impellers on Their Pressure Performance	Hironori Horiguchi, Shinji Matsumoto, Yoshinobu Tsujimoto, Masaaki Sakagami, Shigeo Tanaka	The effect of symmetric and asymmetric micro regenerative pump impellers on their pressure performance was studied. The shut off head of the pump with the symmetric impeller was about 2.5 times as that with the asymmetric impeller. The computation of the internal flow was performed to clarify the cause of the increase of the head. It was found that the contribution of the angular momentum supply was larger than that of shear stress for the head development in both cases. The larger head and momentum supply in the case of the symmetric impeller were caused by larger recirculated flow rate and larger angular momentum difference between the inlet and outlet to the impeller. The larger recirculated flow rate was caused by smaller pressure gradient in the direction of recirculated flow. The decrease of the circumferential velocity in the casing was attributed to the smaller local flow rate in the casing.	Regenerative Pump, Pump Performance, Internal Flow, Leakage Flow, Geometry

	80-91	Prediction of Specific Noise Based on Internal Flow of Forward Curved Fan	Soichi Sasaki, Hidechito Hayashi, Makoto Hatakeyama	In this study, a prediction theory for specific noise that is the overall characteristic of the fan has been proposed. This theory is based on total pressure prediction and broadband noise prediction. The specific noises of two forward curved fans with different number of blades were predicted. The flow around the impeller having 120 blades (MF120) was more biased at a certain positions than the impeller with 40 blades (MF40). An effective domain of the energy conversion of MF40 has extended overall than MF120. The total pressure was affected by the slip factor and pressure loss caused by the vortex flow. The suppression of a major pressure drop by the vortex flow and expansion of the effective domain for energy conversion contributed to an increase in the total pressure of MF40 at the design point. The position of maximum relative velocity was different for each fan. The relative velocity of MF120 was less than that of MF40 due to the deviation angle. The specific noise of MF120 was 2.7 dB less than that of MF40 due to the difference in internal flow. It has been quantitatively estimated that the deceleration in the relative velocity contributed to the improvement in the overall performance.	Centrifugal Fan, Internal Flow, Wake, Vortex, Pressure Drop, Aerodynamic Noise	
	92-101	Performance Comparison of Two Wind Turbine Generator Systems Having Two Types of Control Methods	Sumio Saito, Satoshi Sekizuka	The purpose of this paper is to gain a greater understanding of the performance of practical wind turbine generating systems with differing output power controllers and controlling means for wind turbine speed. Subjected wind turbines, both equipped with an asynchronous power generator, are located at two sites and are defined as wind turbine A and wind turbine B in this study, respectively. Their performance differences are examined by measuring wind speed and electric parameters. The study suggests that both wind turbines have a clear linkage between current and output power fluctuations. Comparison of the fluctuations to wind speed fluctuation, although they are triggered primarily by wind speed fluctuation, clearly indicates the specific behaviors inherent to the respective turbine control mechanisms.	Wind turbine, Wind speed, Induction generator, Asynchronous generator, Power curve, Acoustic noise	
		Page	Title	Author	Abstract	Keywords
	102-109	Design Optimization of a High Specific Speed Francis Turbine Using Multi-Objective Genetic Algorithm	Kazuyuki Nakamura, Sadao Kurosawa	A design optimization system for Francis turbine was developed. The system consists of design program and CFD solver. Flow passage shapes are optimized automatically by using the system with Multi-Objective Genetic Algorithm (MOGA). In this study, the system was applied to a high specific speed Francis turbine (nSP = 250m-KW). The runner profile and the draft tube shape were optimized to decrease hydraulic losses. As the results, it was shown that the turbine efficiency was improved in wide operating range, furthermore, the height of draft tube was reduced with the hydraulic performance kept.	Francis Turbine, Multi-Objective Genetic Algorithm, Optimization Design, Performance	
	110-120	Numerical Analysis of the Whole Field Flow in a Centrifugal Fan for Performance Enhancement - The Effect of Boundary Layer Fences of Different Configurations	K. Vasudeva Karanth, N. Yagnesh Sharma	Generally the fluid flows within the centrifugal impeller passage as a decelerating flow with an adverse pressure gradient along the stream wise path. This flow tends to be in a state of instability with flow separation zones on the suction surface and on the front shroud. Hence several experimental attempts were earlier made to assess the efficacy of using boundary layer fences to trip the flow in the regions of separation and to make the flow align itself into stream wise direction so that the losses could be minimized and overall efficiency of the diffusion process in the fan could be increased. With the development of CFD, an extensive numerical whole field analysis of the effect of boundary layer fences in discrete regions of suspected separation points is possible. But it is found from the literature that there have been no significant attempts to use this tool to explore numerically the utility of the fences on the flow field. This paper attempts to explore the effect of boundary layer fences corresponding to various geometrical configurations on the impeller as well as on the diffuser. It is shown from the analysis that the fences located on the impellers near the trailing edge on pressure side and suction side improves the static pressure recovery across the fan. Fences provided at the radial mid-span on the pressure side of the diffuser vane and near the leading edge and trailing edge of the suction side of diffuser vanes also improve the static pressure recovery across the fan.	Boundary layer fence, Flow separation, Sliding mesh, Unsteady analysis, Recirculation zone, Jets and wakes	
	121-126	Experimental Study on Internal Flow of a Mini Centrifugal Pump by PIV Measurement	Yulin Wu, Huijing Yuan, Jie Shao, Shuhong Liu	The internal flow field in a centrifugal pump working at the several flow conditions has been measured by using the particle image velocimetry (PIV) technique with the laser induced fluorescence (LIF) particles and the refractive index matched (RIM) facilities. The impeller of the centrifugal pump has an outlet diameter in 100mm, and consists of six two-dimensional curvature backward swept blades of constant thickness. Measured results give reliable flow patterns in the pump. It is obvious that application of LIF particle and RIM are the key methods to obtain the right PIV measured results in pump internal flow.	Centrifugal pump, PIV, laser induced fluorescence, refractive index matched	
	127-135	Study on Design of Air-water Two-phase Flow Centrifugal Pump Based on Similarity Law	Naoki Matsushita, Akinori Furukawa, Satoshi Watanabe, Kusuo Okuma	A conventional centrifugal pump causes a drastic deterioration of air-water two-phase flow performances even at an air-water two-phase flow condition of inlet void fraction less than 10% in the range of relatively low water flow rate. Then we have developed a two-phase flow centrifugal pump which consists of a tandem arrangement of double rotating cascades and blades of outer cascade have higher outlet angle more than 90°. In design of the two-phase flow pump for various sized and operating conditions, similarity relations of geometric dimensions to hydraulic performances is very useful. The similarity relations of rotational speed, impeller diameter and blade height are investigated for the developed impeller in the present paper. As the results, the similarity law of rotational speed and impeller diameter is clarified experimentally even in two-phase flow condition. In addition, influences of blade height on air-water two-phase flow performances indicate a little difference from the similarity relations.	Centrifugal pump, Air-water two-phase flow, Pump performance, Similarity law	
	136-146	Experimental Research for Performance and Noise of Small Axial Fan	Takahiro Ito, Gaku Minorikawa, Qinyin Fan	Small axial fans have become widely used as cooling devices in recent years. Because of their increasing importance, studies have been conducted on ways to improve the performance and reduce the noise of such fans. In this report, a small axial fan with a diameter of 85 mm (a type popularity used in personal computer or workstation) was selected for further examination. The influence on aerodynamic performance and noise of such frame design parameters as blade tip clearance results in a decrease of discrete frequency noise and an increase of broad-spectrum noise. As for the most suitable design refinement in terms of fan efficiency, we found that the treatment of outlet corner roundness and altering spoke skew to the direction counter to that of fan rotation was effective.	Axial Flow Fan, Performance, Efficiency, Noise, Frame	
	147-155	Dynamic Characteristics of the Radial Clearance Flow between Axially Oscillating Rotational Disk and Stationary Disk	Hironori Horiguchi, Yoshinori Ueno, Koutaro Takahashi, Kazuyoshi Miyagawa, Yoshinobu Tsumimoto	Dynamic characteristics of the clearance flow between an axially oscillating rotational disk and a stationary disk were examined by experiments and computations based on a bulk flow model. In the case without pressure fluctuations at the inlet and outlet of the clearance, parallel and contracting flow paths had an effect to stabilize the axial oscillation of the rotating disk. The enlarged flow path had an effect to destabilize the axial oscillation due to the negative damping and stiffness for outward and inward flows, respectively. It was shown that the fluid force can be decomposed into the component caused by the inlet or outlet pressure fluctuation without the axial oscillation and that due to the axial oscillation without the inlet or outlet pressure fluctuation. A method to predict the stiffness and damping coefficients is proposed for general cases when the device is combined with an arbitrary flow system.	Pump, Flow Induced Vibration, Axisymmetric Flow, Clearance Flow, Balance Disk, Bulk Flow Model	
	156-164	A Review of Ocean Wave Power Extraction; the primary interface	W. B. Wan Nik, A. M. Muzathik, K. B. Samo, M. Z. Ibrahim	This paper aims to describe the importance of data, data collection methods, parameters to estimate the potential of wave energy and environmental impacts. The technical and economical status in wave energy conversion is outlined. Power and energy efficiency relationships are discussed. Many different types of wave-energy converters have been detailed. The progress in wave energy conversion in Malaysia is reviewed.	Wave energy, Wave parameters, Wave data sources, Energy conversion, Environmental impact	

165-171	Prediction of Cavitation Intensity in Pumps Based on Propagation Analysis of Bubble Collapse Pressure Using Multi-Point Vibration Acceleration Method	Masashi Fukaya, Shigeoyoshi Ono, Ryujiro Udo	We developed a "multi-point vibration acceleration method" for accurately predicting the cavitation intensity in pumps. Pressure wave generated by cavitation bubble collapse propagates and causes pump vibration. We measured vibration accelerations at several points on a casing, suction and discharge pipes of centrifugal and mixed-flow pumps. The measured vibration accelerations scattered because the pressure wave damped differently between the bubble collapse location and each sensor. In a conventional method, experimental constants are proposed without evaluating pressure propagation paths, then, the scattered vibration accelerations cause the inaccurate cavitation intensity. In our method, we formulated damping rate, transmittance of the pressure wave, and energy conversion from the pressure wave to the vibration along assumed pressure propagation paths. In the formulation, we theoretically defined a 'pressure propagation coefficient,' which is a correlation coefficient between the vibration acceleration and the bubble collapse pressure. With the pressure propagation coefficient, we can predict the cavitation intensity without experimental constants as proposed in a conventional method. The prediction accuracy of cavitation intensity is improved based on a statistical analysis of the multi-point vibration accelerations. The predicted cavitation intensity was verified with the plastic deformation rate of an aluminum sheet in the cavitation erosion area of the impeller blade. The cavitation intensities were proportional to the measured plastic deformation rates for three kinds of pumps. This suggests that our method is effective for estimating the cavitation intensity in pumps. We can make a cavitation intensity map by conducting this method and varying the flow rate and the net positive suction head (NPSH). The map is useful for avoiding the	Cavitation intensity, Bubble collapse, Pressure propagation, Vibration acceleration, Pump	
172-178	Design Optimization of Centrifugal Pump Impellers in a Fixed Meridional Geometry using DOE	Sung Kim, Young-Seok Choi, Kyoung-Yong Lee, Joonyoung Yoon	This paper reports on an investigation (using RSM with commercial CFD software) of the performance characteristics of the impeller in a centrifugal pump. Geometric parameters of vane plane development were defined with the meridional shape and frontal view of the impeller. The parameters are focused on the blade-angle distributions through the impeller in a fixed meridional geometry. For screening, a 2k factorial design has been used to identify the important design parameters. The objective functions are defined as the total head rise and the total efficiency at the design flow-rate. From the 2k factorial design results, it is found that the incidence angles and the exit blade angle are the most important parameters influencing the performance of the pump.	Centrifugal pump, Impeller, Optimization, RSM(Response surface method)	
179-188	Surrogate Based Optimization Techniques for Aerodynamic Design of Turbomachinery	Abdus Samad, Kwang-Yong Kim	Recent development of high speed computers and use of optimization techniques have given a big momentum of turbomachinery design replacing expensive experimental cost as well as trial and error approaches. The surrogate based optimization techniques being used for aerodynamic turbomachinery designs coupled with Reynolds-averaged Navier-Stokes equations analysis involve single- and multi-objective optimization methods. The objectives commonly tried to improve were adiabatic efficiency, pressure ratio, weight etc. Presently coupling the fluid flow and structural analysis is being tried to find better design in terms of weight, flutter and vibration, and turbine life. The present article reviews the surrogate based optimization techniques used recently in turbomachinery shape optimizations.	Turbomachinery, surrogate modeling, optimization, Pareto optimal front, CFD	
189-196	Study for the Increase of Micro Regenerative Pump Head	Hironori Horiguchi, Keisuke Wakiya, Yoshinobu Tsujimoto, Masaaki Sakagami, Shigeo Tanaka	The effect of inlet and outlet blade angles on a micro regenerative pump head was examined in experiments. The pump head was little increased by changing the blade angles compared with the original pump with the inlet and outlet blade angles of 0 degree. The effect of the axial clearance between the impeller and the casing on the pump head was also examined. The head was increased largely by decreasing the axial clearance. The computation of the internal flow was performed to clarify the cause of the increase of the pump head due to the decrease of the clearance. The local flow rate in the casing decreased as the leakage flow rate through the axial clearance decreased due to the decrease of the clearance. It was found that the larger head in the smaller clearance was just caused by the smaller local flow rate in the casing. In the case of the smaller clearance, the smaller local flow rate caused the smaller circumferential velocity near the front and rear sides of the impeller. This caused the increase of the angular momentum in the casing and the head.	Micro Regenerative Pump, Internal Flow, Blade Angle, Clearance, Leakage Flow	
197-205	Fluid-Oscillation Coupled Analysis for HAWT Rotor Blade (One Degree of Freedom Weak Coupling Analysis with Hinge-Spring Model)	Hiroshi Imamura, Yutaka Hasegawa, Junsuke Murata, Sho Chihara, Daisuke Takezaki, Naotsugu Kamiya	Since large-scale commercial wind turbine generator systems such as MW-class wind turbines are becoming widely operated, the vibration and distortion of the blade are becoming larger and larger. Therefore the soft structure design instead of the solid-design is one of the important concepts to reduce the structural load and the cost of the wind turbine rotors. The objectives of the study are development of the fluid-structure coupled analysis code and evaluation of soft rotor-blade design to reduce the unsteady structural blade load. In this paper, fluid-structure coupled analysis for the HAWT rotor blade is performed by free wake panel method coupled with hinge-spring blade model for the flapwise blade motion. In the model, the continuous deflection of the rotor blade is represented by flapping angle of the hinge with one degree of freedom. The calculation results are evaluated by comparison with the database of the NREL unsteady aerodynamic experiment. In the analysis the unsteady flapwise moments in yawed inflow conditions are compared for the blades with different flapwise eigen frequencies.	Wind Turbine, Free Wake Panel Method, Weak Coupling Analysis, Hinge-Spring Model, Yawed Inflow	
206-214	Cause of Cavitation Instabilities in Three Dimensional Inducer	Donghyuk Kang, Koichi Yonezawa, Hironori Horiguchi, Yutaka Kawata, Yoshinobu Tsujimoto	Alternate blade cavitation, rotating cavitation and cavitation surge in rocket turbopump inducers were simulated by a three dimensional commercial CFD code. In order to clarify the cause of cavitation instabilities, the velocity disturbance caused by cavitation was obtained by subtracting the velocity vector under non-cavitating condition from that under cavitating condition. It was found that there exists a disturbance flow towards the trailing edge of the tip cavity. This flow has an axial flow component towards downstream which reduces the incidence angle to the next blade. It was found that all of the cavitation instabilities start to occur when this flow starts to interact with the leading edge of the next blade. The existence of the disturbance flow was validated by experiments.	Inducer, Cavitation instabilities, Velocity disturbance, Three dimensional CFD	
215-222	Flow Field Change before Onset of Flow Separation	Hiroaki Hasegawa, Takeru Sugawara	Jets issuing through small holes in a wall into a freestream has proven effective in the control of flow separation. This technique is known as the vortex generator jet (VGJs) method. If a precursor signal of separation is found, the separation control system using VGJs can be operated just before the onset of separation and the flow field with no separation is always attained. In this study, we measured the flow field and the wall static pressure in a two-dimensional diffuser to find a precursor signal of flow separation. The streamwise velocity measurements were carried out in the separated shear layer and spectral analysis was applied to the velocity fluctuations at some angles with respect to the diffuser. The pattern of peaks in the spectral analysis changes as the divergence angle increases over the angle of which the whole separation occurs. This change in the spectral pattern is related to the enhancement of the growth of shear layer vortices and appears just before the onset of separation. Therefore, the growth of shear layer vortices can be regarded as a precursor signal to flow separation.	Separation, Boundary Layer, Unsteady Flow, Diffuser, Precursor	
	Page	Title	Author	Abstract	Keywords

223-231	Experimental Investigation on Separated Flows of Axial Flow Stator and Diagonal Flow Rotor	Yoichi Kinoue, Norimasa Shiomi, Toshiaki Setoguchi, Yingzi Jin	Experimental investigations were conducted for the internal flows of the axial flow stator and diagonal flow rotor. Corner separation near the hub surface and the suction surface of stator blade are mainly focused on. For the design flow rate, the values of the axial velocity and the total pressure at stator outlet decrease between near the suction surface and near the hub surface by the influence of corner wall. For the flow rate of 80-90% of the design flow rate, the corner separation of the stator between the suction surface and the hub surface is observed, which becomes widely spread for 80% of the design flow rate. At rotor outlet for 81% of the design flow rate, the low axial velocity region grows between near the suction surface of rotor and the casing surface because of the tip leakage flow of the rotor.	Axial flow stator, Diagonal flow rotor, Internal flow, Corner separation	
232-238	Rotating Choke and Choked Surge in an Axial Pump Impeller	Toshifumi Watanabe, Hideyoshi Sato, Yasuhiko Henmi, Hironori Horiguchi, Yutaka Kawata, Yoshinobu Tsujimoto	Unlike usual turbopump inducers, the axial flow pump tested operates very stably at design flow rate without rotating cavitation nor cavitation surge. Flow visualization suggests that this is because the tip cavity smoothly extends into the flow passage without the interaction with the leading edge of the next blade. However, at low flow rate and low cavitation number, choked surge and rotating choke were observed. Their correlation with the performance curve under cavitation is discussed and their instantaneous flow fields are shown.	Pump, Cavitation, Cavitation Instability, Rotating Choke, Choked Surge	
239-247	A Study of Performance and Internal Flow in a New Type of Sewage Pump	Yasuyuki Nishi, Junichiro Fukutomi	Sewage pumps are designed with a wide flow channel by, for example, sacrificing some efficiency and reducing the number of blades, in order to prevent plugging with foreign bodies. However, the behavior of foreign bodies which actually flow into a pump is extremely complex, and there are questions about whether the presumed foreign bodies will actually pass through. This paper proposes a new type of sewage pump impeller designed to further improve pump efficiency and performance in passing foreign bodies. This sewage pump impeller has a structure in which the suction flow channel of a closed type non-clog pump is wound in a helical spiral. The focus of this research was to investigate pump performance and internal flow in this single blade sewage pump impeller. The results clearly indicated the following facts: The developed sewage pump impeller exhibits high efficiency over a wide range of flow rates; internal flow of the pump is very complicated; and the internal flow state varies greatly when the flow rate changes.	Turbomachinery, Sewage Pump, Single Blade, Performance, Internal Flow	
248-253	Numerical Prediction of Unsteady Flows through Whole Nozzle-Rotor Cascade Channels with Partial Admission	Yasuhiro Sasao, Kazuhiro Monma, Tadashi Tanuma, Satoru Yamamoto	This paper presents a numerical study for unsteady flows in a high-pressure steam turbine with a partial admission stage. Compressible Navier-Stokes equations are solved by the high-order high-resolution finite-difference method based on the fourth-order compact MUSCL TVD scheme, Roe's approximate Riemann solver, and the LU-SGS scheme. The SST-model is also solved for evaluating the eddy-viscosity. The unsteady two-dimensional flows through whole nozzle-rotor cascade channels considering a partial admission are numerically investigated. 108 nozzle passages with two blockages and 60 rotor passages are simultaneously calculated. The influence of the flange in the nozzle box to the lift of rotors is predicted. Also the efficiency of the partial admission stage changing the number of blockages and the number of nozzles is parametrically predicted.	Numerical Study, Steam Turbine, Unsteady Flow, Partial Admission, Nozzle-rotor Cascade Channels	
254-259	Vibration Analysis of a Rotor considering Nonlinear Reaction of Hydrodynamic Bearing	Soo-Mok Lee, Do-Hyeong Lim, Jong-Gug Bae, Bo-Suk Yang	In this paper it was attempted to treat the hydrodynamic journal bearing as a time-based nonlinear reaction source in each step of rotor rotation in order to observe the bearing effect more realistically and accurately in stead of the conventional method of simple linearized stiffness and damping. Lubrication analysis based on finite element method is employed to calculate the hydrodynamic reaction of bearing and Newmark's method was used to calculate the rotor dynamics in the time domain. Simulation for an industrial electrical motor showed remarkable results with differences compared to those by the conventional method in the dynamic behavior of the rotor.	Rotating Machinery, Lateral Vibration Analysis, Journal Bearing, Nonlinear Hydrodynamic Reaction, Unbalance Response Analysis, Electric Motor	
260-268	One-Dimensional Analysis of Full Load Draft Tube Surge Considering the Finite Sound Velocity in the Penstock	Changkun Chen, Christophe Nicolet, Koichi Yonezawa, Mohamed Farhat, Francois Avellan, Yoshinobu Tsujimoto	The effects of acoustic modes in the penstock on the self-excited oscillation in hydraulic power system were studied by assuming a finite sound velocity in the penstock. The flow in the draft tube is considered to be incompressible assuming that the length of the draft tube is smaller than the wavelength of the oscillation. It was found that various acoustic modes in the penstock can become unstable (amplified) by the diffuser effect of the draft tube or the effect of swirl flow from the runner. Their effects on each mode are discussed.	Draft tube surge, finite sound velocity, higher order frequencies	
	Page	Title	Author	Abstract	Keywords
269-277	Unsteady Flow with Cavitation in Viscoelastic Pipes	Alexandre K. Soares, Dídia I.C. Covas, Helena M. Ramos, Luisa Fernanda R. Reis	The current paper focuses on the analysis of transient cavitating flow in pressurised polyethylene pipes, which are characterized by viscoelastic rheological behaviour. A hydraulic transient solver that describes fluid transients in plastic pipes has been developed. This solver incorporates the description of dynamic effects related to the energy dissipation (unsteady friction), the rheological mechanical behaviour of the viscoelastic pipe and the cavitating pipe flow. The Discrete Vapour Cavity Model (DVCM) and the Discrete Gas Cavity Model (DGCM) have been used to describe transient cavitating flow. Such models assume that discrete air cavities are formed in fixed sections of the pipeline and consider a constant wave speed in pipe reaches between these cavities. The cavity dimension (and pressure) is allowed to grow and collapse according to the mass conservation principle. An extensive experimental programme has been carried out in an experimental set-up composed of high-density polyethylene (HDPE) pipes, assembled at Instituto Superior Técnico de Lisboa, Portugal. The experimental facility is composed of a single pipeline with a total length of 203 m and inner diameter of 44 mm. The creep function of HDPE pipes was determined by using an inverse model based on transient pressure data collected during experimental runs without cavitating flow. Transient tests were carried out by the fast closure of the ball valves located at downstream end of the pipeline for the non-cavitating flow and at upstream for the cavitating flow. Once the rheological behaviour of HDPE pipes were known, computational simulations have been run in order to describe the hydraulic behaviour of the system for the cavitating pipe flow. The calibrated transient solver is capable of accurately describing the attenuation, dispersion and shape of observed transient	Cavitating flow, Fluid transients, Viscoelasticity, Pipelines, Experimental data	
278-285	Uncertainty in Operational Modal Analysis of Hydraulic Turbine Components	Martin Gagnon, S.-Antoine Tahan, André Coutu	Operational modal analysis (OMA) allows modal parameters, such as natural frequencies and damping, to be estimated solely from data collected during operation. However, a main shortcoming of these methods resides in the evaluation of the accuracy of the results. This paper will explore the uncertainty and possible variations in the estimates of modal parameters for different operating conditions. Two algorithms based on the Least Square Complex Exponential (LSCE) method will be used to estimate the modal parameters. The uncertainties will be calculated using a Monte-Carlo approach with the hypothesis of constant modal parameters at a given operating condition. In collaboration with Andritz-Hydro Ltd, data collected on two different stay vanes from an Andritz-Hydro Ltd Francis turbine will be used. This paper will present an overview of the procedure and the results obtained.	Flow induced vibration, modal analysis, system identification, uncertainty, modal parameters	



286-294	Influence of the Francis Turbine location under vortex rope excitation on the Hydraulic System Stability	S. Alligné, C. Nicolet, P. Allenbach, B. Kawkabani, J.-J. Simond, F. Avellan	Hydroelectric power plants are known for their ability to cover variations of the consumption in electrical power networks. In order to follow this changing demand, hydraulic machines are subject to off-design operation. In that case, the swirling flow leaving the runner of a Francis turbine may act under given conditions as an excitation source for the whole hydraulic system. In high load operating conditions, vortex rope behaves as an internal energy source which leads to the self excitation of the system. The aim of this paper is to identify the influence of the full load excitation source location with respect to the eigenmodes shapes on the system stability. For this, a new eigenanalysis tool, based on eigenvalues and eigenvectors computation of the nonlinear set of differential equations in SIMSEN, has been developed. First the modal analysis method and linearization of the set of the nonlinear differential equations are fully described. Then, nonlinear hydro-acoustic models of hydraulic components based on electrical equivalent schemes are presented and linearized. Finally, a hydro-acoustic SIMSEN model of a simple hydraulic power plant, is used to apply the modal analysis and to show the influence of the turbine location on system stability. Through this case study, it brings out that modeling of the pipe viscoelastic damping is decisive to find out stability limits and unstable eigenfrequencies.	Instability, Vortex rope, Eigenvalues, Viscoelastic damping, Francis Turbine
295-302	Axisymmetric Swirling Flow Simulation of the Draft Tube Vortex in Francis Turbines at Partial Discharge	Romeo Susan-Resiga, Sebastian Muntean, Peter Stein, François Avellan	The flow in the draft tube cone of Francis turbines operated at partial discharge is a complex hydrodynamic phenomenon where an incoming steady axisymmetric swirling flow evolves into a three-dimensional unsteady flow field with precessing helical vortex (also called vortex rope) and associated pressure fluctuations. The paper addresses the following fundamental question: is it possible to compute the circumferentially averaged flow field induced by the precessing vortex rope by using an axisymmetric turbulent swirling flow model? In other words, instead of averaging the measured or computed 3D velocity and pressure fields we would like to solve directly the circumferentially averaged governing equations. As a result, one could use a 2D axisymmetric model instead of the full 3D flow simulation, with huge savings in both computing time and resources. In order to answer this question we first compute the axisymmetric turbulent swirling flow using available solvers by introducing a stagnant region model (SRM), essentially enforcing a unidirectional circumferentially averaged meridional flow as suggested by the experimental data. Numerical results obtained with both models are compared against measured axial and circumferential velocity profiles, as well as for the vortex rope location. Although the circumferentially averaged flow field cannot capture the unsteadiness of the 3D flow, it can be reliably used for further stability analysis, as well as for assessing and optimizing various techniques to stabilize the swirling flow. In particular, the methodology presented and validated in this paper is particularly useful in optimizing the blade design in order to reduce the stagnant region extent, thus mitigating the vortex rope and expanding the operating range for Francis turbines.	draft tube, vortex rope, turbulent axisymmetric flow, stagnant region model
303-314	Dynamic Analysis of Francis Runners - Experiment and Numerical Simulation	Stefan Lais, Quanwei Liang, Urs Henggeler, Thomas Weiss, Xavier Escaler, Eduard Egusquiza	The present paper shows the results of numerical and experimental modal analyses of Francis runners, which were executed in air and in still water. In its first part this paper is focused on the numerical prediction of the modal parameters by means of FEM and the validation of the FEM method. Influences of different geometries on modal parameters and frequency reduction ratio (FRR), which is the ratio of the natural frequencies in water and the corresponding natural frequencies in air, are investigated for two different runners, one prototype and one model runner. The results of the analyses indicate very good agreement between experiment and simulation. Particularly the frequency reduction ratios derived from simulation are found to agree very well with the values derived from experiment. In order to identify sensitivity of the structural properties several parameters such as material properties, different model scale and different hub geometries are numerically investigated. In its second part, a harmonic response analysis is shown for a Francis runner by applying the time dependent pressure distribution resulting from an unsteady CFD simulation to the mechanical structure. Thus, the data gained by modern CFD simulation are being fully utilized for the structural design based on life time analysis. With this new approach a more precise prediction of turbine loading and its effect on turbine life cycle is possible allowing better turbine designs to be developed.	Modal analysis, fluid structure interaction, rotor-stator interaction, harmonic response analysis, instationary CFD
315-323	Overload Surge Investigation Using CFD Data	Felix Flemming, Jason Foust, Jiri Koutnik, Richard K. Fisher	Pressure oscillations triggered by the unstable interaction of dynamic flow features of the hydraulic turbine with the hydraulic plant system - including the electrical design - can at times reach significant levels and could lead to damage of plant components or could reduce component lifetime significantly. Such a problem can arise for overload as well as for part load operation of the turbine. This paper discusses an approach to analyze the overload high pressure oscillation problem using computational fluid dynamic (CFD) modeling of the hydraulic machine combined with a network modeling technique of the hydraulic system. The key factor in this analysis is the determination of the overload vortex rope volume occurring within the turbine under the runner which is acting as an active element in the system. Two different modeling techniques to compute the flow field downstream of the runner will be presented in this paper. As a first approach, single phase flow simulations are used to evaluate the vortex rope volume before moving to more sophisticated modeling which incorporates two phase flow calculations employing cavitation modeling. The influence of these different modeling strategies on the simulated plant behavior will be discussed.	Overload Surge, CFD, System Analysis, Cavitation, Vortex Rope, Francis Turbine
324-333	Unstable Operation of Francis Pump-Turbine at Runaway: Rigid and Elastic Water Column Oscillation Modes	Christophe Nicolet, Sébastien Alligné, Basile Kawkabani, Jean-Jacques Simond, François Avellan	This paper presents a numerical simulation study of the transient behavior of a 2x340MW pump-turbine power plant, where the results show an unstable behavior at runaway. First, the modeling of hydraulic components based on equivalent schemes is presented. Then, the 2 pump-turbine test case is presented. The transient behavior of the power plant is simulated for a case of emergency shutdown with servomotor failure on Unit 1. Unstable operation at runaway with a period of 15 seconds is properly simulated using a 1-dimensional approach. The simulation results points out a switch after 200 seconds of the unstable behavior between a period of oscillations initially of 15 seconds to a period of oscillation of 2.16 seconds corresponding to the hydraulic circuit first natural period. The pressure fluctuations related to both the rigid and elastic water column mode are presented for oscillation mode characterization. This phenomenon is described as a switch between a rigid and an elastic water column oscillation mode. The influence of the rotating inertia on the switch phenomenon is investigated through a parametric study.	Transient behavior, runaway, instabilities, oscillation modes

334-345	Dynamic Simulation of Pump-Storage Power Plants with different variable speed configurations using the Simsen Tool	Klaus Krüger, Jiri Koutnik	Pumped storage power plants are playing a significant role in the contribution to the stabilization of an electrical grid, above all by stable operation and fast reaction to sudden load respectively frequency changes. Optimized efficiency and smooth running characteristics both in pump and turbine operation, improved stability for synchronization in turbine mode, load control in pump mode operation and also short reaction times may be achieved using adjustable speed power units. Such variable speed power plants are applicable for high variations of head (e.g. important for low head pump-turbine projects). Due to the rapid development of power semiconductor and frequency converter technology, feasible solutions can be provided even for large hydro power units. Suitable control strategies as well as clear design criteria contribute significantly to the optimal usage of the pump turbine and motor-generators. The SIMSEN tool for dynamic simulations has been used for comparative investigations of different configurations regarding the power converter topology, types of semiconductors and types of motor-generators including the coupling to the hydraulic system. A brief overview of the advantages & disadvantages of the different solutions can also be found in this paper. Using this approach, a customized solution minimizing cost and exploiting the maximum usage of the pump-turbine unit can be developed in the planning stage of new and modernization pump storage projects.	pumped storage, pump-turbine, variable speed, dynamic simulation, optimization, power converters
346-352	Large Eddy Simulation of a High Reynolds Number Swirling Flow in a Conical Diffuser	Cédric Duprat, Olivier Métais, Thomas Laverne	The objective of the present work is to improve numerical predictions of unsteady turbulent swirling flows in the draft tubes of hydraulic power plants. We present Large Eddy Simulation (LES) results on a simplified draft tube consisting of a straight conical diffuser. The basis of LES is to solve the large scales of motion, which contain most of the energy, while the small scales are modeled. LES strategy is here preferred to the average equations strategies (RANS models) because it resolves directly the most energetic part of the turbulent flow. LES is now recognized as a powerful tool to simulate real applications in several engineering fields which are more and more frequently found. However, the cost of large-eddy simulations of wall bounded flows is still expensive. Bypass methods are investigated to perform high-Reynolds-number LES at a reasonable cost. In this study, computations at a Reynolds number about $2 \cdot 10^6$ are presented. This study presents the result of a new near-wall model for turbulent boundary layer taking into account the streamwise pressure gradient (adverse or favorable). Validations are made based on simple channel flow, without any pressure gradient and on the data base ERCOFTAC. The experiments carried out by Clausen et al. [1] reproduce the essential feature of the complex flow and are used to develop and test closure models for such flows.	Large-Eddy Simulation, Near-wall scaling, Swirling flows, CFD
353-362	Surface Roughness Impact on Francis Turbine Performances and Prediction of Efficiency Step Up	Pierre Maruzewski, Vlad Hasmatuchi, Henri-Pascal Mombelli, Danny Burggraave, Jacob Iosfin, Peter Finnegan, François Avellan	In the process of turbine modernizations, the investigation of the influences of water passage roughness on radial flow machine performance is crucial and validates the efficiency step up between reduced scale model and prototype. This study presents the specific losses per component of a Francis turbine, which are estimated by CFD simulation. Simulations are performed for different water passage surface roughness heights, which represents the equivalent sand grain roughness height. As a result, the boundary layer logarithmic velocity profile still exists for rough walls, but moves closer to the wall. Consequently, the wall friction depends not only on roughness height but also on its shape and distribution. The specific losses are determined by CFD numerical simulations for each component of the prototype, taking into account its own specific sand grain roughness height. The model efficiency step up between reduced scale model and prototype value is finally computed by the assessment of specific losses on prototype and by evaluating specific losses for a reduced scale model with smooth walls. Furthermore, surveys of rough walls of each component were performed during the geometry recovery on the prototype and comparisons are made with experimental data from the EPFL Laboratory for Hydraulic Machines reduced scale model measurements. This study underlines that if rough walls are considered, the CFD approach estimates well the local friction loss coefficient. It is clear that by considering sand grain roughness heights in CFD simulations, its forms a significant part of the global performance estimation. The availability of the efficiency field measurements provides an unique opportunity to assess the CFD method in view of a systematic approach for turbine modernization step up evaluation. <b>Moreover, this paper states that CFD is a very promising tool for future.</b>	Francis Turbine, Model and Prototype Testing, CFD Simulation, Efficiency Step Up
363-374	Study of Stay Vanes Vortex-Induced Vibrations with different Trailing-Edge Profiles Using CFD	Alexandre D'Agostini Neto, Fábio Saltara	The 2D flow around 13 similar stay-vane profiles with different trailing edge geometries is investigated to determinate the main characteristics of the excitation forces for each one of them and their respective dynamic behaviors when modeled as a free-oscillating system. The main goal is avoid problems with cracks of hydraulic turbines components. A stay vane profile with a history of cracks was selected as the basis for this work. The commercial finite-volume code FLUENT® was employed in the simulations of the stationary profiles and, then, modified to take into account the transversal motion of elastically mounted profiles with equivalent structural stiffness and damping. The k- $\omega$ SST turbulence model is employed in all simulations and a deforming mesh technique used for models with profile motion. The static-model simulations were carried out for each one of the 13 geometries using a constant far field flow velocity value in order to determine the lift force oscillating frequency and amplitude as a function of the geometry. The free-oscillating stay-vane simulations were run with a low mass-damping parameter ( $m^*c=0.0072$ ) and a single mean flow velocity value (5m/s). The structural bending stiffness of the stay-vane is defined by the Reduced Velocity parameter ( $V_r$ ). The dynamic analyses were divided into two sets. The first set of simulations was carried out only for one profile with $2 \leq V_r \leq 12$ . The second set of simulations focused on determining the behavior of each one of the 13 profiles in resonance.	Vortex-Induced Vibrations, Stay Vanes, Trailing-Edge Profile
375-382	Numerical prediction of pressure pulsation amplitude for different operating regimes of Francis turbine draft tubes	Andrej Lipej, Dragica Jošt, Peter Meznar, Vesko Djelic	Hydraulic instability associated with pressure fluctuations is a serious problem in hydraulic machinery. Pressure fluctuations are usually a result of a strong vortex created in the centre of a flow at the outlet of a runner. At every radial turbine and also at every single regulating axial turbine, the draft tube vortex appears at part-load operating regimes. The consequences of the vortex developed in the draft tube are very unpleasant pressure pulsation, axial and radial forces and torque fluctuation as well as turbine structure vibration. The consequences of the vortex are transferred upstream and downstream with amplitude and frequency modulation in respect of the turbine operating regime, cavitation conditions and air admitted content. Numerical prediction of the vortex appearance in the design stage is a very important task. The amplitude of the pressure pulsation is different for each operating regime therefore the main goal of this research was to numerically predict pressure pulsation amplitude versus different guide vane openings and to compare the results with experimental ones. For the numerical flow analysis of a complete Francis turbine (FT), the computer code ANSYS-CFX11 has been used.	radial turbine, vortex, pressure pulsation
383-391	A New Concept of Hydraulic Design of Water Turbine Runners	Jindrich Vesely, Frantisek Pochlyly, Jiri Obrovsky, Josef Mikulasek	Vibrations at different frequencies with a different intensity as well as a pressure pulsation with different parameters are two phenomena which can be observed at different water turbines. Due to the vibration and the pressure pulsation some restrictions of water turbine operation range are applied. Similar problems with the efficiency level in a wide water turbine operation range are the basic problems which are solved for ages. A theoretical and practical solution of the above mentioned problems is very much time and money consuming. The paper describes a new theoretical solution of the excitation and pressure pulsation decrease as well as extension of the operational range with high efficiency level. The new concept to decrease the vibrations and pressure pulsations is based on a heterogeneous runner blade geometry generation. The new concept of the runner geometry design was numerically tested at a low specific speed pump turbine, see Fig. 1, and basic points of the concept are presented in this paper.	Pressure pulsation, Detuning, Heterogeneous blade passages



392-399	Development of The New High Specific Speed Fixed Blade Turbine Runner	Ales Skotak, Josef Mikulasek, Jiri Obrovsky	The paper concerns the description of the step by step development process of the new fixed blade runner called "Mixer" suitable for the uprating of the Francis turbines units installed at the older low head hydropower plants. In the paper the details of hydraulic and mechanical design are presented. Since the rotational speed of the new runner is significantly higher than the rotational speed of the original Francis one, the direct coupling of the turbine to the generator can be applied. The maximum efficiency at prescribed operational point was reached by the geometry optimization of two most important components. In the first step the optimization of the draft tube geometry was carried out. The condition for the draft tube geometry optimization was to design the new geometry of the draft tube within the original bad draft tube shape without any extensive civil works. The runner blade geometry optimization was carried out on the runner coupled with the draft tube domain. The blade geometry of the runner was optimized using automatic direct search optimization procedure. The method used for the objective function minimum search is a kind of the Nelder-Mead simplex method. The objective function concerns efficiency, required net head and cavitation features. After successful hydraulic design the modal and stress analysis was carried out on the prototype scale runner. The static pressure distribution from flow simulation was used as a load condition. The modal analysis in air and in water was carried out and the results were compared. The final runner was manufactured in model scale and it is going to be tested in hydraulic laboratory. Since the turbine with the fixed blade runner does not allow double regulation like in case of full Kaplan turbine, it can be profitably used mainly at power plants with smaller changes of operational conditions or in	fixed blade turbine, runner, draft tube, optimization, uprating, efficiency	
400-408	Validation of a CFD model for hydraulic seals	Vincent Le Roy, François Guibault, Thi C. Vu	Optimization of seal geometries can reduce significantly the energetic losses in a hydraulic seal [1], especially for high head runner turbine. In the optimization process, a reliable prediction of the losses is needed and CFD is often used. This paper presents numerical experiments to determine an adequate CFD model for straight, labyrinth and stepped hydraulic seals used in Francis runners. The computation is performed with a finite volume commercial CFD code with a RANS low Reynolds turbulence model. As numerical computations in small radial clearances of hydraulic seals are not often encountered in the literature, the numerical results are validated with experimental data on straight seals and labyrinth seals. As the validation is satisfactory enough, geometrical optimization of hydraulic seals using CFD will be studied in future works.	CFD, SST turbulence model, hydraulic seal, straight seal, labyrinth seal, experimental validation	
409-417	Experimental Study on Surge Inception in a Centrifugal Compressor	Hideaki Tamaki	An investigation of surge inception in a centrifugal compressor was done with measurements of steady and unsteady static pressure. Vaneless diffuser and vaned diffuser were tested. Analyses of the static pressure and the pressure fluctuation showed that stall at the impeller leading edge occurred at first, and then it extended to downstream. In case of the vaneless diffuser, deterioration of the pressure rise in the impeller triggered instability. For the vane diffuser, instability that was generated in the impeller propagated into the vane diffuser, however the pressure recovery by the vane diffuser made the operation of the compressor stable at low flow rate.	Centrifugal Compressor, Surge, Pressure Fluctuation, Vaned Diffuser	
418-425	Hydrographic Model Test on Prevention against Vortex Occurrence for Vertical Bulb Turbine	Shoichi Yamato, Shogo Nakamura, Akinori Furukawa	A vertical bulb turbine unit with elbow type draft tube has been developed due to avoidance of complicated assembling and long standstill period at overhaul in comparison with conventional horizontal bulb turbine unit. Before designing the prototype vertical bulb unit, a hydrographic model test was carried out to establish the ideal design concept for this innovative generating unit.	Water turbine, Air entraining vortex, Free surface flow, Surge wave, Hydrographic model test	
426-430	Design and Prototyping Micro Centrifugal Compressor for Ultra Micro Gas Turbine	Toshiyuki Hirano, Hoshio Tsujita, Ronglei Gu, Gaku Minorikawa	In order to investigate the design method for a micro centrifugal compressor, which is the most important component of an ultra micro gas turbine, an impeller having the outer diameter of 20mm was designed, manufactured and tested. The designed rotational speed is 500,000 rpm and the impeller has a fully 3-dimensional shape. The impeller was rotated at 250,000 rpm in the present study. The experimental results of the tested compressor with the vaned and the vaneless diffusers were compared. It was found that the vane diffuser attained the higher flow rate than the vaneless diffuser at the maximum pressure ratio. In addition the maximum pressure ratio was higher for the diffuser having a larger diffuser divergence angle at the high flow rate. These results were compared with those obtained by the prediction method used at the design stage.	Centrifugal Compressor, Performance Characteristics, 3-dimensional Impeller, Prediction method	
431-438	Large Eddy Simulation of the Dynamic Response of an Inducer to Flow Rate Fluctuations	Donghyuk Kang, Koichi Yonezawa, Tatsuya Ueda, Nobuhiro Yamanishi, Chisachi Kato, Yoshinobu Tsujimoto	A Large Eddy Simulation (LES) of the flow in an inducer is carried out under flow rate oscillations. The present study focuses on the dynamic response of the backflow and the unsteady pressure performance to the flow rate fluctuations under non-cavitation conditions. The amplitude of angular momentum fluctuation evaluated by LES is larger than that evaluated by RANS. However, the phase delay of backflow is nearly the same as RANS calculation. The pressure performance curve exhibits a closed curve caused by the inertia effect associated with the flow rate fluctuations. Compared with the simplified one dimensional evaluation of the inertia component, the component obtained by LES is smaller. The negative slope of averaged performance curve becomes larger under unsteady conditions. From the conservations of angular momentum and energy, an expression useful for the evaluation of unsteady pressure rise was obtained. The examination of each term of this expression show that the apparent decrease of inertia effects is caused by the response delay of Euler's head and that the increase of negative slope is caused by the delay of inertial term associated with the delay of backflow response. These results are qualitatively confirmed by experiments.	Inducer, Backflow Vortex, Dynamic Response to flow rate fluctuation, Large Eddy Simulation	
439-448	Inducer Design to Avoid Cavitation Instabilities	Donghyuk Kang, Toshifumi Watanabe, Koichi Yonezawa, Hironori Horiguchi, Yutaka Kawata, Yoshinobu Tsujimoto	Three inducers were designed to avoid cavitation instabilities. This was accomplished by avoiding the interaction of tip cavity with the leading edge of the next blade. The first one was designed with extremely larger leading edge sweep, the second and third ones were designed with smaller incidence angle by reducing the inlet blade angle or increasing the design flow rate, respectively. The inducer with larger design flow rate has larger outlet blade angle to obtain sufficient pressure rise. The inducer with larger sweep could suppress the cavitation instabilities in higher flow rates more than 95% of design flow coefficient, owing to weaker tip leakage vortex cavity with stronger disturbance by backflow vortices. The inducer with larger outlet blade angle could avoid the cavitation instabilities at higher flow rates, owing to the extension of the tip cavity along the suction surface of the blade. The inducer with smaller inlet blade angle could avoid the cavitation instabilities at higher flow rates, owing to the occurrence of the cavity first in the blade passage and its extension upstream. The cavity shape and suction performance were reasonably simulated by three dimensional CFD computations under the steady cavitating condition, except for the backflow vortex cavity. The difference in the growth of cavity for each inducer is explained from the difference of the pressure distribution on the suction side of the blades.	Inducer, Cavitation instability, Tip cavity, CFD computation	
449-455	A New Blade Profile for Bidirectional Flow Properly Applicable to a Two-stage Jet Fan	Michihiro Nishi, Shuhong Liu, Kouichi Yoshida, Minoru Okamoto, Hiroyasu Nakayama	A reversible axial flow fan called jet fan has been widely used for longitudinal ventilation in road tunnels to secure a safe and comfortable environment cost-effectively. As shifting the flow direction is usually made by only switching the rotational direction of an electric motor due to heavy duty, rotor blades having identical aerodynamic performance for bidirectional flow should be necessary. However, such aerodynamically desirable blades haven't been developed sufficiently, since most of the related studies have been done from the viewpoint of unidirectional flow. In the present paper, we demonstrate a method to profile the blade section suitable for bidirectional flow, which is validated by studying the aerodynamic performances of rotor blades of a two-stage jet fan experimentally and numerically.	Blade profile, Wall curvature, Bidirectional flow, Jet fan, Aerodynamic performance, Turbulent flow analysis	
	Page	Title	Author	Abstract	Keywords

1-10	Measurements of Minute Unsteady Pressure on Three-Dimensional Fan with Arbitrary Axis Direction	Katsuya Hirata, Takuya Fuchi, Yusuke Onishi, Akira Takushima, Seiji Sato, Jiro Funaki	The present study is a fundamental approach to develop the measuring technology for minute fluctuating pressures on the three-dimensional blade surfaces of the fan which rotates with an arbitrary rotation-axis direction. In this situation, we are required to correct the centrifugal-force effect, the gravitational-force effect and the other leading-error effects for accurate measurements of the minute pressures. The working fluid is air. A pressure transducer rotating with an arbitrary attitude is closely sealed by a twofold shroud system. The rotational motion with an arbitrary attitude is produced by fixing the pressure transducer to the cantilever which is connected to a motor-driven disc of 500mm in diameter and 5mm in thickness. As a result, we have quantitatively determined main governing effects upon the non-effective component of the pressure-transducer signal	Fluctuating pressure, Pressure measurement, Unsteady pressure, Fan, Low speed
11-19	Flow Investigations in the Crossover System of a Centrifugal Compressor Stage	K. Srinivasa Reddy, G. V. Ramana Murty, A Dasgupta, K. V. Sharma	The performance of the crossover system of a centrifugal compressor stage consisting of static components of 180° U-bend, return channel vanes and exit ducting with a 90° bend is investigated. This study is confined to the assessment of performance of the crossover system by varying the shape of the return channel vanes. For this purpose two different types of Return Channel Vanes (RCV1 and RCV2) were experimentally investigated. The performance of the crossover system is discussed in terms of total pressure loss coefficient, static pressure recovery coefficient and vane surface pressure distribution. The experimentation was carried out on a test setup in which static swirl vanes were used to simulate the flow at the exit of an actual centrifugal compressor impeller with a design flow coefficient of 0.053. The swirl vanes are connected to a mechanism with which the flow angle at the inlet of U-bend could be altered. The measurements were taken at five different operating conditions varying from 70% to 120% of design flow rate. On an overall assessment RCV1 is found to give better performance in comparison to RCV2 for different U-bend inlet flow angles. The performance of RCV2 was verified using numerical studies with the help of a CFD Code. Three dimensional sector models were used for simulating the flow through the crossover system. The turbulence was predicted with standard k-ε, 2-equation model. The iso-Mach contour plots on different planes and development of secondary flows were visualized through this study.	Centrifugal Compressor, Return Channel Vanes, Inlet Flow Angle, Swirl Angle, Performance, Vane Surface, Pressure Coefficient
20-28	Design and Analysis of a Controlled Diffusion Aerofoil Section for an Axial Compressor Stator and Effect of Incidence Angle and Mach No. on Performance of CDA	Nilesh P. Salunke, S. A. Channiwala	This paper deals with the Design and Analysis of a Controlled Diffusion Aerofoil (CDA) Blade Section for an Axial Compressor Stator and Effect of incidence angle and Mach No. on Performance of CDA. CD blade section has been designed at Axial Flow Compressor Research Lab, Propulsion Division of National Aerospace Laboratories (NAL), Bangalore, as per geometric procedure specified in the U.S. patent (4). The CFD analysis has been performed by a 2-D Euler code (Denton's code), which gives surface Mach No. distribution on the profiles. Boundary layer computations were performed by a 2-D boundary layer code (NALSOF0801) available in the SOFTS library of NAL. The effect of variation of Mach no. was performed using fluent. The surface Mach no. distribution on the CD profile clearly indicates lower peak Mach no. than MCA profile. Further, boundary layer parameters on CD aerofoil at respective incidences have lower values than corresponding MCA blade profile. Total pressure loss on CD aerofoil for the same incidence range is lower than MCA blade profile.	CDA, MCA, Blade Profile, Transonic compressor design, Axial Flow Compressor
29-38	Surrogate Modeling for Optimization of a Centrifugal Compressor Impeller	Jin-Hyuk Kim, Jae-Ho Choi, Kwang-Yong Kim	This paper presents a procedure for the design optimization of a centrifugal compressor. The centrifugal compressor consists of a centrifugal impeller, vaneless diffuser and volute. And, optimization techniques based on the radial basis neural network method are used to optimize the impeller of a centrifugal compressor. The Latin-hypercube sampling of design-of-experiments is used to generate the thirty design points within design spaces. Three-dimensional Reynolds-averaged Navier-Stokes equations with the shear stress transport turbulence model are discretized by using finite volume approximations and solved on hexahedral grids to evaluate the objective function of the total-to-total pressure ratio. Four variables defining the impeller hub and shroud contours are selected as design variables in this optimization. The results of optimization show that the total-to-total pressure ratio of the optimized shape at the design flow coefficient is enhanced by 2.46% and the total-to-total pressure ratios at the off-design points are also improved significantly by the design optimization.	Centrifugal compressor, Impeller, Meridian plane, Optimization, Pressure ratio, Efficiency
39-49	Performance Improvement of High Speed Jet Fan	Young-Seok Choi, Joon-Hyung Kim, Kyoung-Yong Lee, Sang-Ho Yang	In this paper, a numerical study has been carried out to investigate the influence of jet fan design variables on the performance of a jet fan. In order to achieve an optimum jet fan design and to explain the interactions between the different geometric configurations in the jet fan, three-dimensional computational fluid dynamics and the DOE method have been applied. Several geometric variables, i.e., hub-tip ratio, meridional shape, rotor stagger angle, number of rotor-stator blades and stator geometry, were employed to improve the performance of the jet fan. The objective functions are defined as the exit velocity and total efficiency at the operating condition. Based on the results of computational analyses, the performance of the jet fan was significantly improved. The performance degradations when the jet fan is operated in the reverse direction are also discussed.	Optimum design, Jet fan, Rotor blade, Stator blade, Bell-mouth, CFD (Computational Fluid Dynamics), DOE (Design of Experiments)
50-57	Cavitation Surge Suppression of Pump Inducer with Axi-asymmetrical Inlet Plate	Jun-Ho Kim, Koichi Ishzaka, Satoshi Watanabe, Akinori Furukawa	The attachment of inducer in front of main impeller is a powerful method to improve cavitation performance. Cavitation surge oscillation, however, often occurs at partial flow rate and extremely low suction pressure. As the cavitation surge oscillation with low frequency of about 10 Hz occurs in a close relation between the inlet backflow cavitation and the growth of blade cavity into the throat section of blade passage, one method of installing an axisymmetrical plate upstream of inducer has been proposed to suppress the oscillation. The inlet flow distortion due to the axi-asymmetrical plate makes different elongations of cavities on all blades, which prevent the flow from becoming simultaneously unstable at all throat sections. In the present study, changes of the suppression effects with the axial distance between the inducer inlet and the plate and the changes with the blockage ratios of plate area to the cross-sectional area of inducer inlet are investigated for helical inducers with tip blade angles of 8° and 14°. Then a conceivable application will be proposed to suppress the cavitation surge oscillation by installing axi-asymmetrical inlet plate.	Turbomachinery, Pump inducer, Cavitation surge suppression, Axi-asymmetrical inlet plate
58-66	A Two-Dimensional Study of Transonic Flow Characteristics in Steam Control Valve for Power Plant	Koichi Yonezawa, Yoshinori Terachi, Toru Nakajima, Yoshinobu Tsujimoto, Kenichi Tezuka, Michitsugu Mori, Ryo Morita, Fumio Inada	A steam control valve is used to control the flow from the steam generator to the steam turbine in thermal and nuclear power plants. During startup and shutdown of the plant, the steam control valve is operated under a partial flow conditions. In such conditions, the valve opening is small and the pressure deference across the valve is large. As a result, the flow downstream of the valve is composed of separated unsteady transonic jets. Such flow patterns often cause undesirable large unsteady fluid force on the valve head and downstream pipe system. In the present study, various flow patterns are investigated in order to understand the characteristics of the unsteady flow around the valve. Experiments are carried out with simplified two-dimensional valve models. Two-dimensional unsteady flow simulations are conducted in order to understand the experimental results in detail. Scale effects on the flow characteristics are also examined. Results show three types of oscillating flow pattern and three types of static flow patterns.	Transonic flow, Control valve, Flow oscillation

67-79	Rotordynamic Instabilities Caused by the Fluid Force Moments on the Backshroud of a Francis Turbine Runner	Bingwei Song, Hironori Horiguchi, Zhenyue Ma, Yoshinobu Tsujimoto	Severe flexural vibration of the rotor shaft of a Francis turbine runner was experienced in the past. It was shown that the vibration was caused by the fluid forces and moments on the backshroud of the runner associated with the leakage flow through the back chamber. The aim of the present paper is to study the self-excited rotor vibration caused by the fluid force moments on the backshroud of a Francis turbine runner. The rotor vibration includes two fundamental motions, one is a whirling motion which only has a linear displacement and the other is a precession motion which only has an angular displacement. Accordingly, two types of fluid force moment are exerted on the rotor, the moment due to whirl and the moment due to precession. The main focus of the present paper is to clarify the contribution of each moment to the self-excited vibration of an overhung rotor. The runner was modeled by a disk and the whirl and the precession moments on the backshroud of the runner caused by the leakage flow were evaluated from the results of model tests conducted before. A lumped parameter model of a cantilevered rotor was used for the vibration analysis. By examining the frequency, the damping rate, the amplitude ratio of lateral and angular displacements for the cases with longer and shorter overhung rotor, it was found that the precession moment is more important for smaller overhung rotors and the whirl moment is more important for larger overhung rotors, although both types of moment due to the leakage flow can cause self-excited vibration of an overhung rotor.	Rotordynamic instabilities, fluid force moments, Francis turbine runner, backshroud	
80-90	Experimental Study and Numerical Simulation of Cavity Oscillation in a Diffuser with Swirling Flow	Changkun Chen, Christophe Nicolet, Koichi Yonezawa, Mohamed Farhat, Francois Avellan, Kazuyoshi Miyagawa, Yoshinobu Tsujimoto	The cavity oscillation with swirling flow in hydraulic power generating systems was studied by a simple experiment and numerical simulation. Several types of fluctuation were observed in the experiment, including the cavitation surge caused by the diffuser effect and the vortex precession by the swirling flow. Both cavitation surge and vortex precession were simulated by CFD. Detailed flow structure was examined through flow visualization and CFD.	Draft tube surge, diffuser effect, swirl effect, cavitation	
91-101	Experimental Study and Numerical Simulation of Cavity Oscillation in a Conical Diffuser	Changkun Chen, Christophe Nicolet, Koichi Yonezawa, Mohamed Farhat, Francois Avellan, Kazuyoshi Miyagawa, Yoshinobu Tsujimoto	Changkun Chen, Christophe Nicolet, Koichi Yonezawa, Mohamed Farhat, Francois Avellan, Kazuyoshi Miyagawa, Yoshinobu Tsujimoto	Draft tube surge, diffuser effect, cavitation	
	Page	Title	Author	Abstract	Keywords
102-112	Computational Study of Magnetically Suspended Centrifugal Blood Pump (The First Report: Main Flow and Gap Flow)	Yoshifumi Ogami, Daisuke Matsuoka, Masaaki Horie	Artificial heart pumps have attracted the attention of researchers around the world as an alternative to the organ used in cardiac transplantation. Conventional centrifugal pumps are no longer considered suitable for long-term application because of the possibility of occurrence of blood leakage and thrombus formation around the shaft seal. To overcome this problem posed by the shaft seal in conventional centrifugal pumps, the magnetically suspended centrifugal pump has been developed; this is a sealless rotor pump which can provide contact-free rotation of the impeller without leading to material wear. In Europe, clinical trials of this pump have been successfully performed, and these pumps are commercially available. One of the aims of our study is to numerically examine the internal flow and the effect of leakage flow through the gap between the impeller and the pump casing on the performance of the pump. The results show that the pressure head increases compared with the pump without a gap for all flow rates because of the leakage of the fluid through the gap. It was observed that the leakage flow rate in the pump is sufficiently large; further, no stagnant fluid or dead flow regions were observed in the pump. Therefore, the present pump can efficiently enhance the washout effect.	Artificial Heart, Blood Pump, Computational Fluid Dynamics, Internal Flow, Turbomachinery	
113-121	Computational Fluid Dynamics of Cavitating Flow in Mixed Flow Pump with Closed Type Impeller	Katsutoshi Kobayashi, Yoshimasa Chiba	LES(Large Eddy Simulation) with a cavitation model was performed to calculate an unsteady flow for a mixed flow pump with a closed type impeller. First, the comparison between the numerical and experimental results was done to evaluate a computational accuracy. Second, the torque acting on the blade was calculated by simulation to investigate how the cavitation caused the fluctuation of torque. The absolute pressure around the leading edge on the suction side of blade surface had positive impulsive peaks in both the numerical and experimental results. The simulation showed that those peaks were caused by the cavitation which contracted and vanished around the leading edge. The absolute pressure was predicted by simulation with -10% error. The absolute pressure around the trailing edge on the suction side of blade surface had no impulsive peaks in both the numerical and experimental results, because the absolute pressure was 100 times higher than the saturated vapor pressure. The simulation results showed that the cavitation was generated around the throat, then contracted and finally vanished. The simulated pump had five throats and cavitation behaviors such as contraction and vanishing around five throats were different from each other. For instance, the cavitations around those five throats were not vanished at the same time. When the cavitation was contracted and finally vanished, the absolute pressure on the blade surface was increased. When the cavitation was contracted around the throat located on the pressure side of blade surface, the pressure became high on the pressure side of blade surface. It caused the 1.4 times higher impulsive peak in the torque than the averaged value. On the other hand, when the cavitation was contracted around the throat located on the suction side of blade surface, the pressure became high on the suction side of	Cavitation, Numerical Simulation, Fluctuating Hydraulic Force, Pump, Blade	
122-128	Cavitation in Pump Inducer with Axi-asymmetrical Inlet Plate Observed by Multi-cameras	Jun-Ho Kim, Takashi Atono, Koichi Ishizaka, Satoshi Watanabe, Akinori Furukawa	The attachment of inducer in front of main impeller is a powerful method to improve cavitation performance; however, cavitation surge oscillation with low frequency occurs with blade cavity growing to each throat section of blade passage simultaneously. Then, one conceptual method of installing suction axi-asymmetrical plate has been proposed so as to keep every throat passage away from being unstable at once, and the effect on suppression of the oscillation were investigated. In the present study, cavitation behaviors in the inducer is observed with distributing multi-cameras circumferentially, recording simultaneously and reconstructing multi-photos on one plane field as moving a linear cascade. Observed results are utilized for discussion with other measuring results as casing wall pressure distribution. Then the suppression mechanism of oscillation by installing axi-asymmetrical inlet plate will be clarified in more details.	Pump inducer, Cavitation surge suppression, Axi-asymmetrical inlet plate, Multi-cameras observation system	
129-136	Internal Flow of a Two-Bladed Helical Inducer at an Extremely Low Flow Rate	Satoshi Watanabe, Naoki Inoue, Koichi Ishizaka, Akinori Furukawa, Jun-Ho Kim	The attachment of inducer upstream of main impeller is an effective method to improve the suction performance of turbopump. However, various types of cavitation instabilities are known to occur even at the designed flow rate as well as in the partial flow rate region. The cavitation surge occurring at partial flow rates is known to be strongly associated with the inlet back flow. In the present study, in order to understand the detailed structure of internal flow of inducer, we firstly carried out the experimental and numerical studies of non-cavitating flow, focusing on the flow field near the inlet throat section and inside the blade passage of a two bladed inducer at a partial flow rate. The steady flow simulation with cavitation model was also made to investigate the difference of flow field between in the cavitating and noncavitating conditions.	Inducer, Internal flow, Cavitation, Laser Doppler velocimetry measurement, Computational fluid dynamics	
137-149	Suppression of Cavitation Instabilities in an Inducer by Circumferential Groove and Explanation of Higher Frequency Components	Donghyuk Kang, Yusuke Arimoto, Koichi Yonezawa, Hironori Horiguchi, Yutaka Kawata, Chunill Hah, Yoshinobu Tsujimoto	The purpose of the present research is to suppress cavitation instabilities by using a circumferential groove. The circumferential groove was designed based on CFD so that the tip leakage vortex is trapped by the groove and does not interact with the next blade. Experimental results show that the groove can suppress rotating cavitation, asymmetric cavitation and cavitation surge. However, weak instabilities with higher frequency could not be suppressed by the groove. From the analysis of pressure pattern similar to that for rotor-stator interaction, it was found that the higher frequency components are caused by the interaction of backflow vortices with the inducer blades.	Inducer, Cavitation instabilities, Circumferential groove, Higher frequency component	

June, 2010					
150-159		Multi-objective Optimization of a Laidback Fan Shaped Film-Cooling Hole Using Evolutionary Algorithm	Ki-Don Lee, Afzal Husain, Kwang-Yong Kim	Laidback fan shaped film-cooling hole is formulated numerically and optimized with the help of three-dimensional numerical analysis, surrogate methods, and the multi-objective evolutionary algorithm. As Pareto optimal front produces a set of optimal solutions, the trends of objective functions with design variables are predicted by hybrid multi-objective evolutionary algorithm. The problem is defined by four geometric design variables, the injection angle of the hole, the lateral expansion angle of the diffuser, the forward expansion angle of the hole, and the ratio of the length to the diameter of the hole, to maximize the film-cooling effectiveness compromising with the aerodynamic loss. The objective function values are numerically evaluated through Reynolds-averaged Navier-Stokes analysis at the designs that are selected through the Latin hypercube sampling method. Using these numerical simulation results, the Response Surface Approximation model are constructed for each objective function and a hybrid multi-objective evolutionary algorithm is applied to obtain the Pareto optimal front. The clustered points from Pareto optimal front were evaluated by flow analysis. These designs give enhanced objective function values in comparison with the experimental designs.	Film-cooling, Numerical analysis, Multi-objective optimization, Response Surface Approximation
160-168		Prediction of Axial Thrust for Mixed-Flow Pumps with Vaned Diffuser by Using CFD	Ichiro Harada, Katsutoshi Kobayasi, Shigeyoshi Ono	It is important in pump design that the axial thrust of mixed-flow pump is predicted with high accuracy. In this paper, predictions of the axial thrust were carried out with CFD for mixed-flow pumps of three specific speeds. The region concerning the axial thrust prediction was picked out, and was divided into two parts. One of them was hydraulic part, which included the impeller and the vaned diffuser. The other was the rear part of impeller. These parts were calculated and evaluated individually. The CFD results were compared with experimental ones. They showed good agreements. It is shown that the axial thrust for a mixed-flow pump can be predicted by using CFD with practical accuracy.	Mixed-Flow Pump, Impeller, Vaned Diffuser, Axial Thrust, CFD, Prediction, Experimental Result
170-180		Cavitation Instabilities in Turbopump Inducers - Analyses in 1-3 dimensions -	Yoshinobu Tsujimoto, Hironori Horiguchi, Koichi Yonezawa	Stability analyses of 1-3 dimensional cavitating flow through turbopump inducers are reviewed with a special focus on the cause of cavitation instabilities. In one-dimensional analysis, cavitation is modeled with the cavitation compliance, defined as the decrease of cavity volume due to the increase of inlet pressure, and the mass flow gain factor, defined as the decrease of cavity volume due to the increase of flow rate. It was shown that the positive mass flow gain factor is the cause of cavitation surge and rotating cavitation. In two-dimensional stability analysis, the blade surface cavity is modeled by a free streamline with a constant pressure. It is shown that various modes of cavitation instabilities start to occur when the cavity length becomes about 65% of the blade spacing. It was found that there is a region near the cavity trailing edge in which the incidence angle to the next blade is decreased. This flow occurs to satisfy the continuity equation near the cavity closure. The cavitation instabilities start to occur when this region starts to interact with the leading edge of the next blade. In three-dimensional real flows, cavitation occurs mostly near the tip. Cavitation instabilities are simulated by three dimensional unsteady cavitating CFD. By separating out the disturbance caused by cavitation, it was found that there exists a flow component towards the trailing edge of tip cavities to fill up the volume of collapsing bubbles. This disturbance flow has an effect to reduce the incidence angle to the next blade. It was found that cavitation instabilities start to occur when this disturbance flow starts to interact with the leading edge of the next blade. So, it was found that the steady cavity length at the tip is the most important parameter in three dimensional real flow. Thus, it was found that the continuity equation plays the most important role in the mechanism of	Inducer, Cavitation, Instability, Stability Analysis, CFD
181-203		Performance Prediction of Vertical Submersible Centrifugal Slurry Pump	Lal Gopal Das, Prasanta Kumar Sen, Timir Kanti Saha, Arunabha Chanda	Performance prediction methodology for centrifugal submersible slurry pump has been presented in this paper. An in-depth study on various energy-head losses occurring through the pump flow in rotating reference frame has been carried out in this research work. Head-flow characteristics of the centrifugal pump have been accomplished in two stages. First performance of the centrifugal pump with clear water has been predicted by analyzing and deducting head losses from the theoretical head. Effects of solid particles size, specific gravity and concentration on pump slurry flow have been investigated. Additional head losses due to solid particles in the slurry have been predicted, analyzed and then deducted from clear water head to establish the performance of centrifugal slurry pump. The performance of centrifugal slurry pump has been predicted at with accuracy of 88 to 90 % for solid concentration of 18 % to 5 % by volume.	centrifugal slurry pump, head loss, performance, concentration, impeller, diffuser
204-210		Lifetime Prediction of Film Cooling Systems with and without Thermal Barrier Coating	Kyung Min Kim, Dong Hyun Lee, Hyung Hee Cho	The present study investigated temperature, thermal stress, and the lifetime in film cooling systems with and without thermal barrier coating. 3D-numerical simulations using a FEM commercial code were conducted to calculate distributions of temperature and thermal stresses. In the simulations, the surface boundary conditions used the surface heat transfer coefficients and adiabatic wall temperature which were converted from the Sherwood numbers and impermeable wall effectiveness obtained from previous mass transfer experiments. Then, the lifetime of the film cooling systems is predicted using thermal analysis data and the material creep data. The minimum lifetime is approximately 1,100 hours on the sides of the hot side surface in case without TBC.	Gas turbine heat transfer, Film cooling, Thermal analysis, Lifetime prediction, Finite element method
211-220	Page	Title	Author	Abstract	Keywords
211-220		Unsteady Swirling Flows Arising in Straight Tubes	Hiromu Tsurusaki	The objective of this study is to clarify the occurrence of the high-speed mode of unsteady swirling flows in straight tubes. The unsteady flows generated in the tube were measured by means of a semiconductor-type pressure transducer and an FFT analyzer. The high-speed mode measured has rotational speed which is approximately equal to or higher than the peripheral velocity of the swirling flow. The unsteady flow is due to cell rotation in the circumferential direction of the tube. The occurrence of the high-speed mode was confirmed, and the characteristics (rotational speed, pressure amplitude, and phase) of this mode were clarified. In order to understand the measured unsteady flows, the three dimensional vortex core profiles were discussed based on the distributions of the pressure amplitude and phase.	Unsteady Flow, Swirling Flow, Whirling Vortex, Pressure Fluctuation, Rotating stall
221-226		A Study on Performance Conversion from Model to Prototype	Masao Oshima	A prototype pump performance converted from that of a model pump shows an increase in efficiency. This paper discusses whether such increase is related to an increase in pump head and/or to a decrease in input power, based on the relationship between the tangential force on impeller blades, head loss and input power. This study revealed that the efficiency increase was brought about not only by an increase in head, but also by a decrease in input power, as the tangential force on the blades constitutes a part of the input power of a pump. A solution is proposed on how the hydraulic efficiency increase of a prototype pump should be related to its head and input power.	Performance conversion, Model pump, Prototype pump, Efficiency increase, Scale effect
227-234		Numerical Study on Mixing Performance of Straight Groove Micromixers	Hossain Shakhawat, Kwang-Yong Kim	Numerical analyses have been performed to investigate the effects of geometric parameters of a straight groove micromixer on mixing performance and pressure drop. Three-dimensional Navier-Stokes equations with two working fluids, water and ethanol have been used to calculate mixing index and pressure drop. A parametric study has been carried out to find the effects of the number of grooves per cycle, arrangement of patterned walls, and additional grooves in triangular dead zones between half cycles of grooves. The three arrangements of patterned walls in a micromixer, i.e., single wall patterned, both walls patterned symmetrically, and both walls patterned asymmetrically, have been tested. The results indicate that as the number of grooves per cycle increases the mixing index increases and the pressure drop decreases. The microchannel with both walls patterned asymmetrically shows the best mixing performance among the three different arrangements of patterned walls. Additional grooves confirm the better mixing performance and lower pressure drop.	Micromixer, Straight Groove, Numerical Analysis, Mixing, Navier-Stokes Equations, Pressure Drop

235-244	Moment Whirl due to Leakage Flow in the Back Shroud Clearance of a Rotor	Yoshinobu Tsujimoto, Zhenyue Ma, Bingwei Song, Hironori Horiguchi	Recent studies on the moment whirl due to leakage flow in the back shroud clearance of hydro-turbine runners or centrifugal pump impellers are summarized. First, destabilizing effect of leakage flow is discussed for lateral vibrations using simplified models. Then it is extended to the case of whirling motion of an overhung rotor and the criterion for the instability is obtained. The fluid moment caused by a leakage clearance flow between a rotating disk and a stationary casing was obtained by model tests under whirling and precession motion of the disk. It is shown that the whirl moment always destabilizes the whirl motion of the overhung rotor while the precession moment destabilizes the precession only when the precession speed is less than half the rotor speed. Then vibration analyses considering both whirl and precession are made by using the hydrodynamic moments determined by the model tests. For larger overhung rotors, the whirl moment is more important and cause whirl instability at all rotor speed. On the other hand, for smaller overhung rotors, the precession moment is more important and cancels the destabilizing effect of the whirl moment.	Rotordynamic Instability, Fluid Force Moment, Whirling Motion, Precession Motion, Leakage Flow	
245-252	Air-Water Two-Phase Flow Performances of Centrifugal Pump with Movable Bladed Impeller and Effects of Installing Diffuser Vanes	Shinji Sato, Akinori Furukawa	It's known that pump head of centrifugal impeller with lager blade outlet angle is kept higher in air-water two phase flow condition, though the efficiency in water single phase flow condition is inferior. In the present study, a centrifugal impeller with variable blade outlet angles, that has higher efficiencies in both water single phase flow and air-water two phase flow conditions, is proposed. And the performances of the centrifugal impeller with variable blade outlet angles were experimentally investigated in both flow conditions of single and two-phase. In addition, effects of installing diffuser vanes on the performances of centrifugal pump with movable bladed impeller were also examined. The results are as follows: (1) The movable bladed impeller that proposed in this study is effective for higher efficiency in both water single phase and air-water two phase flow conditions. (2) When diffuser vanes are installed, the efficiency of movable bladed impeller decreases particularly at large water flow rate in water single-phase flow condition; (3) The performances of movable bladed impeller are improved by installing of diffuser vanes in air-water two-phase flow condition at relatively small water rate. The improvement by installing of diffuser vanes however disappears at large water flow rate.	Centrifugal pump, Variable blade angle, Air-Water two-phase flow performance, Diffuser vane	
253-259	Investigation of Leakage Characteristics of Straight and Stepped Labyrinth Seals	Tong Seop Kim, Soo Young Kang	Leakage characteristics of two labyrinth seals with different configurations (straight vs stepped) were investigated. Leakage flows were predicted by computational fluid dynamics (CFD) for the two configurations and compared with test data. A semi-analytical leakage prediction tool was also tried to predict the leakage. It was confirmed that the CFD gives quite good agreements with test data. The analytical tool also yielded similar leakage behaviors with test results, but the overall agreement with test data was not as good as that of the CFD. The effect of flow direction in the stepped seal on leakage flow was examined. The dependence of leakage performance, in terms of flow function, on the seal clearance size was investigated. Flow function decreased with decreasing clearance in the straight seal, while the trend was reversed in the stepped seal.	labyrinth seal, straight seal, stepped seal, leakage, flow function, clearance	
260-270	Study of the Flow in Centrifugal Compressor	Cheng Xu, Ryoichi Samuel Amano	Reducing the losses of the tip clearance flow is one of the keys in an unshrouded centrifugal compressor design and development because tip clearances are large in relation to the span of the blades and also centrifugal compressors produce a sufficiently large pressure rise in single stage. This problem is more acute for a low flow high-pressure ratio impeller design. The large tip clearance would cause flow separations, and as a result it would drop both the efficiency and surge margin. Thus a design of a high efficiency and wide operation range low flow coefficient centrifugal compressor is a great challenge. This paper describes a recent development of high efficiency and wide surge margin low flow coefficient centrifugal compressor. A viscous turbomachinery optimal design method developed by the authors for axial flow machine was further extended and used in the centrifugal compressor design. The compressor has three main parts: impeller, a low solidity diffuser and volute. The tip clearance is under a special consideration in this design to allow impeller insensitiveness to the clearance. A patented three-dimensional low solidity diffuser design method is used and applied to this design. The compressor test results demonstrated to be successful to extend the low solidity diffusers to high-pressure ratio compressor. The compressor stage performance showed the total to static efficiency of the compressor being about 85% and stability range over 35%. The test results are in good agreement with the design.	centrifugal compressor, flow separation, static efficiency, turbomachinery, impeller, surge	
	Page	Title	Author	Abstract	Keywords
271-278	Performance and Flow Condition of Contra-rotating Small-sized Axial Fan at Partial Flow Rate	Toru Shigemitsu, Junichiro Fukutomi, Yuki Okabe, Kazuhiro Iuchi	Small-sized axial fans are used as air cooler for electric equipments. But there is a strong demand for higher power of fans according to the increase of quantity of heat from electric devices. Therefore, higher rotational speed design is conducted, although, it causes the deterioration of efficiency and the increase of noise. Then the adoption of contra-rotating rotors for the small-sized axial fan is proposed for the improvement of performance. In the present paper, the performance curves of the contra-rotating small-sized axial fan with 100mm diameter are shown and the velocity distributions at a partial flow rate at the inlet and the outlet of each front and rear rotor are clarified with experimental results. Furthermore, the flow conditions between front and rear rotors of the contra-rotating small-sized axial fan are investigated by numerical analysis results and causes of the performance deterioration of the contra-rotating small-sized axial fan at the partial flow rate is discussed.	Small-sized axial fan, Contra-rotating rotors, Performance, Internal flow, Partial flow rate, Numerical analysis	
279-284	Tip Clearance Losses - A Physical Based Scaling Method	Peter F. Pelz, Sascha Karstadt	Tip clearance losses occur in every turbomachine. To estimate the losses in efficiency it is important to understand the mechanism of this secondary flow. Tip clearance losses are mainly caused by a spiral vortex formed on the suction side of the blade of a turbomachine, which induces a drag and also has an influence on the incident flow of the blades. In this paper a physical based scaling method is developed out of an analytical ansatz for the tip clearance losses. This scaling method is validated by measurements on an axial fan with five different tip clearances.	Tip Clearance Losses, , , Scaling, Vortex, Prandtl	
285-291	Physical Model Investigation of a Compact Waste Water Pumping Station	Kilian Kirst, D.-H. Hellmann, Bernd Kothe, Peer Springer	To provide required flow rates of cooling or circulating water properly, approach flow conditions of vertical pump systems should be in compliance with state of the art acceptance criteria. The direct inflow should be vortex free, with low pre-rotation and symmetric velocity distribution. Physical model investigations are common practice and the best tool of prediction to evaluate, to optimize and to document flow conditions inside intake structures for vertical pumping systems. Optimization steps should be accomplished with respect to installation costs and complexity on site. The report shows evaluation of various approach flow conditions inside a compact waste water pumping station. The focus is on the occurrence of free surface vortices and the evaluation of air entrainment for various water level and flow rates. The presentation of the results includes the description of the investigated intake structure, occurring flow problems and final recommendations.	physical model investigation, waste water, pumping station	
292-300	A Numerical Study on Cavitation Suppression Using Local Cooling	Yuan-yuan Zhang, Xiao-jing Sun, Dian-gui Huang	This study strives to develop an effective strategy to inhibit cavitation inception on hydrofoils by using local cooling technique. By setting up a temperature boundary condition and cooling a small area on the upper surface of a hydrofoil, the fluid temperature around the cooling surface will be decreased and thereby the corresponding liquid saturation pressure will drop below the lowest absolute pressure within the flow field. Hence, cavitation can never occur. In this paper, a NACA0015 hydrofoil at 4° angle of attack was numerically investigated to verify the effectiveness of the proposed technique. The CFD results indicate that the cooling temperature and the cooling surface roughness are the critical factors affecting the success of such technique used for cavitation suppression.	CFD, cavitation inception, cavitation suppression, local cooling temperature, surface roughness	

301-308	Hydraulic Runner Design Method for Lifetime	Michel Sabourin, Denis Thibault, David-Alexandre Bouffard, Martin Lévesque	Quest for reliability of hydraulic runners is a concern for all mature electricity producers. The fatigue damage caused by dynamics loads is frequently the root cause of runner failure. This paper presents the damage tolerance approach based on fracture mechanics as the method chosen by Alstom and Hydro-Québec to predict effects of damage on runner lifetime and consequently to be used as a design method. This is sustained by a research on fracture mechanics properties of runner materials and by recommendations on the strategy to define a safety margin for design. The acquired knowledge permits to identify potential improvement of the runner lifetime without significant cost increase, like being more specific on some chemical composition or heat treatment.	Runner Design, Fatigue, Lifetime, Fracture Mechanics, Material Properties, Runner Failure
309-314	Optimization Design of Stainless Steel Stamping Multistage Pump Based on Orthogonal Test	Shi Weidong, Wang Chuan, Lu Weigang, Zhou Ling, Zhang Li	Stainless steel stamping multistage pump has become the mainstream of civil multi-stage pump. Combined with the technological features of stamping and welding pump, the studies of design for hydraulic parts of pump were come out. An $L_{18}(3^7)$ orthogonal experiment was designed with seven factors and three values including blade inlet angle, impeller outer diameter, guide vane blade number, etc. 18 plans were designed. The two stage of whole flow field on stainless steel stamping multistage pump at design point for design was simulated by CFD. According to the test result and optimization design with experimental research, the trends of main parameters which affect hydraulic performance were got. After being manufactured and tested, the efficiency of the optimal model pump reaches 61.36% and the single head is more than 4.8 m. Compared with the standard efficiency of 53%, the design of the stainless steel stamping pump is successful. The result would be instructive to the design of Stainless steel stamping multistage pump designed by the impeller head maximum approach.	stamping and welding pump, orthogonal experiments, internal flows, numerical simulation
315-323	Compressible Simulation of Rotor-Stator Interaction in Pump-Turbines	Jianping Yan, Jiri Koutnik, Ulrich Seidel, Björn Hübner	This work investigates the influence of water compressibility on pressure pulsations induced by rotor-stator interaction (RSI) in hydraulic machinery, using the commercial CFD solver ANSYS-CFX. A pipe flow example with harmonic velocity excitation at the inlet plane is simulated using different grid densities and time step sizes. Results are compared with a validated code for hydraulic networks (SIMSEN). Subsequently, the solution procedure is applied to a simplified 2.5-dimensional pump-turbine configuration in prototype with different speeds of sound as well as in model scale with an adapted speed of sound. Pressure fluctuations are compared with numerical and experimental data based on prototype scale. The good agreement indicates that the scaling of acoustic effects with an adapted speed of sound works well. With respect to pressure fluctuation amplitudes along the centerline of runner channels, incompressible solutions exhibit a linear decrease while compressible solutions exhibit sinusoidal distributions with maximum values at half the channel length, coinciding with analytical solutions of one-dimensional acoustics. Furthermore, in compressible simulation the amplification of pressure fluctuations is observed from the inlet of stay vane channels to the spiral case wall. Finally, the procedure is applied to a three-dimensional pump configuration in model scale with adapted speed of sound. Normalized Pressure fluctuations are compared with results from prototype measurements. Compared to incompressible computations, compressible simulations provide similar pressure fluctuations in vaneless space, but pressure fluctuations in spiral case and penstock may be much higher.	Compressible hydrodynamics, rotor-stator interaction, pump-turbine, unsteady CFD
324-331	Thermal Effects on Cryogenic Cavitating Flows around an Axisymmetric Ogive	Suguo Shi, Guoyu Wang	Cavitation in cryogenic fluids generates substantial thermal effects and strong variations in fluid properties, which in turn alter the cavity characteristics. In order to investigate the cavitation characteristics in cryogenic fluids, numerical simulations are conducted around an axisymmetric ogive in liquid nitrogen and hydrogen respectively. The modified Merkle cavitation model and energy equation which accounts for the influence of cavitation are used, and variable thermal properties of the fluid are updated with software. A good agreement between the numerical results and experimental data are obtained. The results show that vapor production in cavitation extracts the latent heat of evaporation from the surrounding liquid, which decreases the local temperature, and hence the local vapor pressure in the vicinity of cavity becomes lower. The cavitation characteristics in cryogenic fluids are obtained that the cavity seems frothy and the cavitation intense is lower. It is also found that when the fluid is operating close to its critical temperature, thermal effects of cavitation are more obviously in cryogenic fluids. The thermal effect on cavitation in liquid hydrogen is more distinctively compared with that in liquid nitrogen due to the changes of density ratio, vapour pressure gradient and other variable properties of the fluid.	cavitation, cryogenic fluid, thermal effect, numerical simulation, energy equation
332-341	Improved Suter Transform for Pump-Turbine Characteristics	Peter K. Dörfler	Standard dimensionless parameters cannot simultaneously represent all operation modes of a pump-turbine. They either have singularities at $E=0$ and multiple values in the 'unstable' areas, or else get singular at $n=0$ . P. Suter (1966) introduced an alternative set of variables which avoids singularity and always remains unique-valued. This works for non-regulated pumps but not so well for regulated machines. A modification by C.S. Martin avoids distortion at low load. The present paper describes further improvements for the representation of torque, and for closed gate (where Suter's concept does not work). The possibility to interpolate across all operation modes is likewise useful for representing other mechanical parameters of the machine. Practical application for guide vane torque and pressure pulsation data is demonstrated by examples.	Pump turbine, water hammer, characteristics, simulation, pulsation
342-351	Two-Way Coupled Fluid Structure Interaction Simulation of a Propeller Turbine	Hannes Schmucker, Felix Flemming, Stuart Coulson	During the operation of a hydro turbine the fluid mechanical pressure loading on the turbine blades provides the driving torque on the turbine shaft. This fluid loading results in a structural load on the component which in turn causes the turbine blade to deflect. Classically, these mechanical stresses and deflections are calculated by means of finite element analysis (FEA) which applies the pressure distribution on the blade surface calculated by computational fluid dynamics (CFD) as a major boundary condition. Such an approach can be seen as a one-way coupled simulation of the fluid structure interaction (FSI) problem. In this analysis the reverse influence of the deformation on the fluid is generally neglected. Especially in axial machines the blade deformation can result in a significant impact on the turbine performance. The present paper analyzes this influence by means of fully two-way coupled FSI simulations of a propeller turbine utilizing two different approaches. The configuration has been simulated by coupling the two commercial solvers ANSYS CFX for the fluid mechanical simulation with ANSYS Classic for the structure mechanical simulation. A detailed comparison of the results for various blade stiffness by means of changing Young's Modulus are presented. The influence of the blade deformation on the runner discharge and performance will be discussed and shows for the configuration investigated no significant influence under normal structural conditions. This study also highlights that a two-way coupled fluid structure interaction simulation of a real engineering configuration is still a challenging task for today's commercially available simulation tools.	Two-way coupling, fluid structure interaction, propeller turbine, CFD, FEA, FSI, gap vortex



352-359	Performance Optimization of High Specific Speed Pump-Turbines by Means of Numerical Flow Simulation (CFD) and Model Testing	Peter Kerschberger, Arno Gehrler	In recent years, the market has shown increasing interest in pump-turbines. The prompt availability of pumped storage plants and the benefits to the power system achieved by peak lopping, providing reserve capacity, and rapid response in frequency control are providing a growing advantage. In this context, there is a need to develop pump-turbines that can reliably withstand dynamic operation modes, fast changes of discharge rate by adjusting the variable diffuser vanes, as well as fast changes from pumping to turbine operation. In the first part of the present study, various flow patterns linked to operation of a pump-turbine system are discussed. In this context, pump and turbine modes are presented separately and different load cases are shown in each operating mode. In order to create modern, competitive pump-turbine designs, this study further explains what design challenges should be considered in defining the geometry of a pump-turbine impeller. The second part of the paper describes an innovative, staggered approach to impeller development, applied to a low head pump-turbine project. The first level of the process consists of optimization strategies based on evolutionary algorithms together with 3D in-viscid flow analysis. In the next stage, the hydraulic behavior of both pump mode and turbine mode is evaluated by solving the full 3D Navier-Stokes equations in combination with a robust turbulence model. Finally, the progress in hydraulic design is demonstrated by model test results that show a significant improvement in hydraulic performance compared to an existing reference design.	pump-turbine, CFD, multi-objective optimizer, 3D-Euler, 3D-RANS, model testing	
360-368	Study of Flow Field and Pressure Distribution on a Rotor Blade of HAWT in Yawed Flow Conditions	Takao Maeda, Yasunari Kamada, Naohiro Okada, Jun Suzuki	This paper describes the flow field and the blade pressure distribution of a horizontal axis wind turbine in various yawed flow conditions. These measurements were carried out with 2.4m-diameter rotor with pressure sensors and a 2-dimensional laser Doppler velocimeter for each azimuth angle in a wind tunnel. The results show that aerodynamic forces of the blade based on the pressure measurements change according to the local angle of attack during rotation. Therefore the wake of the yawed rotor becomes asymmetric for the rotor axis. Furthermore, the relations between aerodynamic forces and azimuth angles change according to tip speed ratio. By the experimental analysis, the flow field and the aerodynamic forces for each azimuth angle in yawed flow condition were clarified.	Wind Turbine, Rotor Aerodynamics, Fluid Machinery, Yaw, Flow Field, Pressure Distribution	
369-378	Flow Factor Prediction of Centrifugal Hydraulic Turbine for Sea Water Reverse Osmosis (SWRO)	Ying Ma, Eric Kadaj, Kevin Terrasi	The creation of the hydraulic turbine flow factor map will undoubtedly benefit its design by decreasing both the design cycle time and product cost. In this paper, the geometry and flow variables, which effectively affect the flow factor, are proposed, analyzed and determined. These flow variables are further used to create the operating condition maps by using different model approaches categorized into Response Surface Method (RSM) and Artificial Neural Network (ANN). The accuracies of models created by different approaches are compared and the performances of model approaches are analyzed. The influences of chosen variables and the combination of Principle Component Analysis (PCA) and model approaches are also studied. The comparison results between predicted and actual flow factors suggest that two-hidden-layer Feed-forward Neural Network (FFNN), and one-hidden-layer FFNN with PCA has the best performance on forming this mapping, and are accurate sufficiently for hydraulic turbine design.	sea water reverse osmosis, hydraulic turbine, flow factor, flow coefficient	
379-385	Improvement in Efficiency and Operating Range of Centrifugal Blower Stage for Sewage Aeration Blower	Kiyotaka Hiradate, Toshio Kanno, Hideo Nishida, Yasushi Shinkawa, Satoshi Joukou	We developed a high-efficiency, wide-operating-range centrifugal blower stage to meet the demand for reduced total energy-consumption in sewage treatment plants. We improved the efficiency of the two-dimensional impeller using a shape optimization tool and one-dimensional performance prediction tool. A limit of the throat deceleration ratio was set to maintain the stall-margin of the impeller. The low solidity vane diffuser and return channel were designed using a sensitivity analysis with orthogonal arrays and three-dimensional steady flow simulations. The low solidity diffuser was designed in order to improve the performance in the low-flow-rate region. The return channel was designed so that the total pressure loss in the return channel was minimized. Model tests of both the conventional and optimized blower stages were carried out, and the efficiency and operating range of both stages were compared. The optimized blower stage improved in stage efficiency by 3% and in operating range by 5% compared with the conventional blower stage.	Centrifugal Blower, Orthogonal Array, Numerical Simulation, Efficiency, Operating Range	
386-395	Study of the Adhesive Coefficient Effect on the Hydraulic Losses and Cavitation	František Pochylý, Simona Fialová, Milada Kozubková, Lukáš Zavadil	The article is focused in three areas. In the first part there are analyzed the adhesion forces at the liquid and solid surface interface. There are shown the measured values of surface energy for different types of surfaces. The value of surface energy is decisive for determining the extent of the surface wettability by the liquid. The second part points to the possible negative effects of partly wettable surfaces, showing susceptibility to cavitation. The third section describes the practical aspects of surface wettability by the liquid. Under the new boundary conditions bases, expressing the effect of adhesion forces, there are determined the centrifugal pump characteristics.	cavitation, wettability, adhesion force, impeller pump	
	Page	Title	Author	Abstract	Keywords
1-13	J-Groove Technique for Suppressing Various Anomalous Flow Phenomena in Turbomachines	Junichi Kurokawa	In operating a turbomachine at off-design conditions various instabilities caused by anomalous flow phenomena occur and sometimes lead to the damage of a turbomachine. In order to avoid these phenomena various devices characteristic to each phenomenon have been developed, however they make turbomachines large-sized and cause efficiency drop. The present author has developed a very simple and innovative device, termed "J-groove," of suppressing various anomalous flow phenomena commonly by controlling the angular momentum of the main flow. It has been revealed that J-groove makes an operation of a turbomachine stable in all flow range, causes little efficiency drop, and can be easily applied to an existing machine. Here is reviewed totally the results of suppressing various anomalous flow phenomena in turbomachines.	J-Groove, Anomalous Flow Phenomenon, Surge, Cavitation, Performance-curve Instability	
14-24	Design Optimization of Mixed-flow Pump in a Fixed Meridional Shape	Sung Kim, Young-Seok Choi, Kyoung-Yong Lee, Jun-Ho Kim	In this paper, design optimization for mixed-flow pump impellers and diffusers has been studied using a commercial computational fluid dynamics (CFD) code and DOE (design of experiments). We also discussed how to improve the performance of the mixed-flow pump by designing the impeller and diffuser. Geometric design variables were defined by the vane plane development, which indicates the blade-angle distributions and length of the impeller and diffusers. The vane plane development was controlled using the blade-angle in a fixed meridional shape. First, the design optimization of the defined impeller geometric variables was achieved, and then the flow characteristics were analyzed in the point of incidence angle at the diffuser leading edge for the optimized impeller. Next, design optimizations of the defined diffuser shape variables were performed. The importance of the geometric design variables was analyzed using 2k factorial designs, and the design optimization of the geometric variables was determined using the response surface method (RSM). The objective functions were defined as the total head and the total efficiency at the design flow rate. Based on the comparison of CFD results between the optimized pump and base design models, the reason for the performance improvement was discussed.	Mixed-flow Pump, Impeller, Diffuser, Optimal Design, CFD (computational fluid dynamics), DOE (design of experiments)	
25-32	Numerical Study of Inlet and Impeller Flow Structures in Centrifugal Pump at Design and Off-design Points	Kean Wee Cheah, Thong See Lee, Winoto S.H	The objective of present work is to use numerical simulation to investigate the complex three-dimensional and secondary flow structures developed at the inlet and impeller in a centrifugal pump at design and off-design points. The pump impeller is shrouded with 6 backward swept blades and with a specific speed of 0.8574. The characteristic of the pump is measured experimentally with straight and curved intake sections. Numerical computation is carried out to investigate the pump inlet flow structures and subsequently the flow field within the centrifugal pump. The numerical results showed that strong interaction between the impeller eye and intake section. Secondary flow structure occurs upstream at the pump inlet has great influence on the pump performance and flow structure within the impeller.	centrifugal pump, impeller, inlet flow structure, secondary flow	

33-46	Concave Surface Boundary Layer Flows in the Presence of Streamwise Vortices	Sonny H. Winoto, Tandiono, Dilip A. Shah, Hatsari Mitsudharmadi	Concave surface boundary-layer flows are subjected to centrifugal instability which results in the formation of streamwise counter-rotating vortices. Such boundary layer flows have been experimentally investigated on concave surfaces of 1 m and 2 m radius of curvature. In the experiments, to obtain uniform vortex wavelengths, thin perturbation wires placed upstream and perpendicular to the concave surface leading edge, were used to pre-set the wavelengths. Velocity contours were obtained from hot-wire anemometer velocity measurements. The most amplified vortex wavelengths can be pre-set by the spanwise spacing of the thin wires and the free-stream velocity. The velocity contours on the cross-sectional planes at several streamwise locations show the growth and breakdown of the vortices. Three different vortex growth regions can be identified. The occurrence of a secondary instability mode is also shown as mushroom-like structures as a consequence of the non-linear growth of the streamwise vortices. Wall shear stress measurements on concave surface of 1 m radius of curvature reveal that the spanwise-averaged wall shear stress increases well beyond the flat plate boundary layer values. By pre-setting much larger or much smaller vortex wavelength than the most amplified one, the splitting or merging of the streamwise vortices will respectively occur.	Görtler vortices, concave surface, boundary layer, instability, wall shear stress, hot-wire anemometer
47-56	Flow-Feedback for Pressure Fluctuation Mitigation and Pressure Recovery Improvement in a Conical Diffuser with Swirl	Constantin Tanasa, Alin Bosioc, Romeo Susan-Resiga, Sebastian Muntean	Our previous experimental and numerical investigations of decelerated swirling flows in conical diffusers have demonstrated that water jet injection along the symmetry axis mitigates the pressure fluctuations associated with the precessing vortex rope. However, for swirling flows similar to Francis turbines operated at partial discharge, the jet becomes effective when the jet discharge is larger than 10% from the turbine discharge, leading to large volumetric losses when the jet is supplied from upstream the runner. As a result, we introduce the flow-feedback approach for supplying the jet by using a fraction of the discharge collected downstream the conical diffuser. Experimental investigations on mitigating the pressure fluctuations generated by the precessing vortex rope and investigations of pressure recovery coefficient on the cone wall with and without flow-feedback method are presented.	Francis turbine, vortex rope, pressure fluctuations, pressure recovery coefficient, flow-feedback
57-66	A Suggested Mechanism of Significant Stall Suppression Effects by Air Separator Devices in Axial Flow Fans	Nobuyuki Yamaguchi	Radial-vaned air separators show a strong stall suppression effect in an axial flow fans. From a survey of existing literature on the effects and the author's data, a possible mechanism for the significant effects has been proposed here. The stall suppression is suggested to have been achieved by a combination of the following several effects; (1) suction of blade and casing boundary layers and elimination of embryos of stall, (2) separation and straightening of reversed swirling flow from the main flow, (3) induction of the fan main flow toward the casing wall and enhancement of the outward inclination of meridional streamlines across the rotor blade row, thus keeping the Euler head increase in the decrease in fan flow rate, and (4) reinforcement of axisymmetric structure of the main flow. These phenomena have been induced and enhanced by a stable vortex-ring encasing the blade tips and the air separator. These integrated effects appear to have caused the great stall suppression effect that would have been impossible by other types of stall prevention devices. Thus the author would like to name the device "tip-vortex-ring assisted stall suppression device".	Axial Flow Fan, Stall Suppression, Air Separator, Vortex Ring, Internal Flow
67-75	Leakage-free Rotating Seal Systems with Magnetic Nanofluids and Magnetic Composite Fluids Designed for Various Applications	Tünde Borbáth, Doina Bica, Iosif Potencz, István Borbáth, Tibor Boros, Ladislau Vékás	Recent results are presented concerning the development of magnetofluidic leakage-free rotating seals for vacuum and high pressure gases, evidencing significant advantages compared to mechanical seals. The micro-pilot scale production of various types of magnetizable sealing fluids is shortly reviewed, in particular the main steps of the chemical synthesis of magnetic nanofluids and magnetic composite fluids with light hydrocarbon, mineral oil and synthetic oil carrier liquids. Design concepts and some constructive details of the magnetofluidic seals are discussed in order to obtain high sealing capacity. Different types of magnetofluidic sealing systems and applications are reviewed. Testing procedures and equipment are presented, as well as the sealing capabilities of different types of magnetizable fluids.	Rotating Seal, Magnetic Nanofluids, Magnetic Composite Fluids, Gas Valves, Testing Procedures, Magnetofluidic Applications
76-83	Vortex Cavitation from Baffle Plate and Pump Vibration in a Double-Suction Volute Pump	Toshiyuki Sato, Takahide Nagahara, Kazuhiro Tanaka, Masaki Fuchiwaki, Fumio Shimizu, Akira Inoue	This study highlights especially the mechanism of vortex cavitation occurrence from the end of the suction duct in a double-suction volute pump and pump oscillation which causes cavitation noise from the pump. In this study, full 3D numerical simulations have been performed using a commercial code inside the pump from the inlet of suction duct to the outlet of delivery duct. The numerical model is based on a combination of multiphase flow equations with the truncated version of the Rayleigh-Plesset model predicting the complicated growth and collapse process of cavity bubbles. The experimental investigations have also been performed on the cavitating flow with flow visualization to evaluate the numerical results.	Vortex Cavitation, Cavitation Noise, Double-Suction Volute Pump, CFD, Pump Oscillation
85-96	Steady and unsteady flow computation in an elbow draft tube with experimental validation	Thi C. Vu, Christophe Devals, Ying Zhang, Bernd Nennemann, François Guibault	Steady state computations are routinely used by design engineers to evaluate and compare losses in hydraulic components. In the case of the draft tube diffuser, however, experiments have shown that while a significant number of operating conditions can adequately be evaluated using steady state computations, a few operating conditions require unsteady simulations to accurately evaluate losses. This paper presents a study that assesses the predictive capacity of a combination of steady and unsteady RANS numerical computations to predict draft tube losses over the complete range of operation of a Francis turbine. For the prediction of the draft tube performance using k-ε turbulence model, a methodology has been proposed to average global performance indicators of steady flow computations such as the pressure recovery factor over an adequate number of periods to obtain correct results. The methodology will be validated using two distinct flow solvers, CFX and OpenFOAM, and through a systematic comparison with experimental results obtained on the FLINDT model draft tube.	Hydraulic turbine, draft tube, steady flow simulation, unsteady flow simulation
97-103	Numerical Study of Passive Control with Slotted Blading in Highly Loaded Compressor Cascade at Low Mach Number	Mdoui Ramzi, Gérard Bois, Gahmousse Abderrahmane	With the aim to increase blade loadings and stable operating range in highly loaded compressors, this article has been conducted to explore, through a numerical parametric study, the potential of passive control using slotted bladings in cascade configurations. The objective of this numerical investigation is to analyze the influence of location, width and slope of the slots and therefore identify the optimal configuration. The approach is based on two dimensional cascade geometry, low speed regime, steady state and turbulent RANS model. The results show the efficiency of this passive technique to delay separation and enhance aerodynamic performances of the compressor cascade. A maximum of 28.3% reduction in loss coefficient have been reached, the flow turning is increased with approximately 5° and high loading over a wide range of angle of attack have been obtained for the optimized control parameter.	2D cascade, Highly loaded compressor, Separation, Passive control, Slotted blading, Low Mach number
104-113	A New Approach in Numerical Assessment of the Cavitation Behaviour of Centrifugal Pumps	Adrian Stuparu, Romeo Susan-Resiga, Liviu Eugen Anton, Sebastian Muntean	The paper presents a new method for the analysis of the cavitation behaviour of hydraulic turbomachines. This new method allows determining the coefficient of the cavitation inception and the cavitation sensitivity of the turbomachines. We apply this method to study the cavitation behaviour of a large storage pump. By plotting in semi-logarithmic coordinates the vapour volume versus the cavitation coefficient, we show that all numerical data collapse in an exponential manner. By analysis of the slope of the curve describing the evolution of the vapour volume against the cavitation coefficient we determine the cavitation sensitivity of the pump for each operating point.	cavitation inception, cavitation sensibility, pumping head drop, storage pump

114-132	Theory and Prediction of Turbulent Transition	Hua-Shu Dou, Boo Cheong Khoo	We have proposed a new approach based on energy gradient concept for the study of flow instability and turbulent transition in parallel flows in our previous works. It was shown that the disturbance amplitude required for turbulent transition is inversely proportional to Re, which is in agreement with the experiments for imposed transverse disturbance. In present study, the energy gradient theory is extended to the generalized curved flows which have much application in turbomachinery and other fluid delivery devices. Within the frame of the new theory, basic theorems for flow instability in general cases are provided in details. Examples of applications of the theory are given from our previous studies which show comparison of the theory with available experimental data. It is shown that excellent agreement has been achieved for several configurations. Finally, various prediction methods for turbulent transition are reviewed and commented.	Turbulent transition, Mechanism, Theory, Theorem, Prediction
133-139	Numerical Design and Performance Prediction of Low Specific Speed Centrifugal Pump Impeller	Zhang Yongxue, Zhou Xin, Ji Zhongli, Jiang Cuiwei	In this paper, Based on Two-dimensional Flow Theory, adopting quasi-orthogonal method and point-by-point integration method to design the impeller of the low specific speed centrifugal pump by code, and using RANS (Reynolds Averaged N-S) Equation with a standard k-ε two-equation turbulence model and log-law wall function to solve 3D turbulent flow field in the impeller of the low specific speed pump. An analysis of the influences of the blade profile on velocity distributions, pressure distributions and pump performance and the investigation of the flow regulation pattern in the impeller of the centrifugal pump are presented. And the result shows that this method can be used as a new way in low speed centrifugal pump impeller design.	Low Specific Speed, Centrifugal Pump impeller, Two-dimensional Flow Theory, Numerical Design, Performance Prediction
140-149	Incipient Cavitation in a Bulb Turbine: Model Test and CFD Calculation	Jörg Necker, Thomas Aschenbrenner	For a certain operating point of a horizontal shaft bulb turbine (i.e. volume flow, net head, blade angle, guide vane angle) the efficiency for different pressure levels (i.e. different Thoma-coefficient $\sigma$ ) is calculated using a commercial Computational Fluid Dynamics (CFD)-code including two-phase flow and a cavitation model. The results are compared with experimental results achieved at a closed loop test rig for model turbines. The comparison of the experimentally and numerically obtained efficiency and the visual impression of the cavitation show a good agreement. Especially the drop in efficiency is calculated with satisfying accuracy. This drop in efficiency in combination with the visual impression is of high practical importance since it contributes to determine the admissible cavitation in a bulb-turbine. It is seen that the incipient cavitation in Kaplan type turbines has no major importance in determining this admissible amount of cavitation.	Bulb turbines, multi-phase cfd, incipient cavitation, admissible cavitation
150-160	Efficiency of Marine Hydropower Farms Consisting of Multiple Vertical Axis Cross-Flow Turbines	Andrei-Mugur Georgescu, Sanda-Carmen Georgescu, Costin Ioan Cosoiu, Nicolae Alboiu	This study focuses on the Achard turbine, a vertical axis, cross-flow, marine current turbine module. Similar modules can be superposed to form towers. A marine or river hydropower farm consists of a cluster of barges, each gathering several parallel rows of towers, running in stabilized current. Two-dimensional numerical modelling is performed in a horizontal cross-section of all towers, using FLUENT and COMSOL Multiphysics. Numerical models validation with experimental results is performed through the velocity distribution, depicted by Acoustic Doppler Velocimetry, in the wake of the middle turbine within a farm model. As long as the numerical flow in the wake fits the experiments, the numerical results for the power coefficient (turbine efficiency) are trustworthy. The overall farm efficiency, with respect to the spatial arrangement of the towers, was depicted by 2D modelling of the unsteady flow inside the farm, using COMSOL Multiphysics. Rows of overlapping parallel towers ensure the increase of global efficiency of the farm.	Achard turbine, cross-flow current turbine, hydropower farm, farm efficiency, force coefficient, power coefficient
161-171	Steady-state Capabilities for Hydroturbines with OpenFOAM	Maryse Page, Martin Beaudoin, Anne-Marie Giroux	The availability of a high quality open source CFD simulation platform like OpenFOAM offers new R&D opportunities by providing direct access to models and solver implementation details. Efforts have been made by Hydro-Québec to adapt OpenFOAM to hydroturbines for the development of steady-state capabilities. This paper describes the developments that have been made to implement new turbomachinery related capabilities: multiple frames of reference solver, domain coupling interfaces (GGI, cyclicGGI and mixing plane) and specialized boundary conditions. Practical use of the new turbomachinery capabilities are demonstrated for the analysis of a 195-MW Francis turbine.	OpenFOAM, hydroturbine, GGI interface, mixing plane interface, multiple frames of reference
172-178	Optimization of Vane Diffuser in a Mixed-Flow Pump for High Efficiency Design	Jin-Hyuk Kim, Kwang-Yong Kim	This paper presents an optimization procedure for high-efficiency design of a mixed-flow pump. Optimization techniques based on a weighted-average surrogate model are used to optimize a vane diffuser of a mixed-flow pump. Validation of the numerical results is performed through experimental data for head, power and efficiency. Three-level full factorial design is used to generate nine design points within the design space. Three-dimensional Reynolds-averaged Navier-Stokes equations with the shear stress transport turbulence model are discretized by using finite volume approximation and solved on hexahedral grids to evaluate the efficiency as the objective function. In order to reduce pressure loss in the vane diffuser, two variables defining the straight vane length ratio and the diffusion area ratio are selected as design variables in the present optimization. As the results of the design optimization, the efficiency at the design flow coefficient is improved by 7.05% and the off-design efficiencies are also improved in comparison with the reference design.	Mixed-flow pump, Impeller, Vane diffuser, Efficiency, Optimization, Weighted-average surrogate model
179-190	Experimental Investigations on Upper Part Load Vortex Rope Pressure Fluctuations in Francis Turbine Draft Tube	Christophe Nicolet, Amirreza Zobeiri, Pierre Maruzewski, François Avellan	The swirling flow developing in Francis turbine draft tube under part load operation leads to pressure fluctuations usually in the range of 0.2 to 0.4 times the runner rotational frequency resulting from the so-called vortex breakdown. For low cavitation number, the flow features a cavitation vortex rope animated with precession motion. Under given conditions, these pressure fluctuations may lead to undesirable pressure fluctuations in the entire hydraulic system and also produce active power oscillations. For the upper part load range, between 0.7 and 0.85 times the best efficiency discharge, pressure fluctuations may appear in a higher frequency range of 2 to 4 times the runner rotational speed and feature modulations with vortex rope precession. It has been pointed out that for this particular operating point, the vortex rope features elliptical cross section and is animated of a self-rotation. This paper presents an experimental investigation focusing on this peculiar phenomenon, defined as the upper part load vortex rope. The experimental investigation is carried out on a high specific speed Francis turbine scale model installed on a test rig of the EPFL Laboratory for Hydraulic Machines. The selected operating point corresponds to a discharge of 0.83 times the best efficiency discharge. Observations of the cavitation vortex carried out with high speed camera have been recorded and synchronized with pressure fluctuations measurements at the draft tube cone. First, the vortex rope self rotation frequency is evidenced and the related frequency is deduced. Then, the influence of the sigma cavitation number on vortex rope shape and pressure fluctuations is presented. The waterfall diagram of the pressure fluctuations evidences resonance effects with the hydraulic circuit. The influence of outlet bubble cavitation and air injection is also investigated for low cavitation number. The	Francis turbine, cavitating vortex rope, High Speed Camera visualization, pressure fluctuations
191-198	Cavitation in a Shaft-less Double Suction Centrifugal Miniature Pump	Baotang Zhuang, Xianwu Luo, Lei Zhu, Xin Wang, Hongyuan Xu	Based on the consideration that the cavitation would affect the operation stability of miniature pumps, the 3-D turbulent cavitating flow in a test pump was simulated by using a mixed cavitation model and k-ω SST turbulence model. In order to investigate the influence of inlet geometry parameters on the cavitation performance of the miniature pump, two more impellers are designed for comparison. Based on the results, the following conclusions are drawn: 1) Cavitation performance of the double suction shaft-less miniature pump having different impeller is equivalent to the centrifugal pump having ordinary size, though the flow passage at impeller inlet is small; 2) The miniature pump having radial impeller can produce much higher pump head, but lower cavitation performance than that having the impeller based on the conventional design method; 3) It is believed that by applying the double suction design, the miniature pump achieved relatively uniform flow pattern upstream the impeller inlet, which is favorable for improving cavitation performance.	double suction, shaft-less, miniature pump, cavitation, numerical simulation

	199-208	Unsteady Simulations of the Flow in a Swirl Generator, Using OpenFOAM	Olivier Petit, Alin I. Bosioc, Håkan Nilsson, Sebastian Muntean, Romeo F. Susan-Resiga	This work presents numerical results, using OpenFOAM, of the flow in the swirl flow generator test rig developed at Politehnica University of Timisoara, Romania. The work shows results computed by solving the unsteady Reynolds Averaged Navier Stokes equations. The unsteady method couples the rotating and stationary parts using a sliding grid interface based on a GGI formulation. Turbulence is modeled using the standard k-ε model, and block structured wall function ICEM-Hexa meshes are used. The numerical results are validated against experimental LDV results, and against design velocity profiles. The investigation shows that OpenFOAM gives results that are comparable to the experimental and design profiles. The unsteady pressure fluctuations at four different positions in the draft tube is recorded. A Fourier analysis of the numerical results is compared with that of the experimental values. The amplitude and frequency predicted by the numerical simulation are comparable to those given by the experimental results, though slightly over estimated.	Swirl generator, OpenFOAM, CFD, Validation, Runner, Draft tube, Rotor-Stator Interaction
	209-216	Investigation on the Flow Field Upstream of a Centrifugal Pump Impeller	Yao Zhang, Xianwu Luo, Yunchi Yi, Baotang Zhuang, Hongyuan Xu	The flow upstream of a centrifugal pump impeller has been investigated by both experimental test and numerical simulation. For experimental study, the flow field at four sections in the pump suction is measured by using PIV method. For calculation, the three dimensional turbulent flow for the full flow passage of the pump is simulated based on RANS equations combined with RNG k-ε turbulence model. From those results, it is noted that at both design load and quarter load condition, the pre-swirl flow whose direction is the same as the impeller rotation exists at all four sections in suction pipe of the pump, and at each section, the pre-swirl velocity becomes obviously larger at higher rotational speed. It is also indicated that at quarter load condition, the low pressure region at suction surface of the vane is large because of the unfavorable flow upstream of the pump impeller.	centrifugal pump, flow upstream impeller inlet, PIV, numerical simulation
	Page	Title	Author	Abstract	Keywords
Vol. 4, No. 2, April-June, 2011	217-222	The Effect of Casing Geometry on Rotordynamic Fluid Forces on a Closed Type Centrifugal Impeller in Whirling Motion	Julien Richert, Yumeto Nishiyama, Shinichiro Hata, Hironori Horiguchi, Yoshinobu Tsujimoto	The rotordynamic fluid forces acting on a closed type impeller in whirling motion were measured and the influence of the clearance geometry on the stability of the impeller was examined. At small positive whirling speed, the rotordynamic forces acted as destabilizing forces for all casings. A small clearance between the shroud of the impeller and the casing caused large fluid force, but did not change the destabilizing region. Radial grooves in the clearance were effective for reducing the fluid forces and destabilizing region due to the reduction of the circumferential velocity without the deterioration of the pump performance. A rotating phenomenon like a rotating stall of the impeller occurred at low flow rate and the resonance between it and the whirling motion led to a sudden increase in force at the whirling speed ratio of 0.7.	Pump, Rotordynamic Fluid Force, Whirling Motion, Unsteady Phenomenon
	223-228	Study on the Performance Characteristics of Centrifugal Pump with Drag-reducing Surfactant Additives	Lu Wang, Feng-Chen Li, Yong Dong, Wei-Hua Cai, Wen-Tao Su	The performance characteristics of centrifugal pump were measured experimentally when running with tap water and drag-reducing surfactant (Octadecyl dimethyl amine oxide (OB-8)) solutions. Tests have been performed on five cases of surfactant solutions with different concentrations (0ppm (tap water), 200ppm, 500ppm, 900ppm and 1500ppm) and four different rotating speeds of pump (1500rpm, 2000rpm, 2500rpm and 2900rpm). Compared with tap water case, the experimental results show that the total pump heads for surfactant solution cases are higher. And the pump efficiency with surfactant solutions also increases, but the shaft power for surfactant solutions cases decreases compared to that for tap water. There exists an optimal temperature for surfactant solutions, which maximizes the pump efficiency.	centrifugal pump, surfactant solutions, drag reduction
	229-234	Influence of Reynolds Number and Scale on Performance Evaluation of Lift-type Vertical Axis Wind Turbine by Scale-model Wind Tunnel Tests	Tadakazu Tanino, Shinichiro Nakao, Takeshi Miyaguni, Kazunobu Takahashi	For Lift-type Vertical Axis Wind Turbine (VAWT), it is difficult to evaluate the performance through the scale-model wind tunnel tests, because of the scale effect relating to Reynolds number. However, it is beneficial to figure out the critical value of Reynolds number or minimum size of the Lift-type VAWT, when designing this type of micro wind turbine. Therefore, in this study, the performance of several scale-models of Lift-type VAWT (Reynolds number : 1.5x10 <sup>4</sup> to 4.6x10 <sup>4</sup> ) was investigated. As a result, the Reynolds number effect depends on the blade chord rather than the inlet velocity. In addition, there was a transition point of the Reynolds number to change the dominant driving force from Drag to Lift.	Vertical Axis Wind Turbine, Wind tunnel, Reynolds number, Performance, Scale effect
	235-242	Internal Flow and Limiting Streamlines Observations of Contra-Rotating Axial Flow Pump at Partial Flow Rate	Satoshi Watanabe, Shimpei Momosaki, Satoshi Usami, Akinori Furukawa	An application of contra-rotating rotors, in which a rear rotor is in tandem with a front one and these rotors rotate in the opposite direction each other, has been proposed against a demand for developing higher specific speed axial flow pump. One prototype rotors, which we have designed with a conventional method, has given the positive slope of head characteristic curve especially in the rear rotor. It is necessary to understand the internal flow behavior in the rear rotor to establish the design guideline for achieving higher and more reliable performance. In the present study, we carried out the experimental investigations of the internal flow field of the rear rotor, especially at the partial flow rate, by Laser Doppler Velocimetry (LDV) for the main flow and the limiting streamlines observation on rotor surfaces for the boundary layer flows.	Axial flow pump, Contra-rotating rotors, Partial flow rate, LDV, Limiting streamlines
	243-254	Flow simulation and efficiency hill chart prediction for a Propeller turbine	Thi Vu, Marcel Koller, Maxime Gauthier, Claire Deschênes	In the present paper, we focus on the flow computation of a low head Propeller turbine at a wide range of design and off-design operating conditions. First, we will present the results on the efficiency hill chart prediction of the Propeller turbine and discuss the consequences of using non-homologous blade geometries for the CFD simulation. The flow characteristics of the entire turbine will be also investigated and compared with experimental data at different measurement planes. Two operating conditions are selected, the first one at the best efficiency point and the second one at part load condition. At the same time, for the same selected operating points, the numerical results for the entire turbine simulation will be compared with flow simulation with our standard stage calculation approach which includes only guide vane, runner and draft tube geometries.	Propeller Turbine, CFD, Flow Simulation, Performance prediction, Non-homologous geometry, Draft Tube
	255-261	Effects of the Air Volume in the Air Chamber on the Performance of Water Hammer Pump System	Sumio Saito, Masaaki Takahashi, Yoshimi Nagata	Recently, as global-scale problems, such as global warming and energy depletion, have attracted attention, the importance of future environmental preservation has been emphasized worldwide, and various measures have been proposed and implemented. This study focuses on water hammer pumps that can effectively use the water hammer phenomenon and allow fluid transport without drive sources, such as electric motors. An understanding of operating conditions of water hammer pumps and an evaluation of their basic hydrodynamic characteristics are significant for determining whether they can be widely used as an energy-saving device in the future. However, conventional studies have not described the pump performance in terms of pump head and flow rate, common measures indicating the performance of pumps. As a first stage for the understanding of water hammer pump performance in comparison to the characteristics of typical turbo pumps, the previous study focused on understanding the basic hydrodynamic characteristics of water hammer pumps and experimentally examined how the hydrodynamic characteristics were affected by the inner diameters of the drive and lift pipes and the angle of the drive pipe. This paper suggests the effect of the air volume in the air chamber that affects the hydrodynamic characteristics and operating conditions of the water hammer pump.	Water Hammer Pump, Fluid Transients, Pump Performance, Pressure Fluctuation, Flow Visualization

262-270	Vortices within a Three-Dimensional Separation in an Axial Flow Stator of a Diagonal Flow Fan	Yoichi Kinoue, Norimasa Shiomi, Toshiaki Setoguchi	Experimental and numerical investigations were conducted for an internal flow in an axial flow stator of a diagonal flow fan. A corner separation near the hub surface and the suction surface of a stator blade was focused on, and further, three-dimensional vortices in separated flow were investigated by the numerical analysis. At low flow rate of 80% of the design flow rate, a corner separation of the stator between the suction surface and the hub surface can be found in both experimental and calculated results. Separation vortices are observed in the limiting streamline patterns both on the blade suction and on the hub surfaces at 80% of the design flow rate in the calculated results. It also can be observed in the streamline pattern that both vortices from the blade suction surface and from the hub surface keep vortex structures up to far locations from these wall surfaces. An attempt to explain the vortices within a three-dimensional separation is introduced by using vortex filaments.	Diagonal flow fan, Axial flow stator, Internal flow, Corner separation, Five-hole probe survey, CFD	
271-288	Review of Mathematical Models in Performance Calculation of Screw Compressors	Nikola Stosic, Ian K. Smith, Ahmed Kovacevic, Elvedin Mujic	The mathematical modelling of screw compressor processes and its implementation in their design began about 30 years ago with the publication of several pioneering papers on this topic, mainly at Purdue Compressor Conferences. This led to the gradual introduction of computer aided design, which, in turn, resulted in huge improvements in these machines, especially in oil-flooded air compressors, where the market is very competitive. A review of progress in such methods is presented in this paper together with their application in successful compressor designs. As a result of their introduction, even small details are now considered significant in efforts to improve performance and reduce costs. Despite this, there are still possibilities to introduce new methods and procedures for improved rotor profiles, design optimisation for each specified duty and specialized compressor design, all of which can lead to a better product and new areas of application. A review of methods and procedures which lead to modern screw compressor practice is presented in this paper. This paper is intended to give a cross section through activities being done in mathematical modelling of screw compressor process through last five decades. It is expected to serve as a basis for further contributions in the area and as a challenge to the forthcoming generations of scientists and engineers to concentrate their efforts in finding future and more extended approaches and submit their contributions.	Screw Compressor, Mathematical Model, Performance Calculation	
289-306	Vibration Behavior and Dynamic Stress of Runners of Very High Head Reversible Pump-turbines	Hiroshi Tanaka	In the development of very high head pumped storage projects, one of the critical problems is the strength of pump-turbine runners. Data obtained by stress measurements of high head pump-turbine runners indicated that dynamic stress due to the vibration of runner might be detrimental, possibly to cause fatigue failure, if the runner were designed without proper consideration on its dynamic behaviour. Numerous field stress measurements of runners and model tests conducted with hydrodynamic similarity revealed that the hydraulic excitation force developed by the interference of rotating runner blades with guide vane wakes sometimes would induce such heavy vibration of runner. Theoretical and experimental investigations on both the hydraulic excitation force and the natural frequencies of runner have been conducted to explore this forced vibration problem.	Vibration behavior, Dynamic stress, Hydraulic excitation force, Vibration mode, Prototype head model test	
	Page	Title	Author	Abstract	Keywords
307-316	Vortex Features in a Half-ducted Axial Fan with Large Bellmouth (Effect of Tip Clearance)	Norimasa Shiomi, Yoichi Kinoue, Toshiaki Setoguchi, Kenji Kaneko	In order to clarify the features of tip leakage vortex near blade tip region in a half-ducted axial fan with large bellmouth, the experimental investigation was carried out using a 2-dimensional LDV system. Three sizes of tip clearance (TC) were tested: those sizes were 1mm (0.55% of blade chord length at blade tip), 2mm (1.11% of blade chord length at blade tip) and 4mm (2.22% of blade chord length at blade tip), and those were shown as TC=1mm, TC=2mm and TC=4mm, respectively. Fan characteristic tests and the velocity field measurements were done for each TC. Pressure - flow-rate characteristics and two-dimensional velocity vector maps were shown. The vortex trace and the vortex intensity distribution were also illustrated. As a result, a large difference on the pressure - flow-rate characteristics did not exist for three tip clearance sizes. In case of TC=4mm, the tip leakage vortex was outflow to downstream of rotor was not confirmed at the small and reference flow-rate conditions. Only at the large flow-rate condition, its outflow to downstream of rotor existed. In case of TC=2mm, overall vortex behaviors were almost the same ones in case of TC=4mm. However, the vortex trace inclined toward more tangential direction. In case of TC=1mm, the clear vortex was not observed for all flow-rate conditions.	Tip leakage vortex, Tip clearance, Ventilation fan, LDV measurement	
317-323	Influence of Blade Outlet Angle and Blade Thickness on Performance and Internal Flow Conditions of Mini Centrifugal Pump	Toru Shigemitsu, Junichiro Fukutomi, Kensuke Kaji	Mini centrifugal pumps having a diameter smaller than 100mm are employed in many fields; automobile radiator pump, ventricular assist pump, cooling pump for electric devices and so on. Further, the needs for mini centrifugal pumps would become larger with the increase of the application of it for electrical machines. It is desirable that the mini centrifugal pump design be as simple as possible as precise manufacturing is required. But the design method for the mini centrifugal pump is not established because the internal flow condition for these small-sized fluid machines is not clarified and conventional theory is not suitable for small-sized pumps. Therefore, we started research on the mini centrifugal pump for the purpose of development of high performance mini centrifugal pumps with simple structure. Three types of rotors with different outlet angles are prepared for an experiment. The performance tests are conducted with these rotors in order to investigate the effect of the outlet angle on performance and internal flow condition of mini centrifugal pumps. In addition to that, the blade thickness is changed because blockage effect in the mini centrifugal pump becomes relatively larger than that of conventional pumps. On the other hand, a three dimensional steady numerical flow analysis is conducted with the commercial code (ANSYS-Fluent) to investigate the internal flow condition. It is clarified from the experimental results that head of the mini centrifugal pump increases according to the increase of the blade outlet angle and the decrease of the blade thickness. In the present paper, the performance of the mini centrifugal pump is shown and the internal flow condition is clarified with the results of the experiment and the numerical flow analysis. Furthermore, the effects of the blade outlet angle and the blade thickness on the performance are investigated and the internal	Mini centrifugal pump, Performance, Internal flow condition, Blade outlet angle, Blade thickness	
324-333	Phase Resonance in a Centrifugal Compressor	Yumeto Nishiyama, Takayuki Suzuki, Koichi Yonezawa, Hiroshi Tanaka, Peter Doerfler, Yoshinobu Tsujimoto	Phase resonance in a centrifugal compressor was experimentally observed and simulated with a commercial CFD code. It was found that pressure fluctuation at the volute outlet becomes the maximum when the rotational speed of the modes caused by the rotor-stator interaction agrees with the sound velocity. A simple one-dimensional theory is presented to explain the phase resonance in turbomachinery.	Phase resonance, Rotor-stator interaction, Centrifugal compressor, Volute, Numerical analysis	

334-340	Counter-Rotating Type Pumping Unit (Impeller Speeds in Smart Control)	Toshiaki Kanemoto, Keiichi Komaki, Masaaki Katayama, Makoto Fujimura	Turbo-pumps have weak points, such as the pumping operation is unstable on the positive slope of the head curve and/or the cavitation occurs at the low suction head. To improve simultaneously both weak points, the first author invented the unique pumping unit composed of the tandem impellers and the peculiar motor with the double rotational armatures. The front and the rear impellers are driven by the inner and the outer armatures of the motor, respectively. Both impeller speeds are automatically and smartly adjusted in response to the pumping discharge, while the rotational torques between both impellers/armatures are counter-balanced. Such speeds contribute to suppress successfully not only the unstable operation at the low discharge but also the cavitation at the high discharge, as verified with the axial flow type pumping unit in the previous paper. Continuously, this paper investigates experimentally the effects of the tandem impeller profiles on the pump performances and the rotational speeds against the discharge, using the impellers whose loads are low and/or high at the normal discharge. The worthy remarks are that (a) the unstable operation is suppressed as expected and the shut off power is scarcely large in the smart control, (b) the blade profile contributes to determine the discharge giving the maximum/minimum rotational speed where the reverse flow may incipiently appears at the front impeller inlet, (c) the tandem impeller profiles scarcely affect the rotational speeds, while the loads of the front and the rear impellers are same, but (d) the impeller with the low load must run faster and the impeller with the high load must run slower at the same discharge to take the same rotational torque, and (e) the reverse flow at the inlet and the swirling velocity component at the outlet of the front impeller with the high load require making the rotational speed of	counter-rotation, tandem impellers, pump, armature, smart control, performance, rotational speed	
341-348	Computational Fluid Dynamics Modeling Studies on Bacterial Flagellar Motion	Manickam Siva Kumar, Pichai Philominathan	The study of bacterial flagellar swimming motion remains an interesting and challenging research subject in the fields of hydrodynamics and bio-locomotion. This swimming motion is characterized by very low Reynolds numbers, which is unique and time reversible. In particular, the effect of rotation of helical flagella of bacterium on swimming motion requires detailed multi-disciplinary analysis. Clear understanding of such swimming motion will not only be beneficial for biologists but also to engineers interested in developing nanorobots mimicking bacterial swimming. In this paper, computational fluid dynamics (CFD) simulation of a three dimensional single flagellated bacteria has been developed and the fluid flow around the flagellum is investigated. CFD-based modeling studies were conducted to find the variables that affect the forward thrust experienced by the swimming bacterium. It is found that the propulsive force increases with increase in rotational velocity of flagellum and viscosity of surrounding fluid. It is also deduced from the study that the forward force depends on the geometry of helical flagella (directly proportional to square of the helical radius and inversely proportional to pitch).	Flagellar hydrodynamics, helical swimming motion, low Reynolds number, computational fluid dynamics	
349-359	Unsteady Analysis of Impeller-Volute Interaction in Centrifugal Pump	Kean Wee Cheah, Thong See Lee, Sonny H. Winoto	An unsteady numerical analysis has been carried out to study the strong impeller volute interaction of a centrifugal pump with six backward swept blades shrouded impeller. The numerical analysis is done by solving the three-dimensional Reynolds Averaged Navier-Stokes codes with standard k-ε two-equations turbulence model and wall regions are modeled with a scalable log-law wall function. The flow within the impeller passage is very smooth and following the curvature of the blade in stream-wise direction. However, the analysis shows that there is a recirculation zone near the leading edge even at design point. When the flow is discharged into volute casing circumferentially from the impeller outlet, the high velocity flow is severely distorted and formed a spiraling vortex flow within the volute casing. A spatial and temporal wake flow core development is captured dynamically and shows how the wake core diffuses. Near volute tongue region, the impeller/volute tongue strong interaction is observed based on the periodically fluctuating pressure at outlet. The results of existing analysis also proved that the pressure fluctuation periodically is due to the position of impeller blade relative to tongue.	centrifugal pump, impeller, spiraling vortex flow, pressure fluctuation	
	Page	Title	Author	Abstract	Keywords
360-366	Prediction of Cascade Performance of Circular-Arc Blades with CFD	Masami Suzuki, Toshiaki Setoguchi, Kenji Kaneko	Thin circular-arc blade is often used as a guide vane, a deflecting vane, or a rotating blade of low pressure axial-flow turbomachine because of its easy manufacture. Ordinary design of the blade elements of these machines is done by use of the carpet diagrams for a cascade of circular-arc blades. However, the application of the carpet diagrams is limited to relatively low cambered blade operating under optimum inlet flow conditions. In order to extend the applicable range, additional design data is necessary. Computational fluid dynamics (CFD) is a promising method to get these data. In this paper, two-dimensional cascade performances of circular-arc blade are widely analyzed with CFD. The results have been compared with the results of experiment and potential theory, and useful information has been obtained. Turning angle and total pressure loss coefficients are satisfactorily predicted for lowly cambered blade. For high camber angle of 67°, the CFD results agree with experiment for the angle of attack less than that for shockless inlet condition.	Circular-arc blades, Cascade, Guide Vane, Potential Flow, CFD	
367-374	Effects of the Geometry of Components Attached to the Drain Valve on the Performance of Water Hammer Pumps	Sumio Saito, Masaaki Takahashi, Yoshimi Nagata, Keita Dejima	Water hammer pumps can effectively use the water hammer phenomenon in long-distance pipeline networks that include pumps and allow fluid transport without drive sources, such as electric motors. The results of experiments that examined the effect of the geometric form of water hammer pumps by considering their major dimensions have been reported. In addition, a paper has also been published analyzing the water hammer phenomenon numerically by using the characteristic curve method for comparison with experimental results. However, these conventional studies have not fully evaluated the pump performance in terms of pump head and flow rate, common measures indicating the performance of pumps. Therefore, as a first stage for the understanding of water hammer pump performance in comparison with the characteristics of typical turbo pumps, the previous paper experimentally examined how the hydrodynamic characteristics were affected by the inner diameter ratio of the drive and lifting pipes, the form of the air chamber, and the angle of the drive pipe. To understand the behavior of the components attached to the valve chamber and the air chamber that affects the performance of water hammer pumps, the previous study also determined the relationship between the water hammer pump performance and temporal changes in valve chamber and air chamber pressures according to the air chamber capacity. For the geometry of components attached to the drain valve, which is another major component of water hammer pumps, this study experimentally examines how the water hammer pump performance is affected by the length of the spring and the angle of the drain pipe.	Water Hammer Pump, Fluid Transients, Pump Performance, Pressure Fluctuation, Flow Visualization	
375-386	Computational Study of the Magnetically Suspended Centrifugal Blood Pump (2nd Report: Pressure Fluctuation and Stability of Impeller Rotation for Different Volute Shapes)	Yoshifumi Ogami, Daisuke Matsuoka, Masaaki Horie	The turbo-type blood pump studied in this paper has an impeller that is magnetically suspended in a double volute casing. The impeller rotates with minimal fluctuations caused by fluid and magnetic forces. In order to improve stability of the rotating impeller and to facilitate long-term use, a careful investigation of the pressure fluctuations and of the fluid force acting on the impeller is necessary. For this purpose, two models of the pump with different volute cross-sectional area are designed and studied with computational fluid dynamics software. The results show that the fluid force varies with the flow rate and shape of the volute, that the fluctuations of fluid force decrease with increasing flow rate and that the vibratory movement of the impeller is more efficiently suppressed in a narrow volute.	Computational Fluid Dynamics, Centrifugal Blood Pump, Pressure Fluctuation, Stability of Impeller Rotation, Numerical Analysis	



387-395	Radial Thrust of Single-Blade Centrifugal Pump	Yasuyuki Nishi, Junichiro Fukutomi, Ryota Fujiwara	<p>Single-blade centrifugal pumps are widely used as sewage pumps. However, the impeller of a single-blade pump is subjected to strong radial thrust during pump operation because of the geometrical axial asymmetry of the impeller.</p> <p>Therefore, to improve pump reliability, it is necessary to quantitatively understand radial thrust and elucidate the behavior and mechanism of thrust generating. This study investigates the radial thrust acting up on a single-blade centrifugal impeller by conducting experiments and CFD analysis. The results show that the fluctuating component of radial thrust increases as the flow rate deviates from the design flow rate to low or high value. Radial thrust was modeled by a combination of three components, inertia, momentum, and pressure by applying an unsteady conservation of momentum to the impeller.</p> <p>The sum of these components agrees with the radial thrust calculated by integrating the pressure and the shearing stress on the impeller surface. The behavior of each component was shown, and the effects of each component on radial thrust were clarified. The pressure component has the greatest effect on the time-averaged value and the fluctuating component of radial thrust. The time-averaged value of the inertia component is nearly 0, irrespective of the change in the flow rate. However, its fluctuating component has a magnitude nearly comparable with the pressure component at a low flow rate and slightly decreased with the increase in flow rate.</p>	Turbomachinery, Centrifugal Pump, Sewage pump, Single-Blade, Radial Thrust, CFD
396-409	Mathematical Model for the Effect of Blade Friction on the Performance of Pelton Turbine	Iresha Udayangani Atthanayake, Thusitha Sugathapala, Rathna Fernando	<p>Water turbines have been used in electricity generation for well over a century. Hydroelectricity now supplies 19% of world electricity. Many hydro power plants are operated with Pelton turbines, which is an impulse turbine. The main reasons for using impulse turbines are that they are very simple and relatively cheap. As the stream flow varies, water flow to the turbine can be easily controlled by changing the number of nozzles or by using adjustable nozzles. Scientific investigation and design of turbines saw rapid advancement during last century. Most of the research that had been done on turbines were focused on improving the performance with particular reference to turbine components such as shaft seals, speed increasers and bearings. There is not much information available on effects of blade friction on the performance of turbine. The main focus in this paper is to analyze the performance of Pelton turbine particularly with respect to their blade friction.</p>	Pelton Turbine, Blade friction, Direct friction, Indirect friction

Page	Title	Author	Abstract	Keywords
------	-------	--------	----------	----------

1-9	Numerical Analysis of the Influence of Acceleration on Cavitation Instabilities that arise in Cascade	Yuka Iga, Tasuku Konno	<p>In the turbopump inducer of a liquid propellant rocket engine, cavitation is affected by acceleration that occurs during an actual launch sequence. Since cavitation instabilities such as rotating cavitations and cavitation surges are suppressed during launch, it is difficult to obtain data on the influence of acceleration on cavitation instabilities. Therefore, as a fundamental investigation, in the present study, a three-blade cyclic cascade is simulated numerically in order to investigate the influence of acceleration on time-averaged and unsteady characteristics of cavitation that arise in cascade. Several cases of acceleration in the axial direction of the cascade, including accelerations in the upstream and downstream directions, are considered. The numerical results reveal that cavity volume is suppressed in low cavitation number condition and cavitation performance increases as a result of high acceleration in the axial-downstream direction, also, the inverse tendency is observed in the axial-upstream acceleration. Then, the regions in which the individual cavitation instabilities occur shift slightly to a low-cavitation-number region as the acceleration increases downstream. In addition, in a downstream acceleration field, neither sub-synchronous rotating cavitation nor rotating-stall cavitation are observed. On the other hand, rotating-stall cavitation occurs in a relatively higher-cavitationnumber region in an upstream acceleration field. Then, acceleration downstream is robust against cavitation instabilities, whereas cavitation instabilities easily occur in the case of acceleration upstream. Additionally, comparison with the Froude number under the actual launch conditions of a Japanese liquid propellant rocket reveals that the cavitation performance will not be affected by the acceleration under the current launch conditions.</p>	Cavitation Instability, Cascade, Acceleration, Homogeneous Model, CFD
10-17	Oxygen Transfer Characteristics of an Ejector Aeration System	Hei-Cheon Yang, Sang-Kyoo Park	<p>The objective of this study was to investigate the oxygen transfer characteristics of an ejector aeration system. In order to evaluate the oxygen transfer performance of the ejector aeration system, a comparative experiment was conducted on a conventional blower aeration system. The effect of entrained air flow rate and aerating water temperature on the oxygen transfer efficiency was investigated. The dissolved oxygen concentration increased with increasing entrained air flow rate, but decreased with increasing aerating water temperature for two aeration systems. The volumetric mass transfer coefficient increased with increasing entrained air flow rate and with increasing aerating water temperature for both aeration systems. The average mass transfer coefficient for the ejector aeration system was about 20% and 42% higher than that of the blower aeration system within the experimental range of entrained air flow rates and aerating water temperatures.</p>	Ejector aeration system, Dissolved oxygen, Oxygen mass transfer, Aerating water, Entrained air
18-29	CFD Analysis of Cavitation Phenomena in Mixed-Flow Pump	Milan Sedlar, Oldrich Sputa, Martin Komarek	<p>This paper deals with the CFD analysis of cavitating flow in the mixed-flow pump with the specific speed of 1.64 which suffers from a high level of noise and vibrations close to the optimal flow coefficient. The ANSYS CFX package has been used to solve URANS equations together with the Rayleigh-Plesset model and the SST-SAS turbulence model has been employed to capture highly unsteady phenomena inside the pump. The CFD analysis has provided a good picture of the cavitation structures inside the pump and their dynamics for a wide range of flow coefficients and NPSH values. Cavitation instabilities were detected at 70% of the optimal flow coefficient close to the NPSH3 value (NPSH3 is the net positive suction head required for the 3% drop of the total head of the pump).</p>	mixed-flow pump, cavitation, CFD, unsteady flow phenomena
30-37	Experimental Study on Adjustment of Inlet Nozzle Section to Flow Rate Variation for Darrieus-type Hydro-Turbine	Satoshi Watanabe, Kai Shimokawa, Akinori Furukawa, Kusuo Okuma, Daisuke Matsushita	<p>A two dimensional Darrieus-type turbine has been proposed for the hydropower utilization of extra-low head less than 2m. In a practical use of Darrieus-type hydro-turbine, head and flow rate may be varied temporally and seasonally. Considering that the cost advantage is required for the low head hydro turbine system, the Darrieus turbine should be operated with high efficiency in the wider range of flow rate possibly by using an additional device with simpler mechanism. In the present paper, an adjustment of inlet nozzle section by lowering the inlet nozzle height is proposed to obtain the preferable inlet velocity in low flow rate conditions. Effects of resulting spanwise partial inlet flow are investigated. Finally, an effective modification of inlet nozzle height over flow rate variation is shown.</p>	Hydro-turbine, Darrieus-type runner, Inlet nozzle, Partial inlet flow, Self-starting characteristics
38-48	Performance Prediction of Centrifugal Compressor Based on Performance Test, Similarity Conversion and CFD Simulation	Changyun Zhu, Guoliang Qin	<p>One centrifugal compressor is applied for refrigeration and its working substance is R134a. The operating points obtained by using similar conversion at different rotation speeds are compared with the numerical results. They keep consistent with each other while the rotation speeds are lower, but the error between them will become large with the increasing of the rotation speed. Then the operating points are obtained when the working substance is air by using two similar conversion methods separately. Based on the comparison, it can be obtained that the result of keeping the specific volume ratio of inlet and outlet is more accurate than the result of maintaining Ma number. Then the test result is compared with the similarity result and the numerical result when the working substance is air. It is obtained that the similarity result is more consistent with the test result better than the numerical result and the trend of efficiency and pressure ratio change with the flow rate is consistent with the test result. In the process of similar conversion, the efficiency <math>\eta</math> is no useful for similitude design and it has less influence on the conversion result.</p>	Performance Prediction of Centrifugal Compressor Based on Performance Test, Similarity Conversion and CFD Simulation

Vol. 5, No. 1, January-March, 2012

Vol. 5, No. 2, April-June, 2012	49-59	Influence of a weak superposed centripetal flow in a rotor-stator system for several pre-swirl ratios	Fadi Abdel Nour, Andrea Rinaldi, Roger Debuchy, Gérard Bois	The present study is devoted to the influence of a superposed radial inflow in a rotor-stator cavity with a peripheral opening. The flow regime is turbulent, the two boundary layers being separated by a core region. An original theoretical solution is obtained for the core region, explaining the reason why a weak radial inflow has no major influence near the periphery of the cavity but strongly affects the flow behavior near the axis. The validity of the theory is tested with the help of a new set of experimental data including the radial and tangential mean velocity components, as well as three components of the Reynolds stress tensor measured by hot-wire anemometry. The theoretical results are also in good agreement with numerical results obtained with the Fluent code and experimental data from the literature.	rotor-stator cavity, turbulent flow, superposed radial inflow, analytical solution
	60-64	Axial Wall Slits Effect on the Helical Flow in the Gap between two Concentric Cylinders	Liu Dong, Yang Xiao-yong, Ding Jian, Kim Hyoung-Bum	The helical flow regime was investigated by using DPIV when the rotating Reynolds number is small. The wall slits were azimuthally located along the inner wall of outer cylinder and the slits number of each model was 9 and 18, another plain wall model was also studied for comparison purpose. The helical vortex flow regime can be observed in all the three models. The negative temperature gradients determine the direction of the rotation and movement of the helical vortex. But the helical wavy vortex flow can only be found in the plane and 9-slit models. And the result showed that the existence of slit wall accelerated the transition process.	Helical flow regime, Slit wall, DPIV, Negative temperature gradient, Flow transition
	65-71	Application of Gurney Flaps on a Centrifugal Fan Impeller	Thomas Manoj Kumar Dundi, Nekkanti Sitaram, Munivenkatarreddy Suresh	The objective of the present investigation is to explore the possibility of improving the performance of a centrifugal fan at low Reynolds numbers using a simple passive means, namely Gurney flap (GF). GFs of 1/8th inch brass angle (3.175 mm) corresponding to 15.9% of blade exit height or 5.1% of blade spacing at the impeller tip are attached to the impeller blade tip on the pressure surface. Performance tests are carried out on the centrifugal fan with vaneless diffuser at five Reynolds numbers (viz. 0.30, 0.41, 0.55, 0.69, 0.82x10 <sup>5</sup> , i.e., at five speeds respectively at 1,100, 1,500, 2,000, 2,500 and 3,000 rpm) without and with GF. Static pressures on the vaneless diffuser hub and shroud are also measured for each speed at four flow coefficients ( $\phi=0.23$ (below design flow coefficient), $\phi=0.34$ (design flow coefficient), $\phi=0.45$ (above design flow coefficient) and $\phi=0.60$ (above design flow coefficient)) with and without GF. From the performance curves it is found that the performance of the fan improves considerably with GFs at lower Reynolds numbers and improves marginally at higher Reynolds number. Similar improvements are observed for the static pressures on the diffuser hub and shroud. The effect of Reynolds number on the performance and static pressures is considerable. However the effect is reduced with GFs.	Centrifugal fan, Gurney flap, Experimental investigation, Performance, Static pressure
	72-90	Computational Investigations of Impingement Heat Transfer on an Effused Concave Surface	M. Ashok Kumar, Bhamidi V.S.S.S. Prasad	A computational study is reported on flow and heat transfer characteristics from five rows of circular air jets impinging on a concave surface with four rows of effusion holes. The effects of exit configurations of spent air and the arrangement of jet orifices and effusion holes for a jet Reynolds number of 7500 is investigated. In all, eight cases are studied and a good qualitative correlation is found among their flow patterns, pressure variations and heat transfer distributions	Gas turbine blade cooling, Impingement cooling, Film holes, CFD
	91-99	Effects of Misalignment of High Speed Flexible Coupling on the Fighter Aircraft Transmission Characteristics	Nagesh Samikanu, Abu Muhammed Junaid Basha	The Fighter aircraft transmission system consists of a light weight, High Speed Flexible Coupling (HSFC) known as Power Take-Off shaft (PTO) for connecting Engine gearbox (EGB) with Accessory Gear Box (AGB). The HSFC transmits the power through series of specially contoured metallic annular thin flexible plates whose planes are normal to the torque axis. The HSFC operates at high speed ranging from 10,000 to 18,000 rpm. The HSFC is also catered for accommodating larger lateral and axial misalignment resulting from differential thermal expansion of the aircraft engine and mounting arrangement. The contoured titanium alloy flexible plates are designed with a thin cross sectional profile to accommodate axial and parallel misalignment by the elastic material flexure. This paper investigates the effect of misalignment on the transmission characteristics of the HSFC couplings. A mathematical model for the HSFC coupling with misalignment has been developed for analyzing the torque transmission and force interaction characteristics. An extensive testing has been conducted for validating characteristics of the designed coupling under various misalignment conditions. With this the suitability of the model adapted for the design iteration of HSFC development is validated. This method will reduce the design iteration cycles of HSFC and can be extended for the similar development of flexible couplings.	Flexible Couplings, Misalignment, rotor dynamics, system dynamics and simulation, Experimental validation, Failure Prevention
	Page	Title	Author	Abstract	Keywords
Vol. 5, No. 3, July-September, 2012	100-108	Improving Flow Distribution in a Suction Channel for a Highly Efficient Centrifugal Compressor	Manabu Yagi, Takanori Shibata, Hiromi Kobayashi, Masanori Tanaka, Hideo Nishida	Design parameters for suction channels of process centrifugal compressors were investigated, and an optimization method to enhance stage efficiency by using the new design parameters was proposed. From results of computational fluid dynamics, the passage sectional area ratios $A_c/A_e$ , $A_e/A_s$ and $A_c/A_s$ were found to be the dominant parameters for the pressure loss and circumferential flow distortion, where $A_c$ , $A_e$ and $A_s$ are passage sectional areas for the casing upstream side, casing entrance and impeller eye, respectively. The Base suction channel was optimized using the new design parameters, and the Base and Optimized types were tested. Test results showed that the Optimized suction channel achieved 3.8% higher stage efficiency than the Base suction channel while maintaining the same operating range.	Design parameter, Suction channel, Compressor, Stage efficiency, Circumferential flow distortion
	109-116	Effects of the Lift Valve Opening Area on Water Hammer Pump Performance and Flow Behavior in the Valve Chamber	Sumio Saito, Keita Dejima, Masaaki Takahashi, Gaku Hijikata, Takuya Iwamura	Water hammer pumps can effectively use the water hammer phenomenon for water pumping. They are capable of providing an effective fluid transport method in regions without a well-developed social infrastructure. The results of experiments examining the effect of the geometric form of water hammer pumps by considering their major dimensions have been reported. However, these conventional studies have not fully evaluated pump performance in terms of pump head and flow rate, common measures of pump performance. The authors have focused on the effects on the pump performance of various geometric form factors in water hammer pumps. The previous study examined how the hydrodynamic characteristics was affected by the inner diameter ratio of the drive and lift pipes and the angle of the drive pipe, basic form factors of water hammer pumps. The previous papers also showed that the behavior of water hammer pump operation could be divided into four characteristic phases. The behavior of temporal changes in valve chamber and air chamber pressures according to the air volume in the air chamber located downstream of the lift valve was also clarified in connection with changes in water hammer pump performance. In addition, the effects on water hammer pump performance of the length of the spring attached to the drain valve and the drain pipe angle, form factors around the drain valve, were examined experimentally. This study focuses on the form of the lift valve, a major component of water hammer pumps, and examines the effects of the size of the lift valve opening area on water hammer pump performance. It also clarifies the behavior of flow in the valve chamber during water hammer pump operation.	Water Hammer Pump, Fluid Transients, Pump Performance, Pressure Fluctuation, Lift Valve Opening Area
	117-125	Aspect-Ratio Effects and Unsteady Pressure Measurements inside a Cross-Flow Impeller	Katsuya Hirata, Yusuke Onishi, Shigeya Nagasaka, Ryo Matsumoto, Jiro Funaki	In the present experimental study, the authors try to clarify the characteristics of the flow around and inside a cross-flow impeller in a typical geometry, over a wide parameter range of an aspect ratio $L/D_2$ . In order to eliminate the complicated casing factors, the impeller rotates in open space without any casings. As a result, by using hot wire anemometer measurements and by conventional flow visualisations with a particle image velocimetry technique, the authors show that both the outflow rate and the maximum vorticity attain the maximum for $L/D_2 = 0.6$ . In order to investigate the aspect-ratio effect, we further reveal minute fluctuating pressures on an impeller end wall for a singular $L/D_2 = 0.6$ . Especially in these pressure measurements, the eccentric vortex is prevented to revolute by the insertion of a tongue, in order to consider the spatial structure of flow more precisely.	Cross-Flow Fan, Blower, Fan, Aspect Ratio, Pressure Measurement

	126-133	erical Evaluation of Dynamic Transfer Matrix and Unsteady Cavitation Characteristics of an Inducer	Koichi Yonezawa, Jun Aono, Donghyuk Kang, Hironori Horiguchi, Yutaka Kawata, Yoshinobu Tsujimoto	The transfer matrix and unsteady cavitation characteristics, cavitation compliance and mass flow gain factor, of cavitating inducer were evaluated by CFD using commercial software. Quasi-steady values of cavitation compliance and mass flow gain factor were obtained first by using steady calculations at various flow rate and inlet cavitation number. Then unsteady calculations were made to determine the transfer matrix and the cavitation characteristics. The results are compared with experiments to show the validity of calculations.	Inducer, dynamic transfer matrix, cavitation compliance, mass flow gain factor, CFD
	134-142	Experimental and computational analysis of behavior of three-way catalytic converter under axial and radial flow conditions	Arif Zakaria Taibani, Vilas Kalamkar	The competition to deliver ultra-low emitting vehicles at a reasonable cost is driving the automotive industry to invest significant manpower and test laboratory resources in the design optimization of increasingly complex exhaust after-treatment systems. Optimization can no longer be based on traditional approaches, which are intensive in hardware use and laboratory testing. The CFD is in high demand for the analysis and design in order to reduce developing cost and time consuming in experiments. This paper describes the development of a comprehensive practical model based on experiments for simulating the performance of automotive three-way catalytic converters, which are employed to reduce engine exhaust emissions. An experiment is conducted to measure species concentrations before and after catalytic converter for different loads on engine. The model simulates the emission system behavior by using an exhaust system heat conservation and catalyst chemical kinetic sub-model. CFD simulation is used to study the performance of automotive catalytic converter. The substrate is modeled as a porous media in FLUENT and the standard k-e model is used for turbulence. The flow pattern is changed from axial to radial by changing the substrate model inside the catalytic converter and the flow distribution and the conversion efficiency of CO, HC and NOx are achieved first, and the predictions are in good agreement with the experimental measurements. It is found that the conversion from axial to radial flow makes the catalytic converter more efficient. These studies help to understand better the performance of the catalytic converter in order to optimize the converter design.	Catalyst, CFD modeling, chemical reaction, conversion efficiency, simulation
Vol. 5, No. 4, October-December, 2012	143-151	Machine Condition Prognostics Based on Grey Model and Survival Probability	Stenly Tangkuman, Bo-Suk Yang, Seon-Jin Kim	Predicting the future condition of machine and assessing the remaining useful life are the center of prognostics. This paper contributes a new prognostic method based on grey model and survival probability. The first step of the method is building a normal condition model then determining the error indicator. In the second step, the survival probability value is obtained based on the error indicator. Finally, grey model coupled with one-step-ahead forecasting technique are employed in the last step. This work has developed a modified grey model in order to improve the accuracy of prediction. For evaluating the proposed method, real trending data of low methane compressor acquired from condition monitoring routine were employed.	Prognostics, Grey model, Survival probability, Condition monitoring, Maintenance
	152-160	Cavitation Surge in a Small Model Test Facility Simulating a Hydraulic Power Plant	Koichi Yonezawa, Daisuke Konishi, Kazuyoshi Miyagawa, François Avellan, Peter Doerfler, Yoshinobu Tsujimoto	Model tests and CFD were carried out to find out the cause of cavitation surge in hydraulic power plants. In experiments the cavitation surge was observed at flow rates higher and lower than the swirl free flow rate, both with and without a surge tank placed just upstream of the inlet volute. The surge frequency at smaller flow rate was much smaller than the swirl mode frequency caused by the whirl of vortex rope. An unsteady CFD was carried out with two boundary conditions: (1) the flow rate is fixed to be constant at the volute inlet, (2) the total pressure is kept constant at the volute inlet, corresponding to the experiments without/with the surge tank. The surge was observed with both boundary conditions at both higher and lower flow rates. Discussions as to the cause of the surge are made based on additional tests with an orifice at the diffuser exit, and with the diffuser replaced with a straight pipe.	Draft tube surge, cavitation, Hydro turbine
	161-167	Influence of Blade Row Distance on Performance and Flow Condition of Contra-Rotating Small-Sized Axial Fan	Toru Shigemitsu, Junichiro Fukutomi, Hiroki Shimizu	Small-sized axial fans are used as air coolers for electric equipment. There is a strong demand for higher power of fans according to the increase of quantity of heat from electric devices. Therefore, higher rotational speed design is conducted, although, it causes the deterioration of the efficiency and the increase of noise. Then, the adoption of contra-rotating rotors for small-sized fans is proposed for the improvement of the performance. In the case of contra-rotating rotors, blade row distance between the front and the rear rotors influences on the performance and the noise. Therefore, it is important to clarify the optimum blade row distance between front and rear rotors. The performance curves of the contra-rotating small-sized axial fan under the condition of different blade row distances are shown and the blade row interaction between the front and the rear rotors are discussed by the numerical results. Furthermore, the optimum blade row distance of the contra-rotating small-sized axial fan is considered.	Small-sized axial fan, Contra-rotating rotors, Performance, Internal flow, Blade row distance
	168-173	Numerical Investigation on Aerodynamic Performance of a Centrifugal Fan with Splitter Blades	Jin-Hyuk Kim, Kyung-Hun Cha, Kwang-Yong Kim, Choon-Man Jang	This paper presents a numerical investigation on the aerodynamic performance according to the application of splitter blades in an impeller of a centrifugal fan used for a refuse collection system. Numerical analysis of a centrifugal fan was carried out by solving three-dimensional Reynolds-averaged Navier-Stokes equations with the shear stress transport turbulence model. A validation of numerical results was conducted by comparison with experimental data for the pressure and efficiency. From analyses of the internal flow field of the reference fan, the losses by the reverse-flows were observed in the region of the blade passage. In order to reduce these losses and enhance fan performance, two splitter blades were applied evenly between the main blades, and centrifugal impellers having the different numbers of the main blades were tested with their application. Throughout the numerical analyses of the centrifugal fan with splitter blades, it was found that the reverse-flow regions in the blade passage can be reduced by controlling the main blade numbers with splitter blades. The application of splitter blades in a centrifugal fan leads to significant improvement in the overall fan performance.	Centrifugal fan, impeller, splitter, pressure, efficiency, Reynolds-averaged Navier-Stokes equations
	174-179	Effect of Intake Vortex Occurrence on the Performance of an Axial Hydraulic Turbine in Sihwa-Lake Tidal Power Plant, Korea	Jin-Hyuk Kim, Man-Woong Heo, Kyung-Hun Cha, Kwang-Yong Kim, Se-Wyan Tac, Yong Cho, Jae-Chun Hwang, Maria Collins	A numerical study to investigate the effect of intake vortex occurrence on the performance of an axial hydraulic turbine for generating tidal power energy in Sihwa-lake tidal power plant, Korea, is performed. Numerical analysis of the flow through an axial hydraulic turbine is carried out by solving three-dimensional Reynolds-averaged Navier-Stokes equations with the shear stress transport turbulence model. In the real turbine operation, the vortex flows are occurred in both the side corners around the intake of an axial hydraulic turbine due to the interaction between the inflow angle of water and intake structure. To analyze these vortex phenomena and to evaluate their impacts on the turbine performance, the internal flow fields of the axial hydraulic turbines with the different inflow angles are compared with their performances. As the results of numerical analysis, the vortex flows do not directly affect the turbine performance.	Sihwa-lake tidal power plant, axial hydraulic turbine, vortex flow behavior, numerical analysis, Reynolds-averaged Navier-Stokes equations
	Page	Title	Author	Abstract	Keywords
	1-10	Flow Analyses Inside Jet Pumps Used for Oil Wells	Abdus Samad, Mohammad Nizamuddin	Jet pump is one type of artificial lifts and is used when depth and deviation of producing wells increases and pressure depletion occurs. In the present study, numerical analysis has been carried out to analyze the flow behavior and find the performance of the jet pump. Reynolds-averaged Navier Stokes equations were solved and k-ε turbulence model was used for simulations. Water and light oil as primary fluids were used to pump water, light oil and heavy oil. The ratios of area and length to diameter of the mixing tube were considered as design parameters. The pump efficiency was considered to maximize for the downhole conditions. It was found that the increase in viscosity and density of the secondary fluid reduced efficiency of the system. Water as primary fluid produced better efficiency than the light oil. It was also found that the longer throat length increased efficiency upto 40% if light oil was used as primary fluid and secondary fluid viscosity was 350 cSt.	Artificial lift, well pumping, jet pump, hydraulic lift, primary fluid, secondary fluid

11-17	Performance and Internal Flow Condition of Mini Centrifugal Pump with Splitter Blades	Toru Shigemitsu, Junichiro Fukutomi, Kensuke Kaji, Takashi Wada	Mini centrifugal pumps having a diameter smaller than 100mm are employed in many fields. But the design method for the mini centrifugal pump is not established because the internal flow condition for these small-sized fluid machines is not clarified and conventional theory is not suitable for small-sized pumps. Therefore, mini centrifugal pumps with simple structure were investigated by this research. Splitter blades were adopted in this research to improve the performance and the internal flow condition of mini centrifugal pump which had large blade outlet angle. The original impeller without the splitter blades and the impeller with the splitter blades were prepared for an experiment. The performance tests are conducted with these rotors in order to investigate the effect of the splitter blades on performance and internal flow condition of mini centrifugal pump. On the other hand, a three dimensional steady numerical flow analysis is conducted with the commercial code (ANSYS-CFX) to investigate the internal flow condition in detail. It is clarified from experimental results that the performance of the mini centrifugal pump is improved by the effect of the splitter blades. Blade-to-blade low velocity regions are suppressed in the case with the splitter blades and total pressure loss regions are decreased. The effects of the splitter blades on the performance and the internal flow condition are discussed in this paper.	Mini centrifugal pump, Performance, Internal flow, Splitter blade
18-24	Design and Experimental Studies of Radial-Outflow Type Diagonal Flow Fan	Yoichi Kinoue, Norimasa Shiomi, Toshiaki Setoguchi	In order to apply the design method of diagonal flow fan based on axial flow design to the design of radial-outflow type diagonal flow fan which has lower specific speed of 600-700 [min-1, m <sup>3</sup> /min, m], radial-outflow type diagonal flow fan which specific speed was 670 [min-1, m <sup>3</sup> /min, m] was designed by a quasi three-dimensional design method. Experimental investigations were conducted by fan characteristics test, flow surveys by a five-hole probe and a hot wire probe. Fan characteristics test agreed well with the design values. In the flow survey at rotor outlet, the characteristic region was observed. Two flow phenomena are considered as the cause of the characteristic region, one is tip leakage vortex near rotor tip and another is pressure surface separation on the rotor blade.	Diagonal flow fan, Radial outflow, Five-hole probe, Hot wire probe
25-32	Internal Flow Condition of High Power Contra-Rotating Small-Sized Axial Fan	Toru Shigemitsu, Junichiro Fukutomi, Takuya Agawa	Data centers have been built with spread of cloud computing. Further, electric power consumption of it is growing rapidly. High power cooling small-sized fans; high pressure and large flow rate small-sized fan, are used for servers in the data centers and there is a strong demand to increase power of it because of increase of quantity of heat from the servers. Contra-rotating rotors have been adopted for some of high power cooling fans to meet the demand for high power. There is a limitation of space for servers and geometrical restriction for cooling fans because spokes to support fan motors, electrical power cables and so on should be installed in the cooling fans. It is important to clarify complicated internal flow condition and influence of a geometric shape of the cooling fans on performance to achieve high performance of the cooling fans. In the present paper, the performance and the flow condition of the high power contra-rotating small-sized axial fan with a 40mm square casing are shown by experimental and numerical results. Furthermore, influence of the geometrical shape of the small-sized cooling fan on the internal flow condition is clarified and design guideline to improve the performance is discussed.	Axial flow fan, Cooling fan, Performance, Internal flow
33-48	An Outlook on the Draft-Tube-Surge Study	Michihiro Nishi, Shuhong Liu	If large pressure fluctuation is observed in the draft tube of a Francis turbine at part-load operation, we have generally called it draft-tube-surge. As occurrence of this phenomenon seriously affects the limit of turbine operating range, extensive studies on the surge have been made since proposal of surge-frequency criterion given by Rheingans. According to the literature survey of related topics in recent IAHR symposiums on hydraulic machinery and systems, in which state-of-the-art contributions were mainly presented, a certain review of them may be desirable for an outlook on the future studies in this research field. Thus, in this review paper, the authors' previous attempts for the last three decades to challenge the following topics: a rational method for component test of a draft tube, nature of spiral vortex rope and its behavior in a draft tube and cavitation characteristics of pressure fluctuations, are introduced together with other related contributions, expecting that more useful and significant studies will be accomplished in the future.	draft tube, pressure surge, swirl flow, vortex rope, cavitation, review paper
49-55	Water Lubricated Guide Bearing with Self-aligning Segments	Tadashi OGUMA, Naritoshi NAKAGAWA, Makoto MIKAMI, Long THANTRONG, Yasumi KIZAKI, Fumio TAKIMOTO	Water lubricated guide bearing was newly released and has been applied to actual hydro turbines with vertical shaft. As a result, they can have not only high bearing performance but environmental advantages in meeting the demand for reducing river pollution by oil leakage from oil lubricated guide bearing. The PTFE composite guide bearing was tested by experimental equipment operated under conditions similar to those of actual hydro turbines. Circumferential and axial tilting bearing segments help to improve the bearing performance and efficiency due to low friction loss in the bearing system. Furthermore, bearing cooling systems could be eliminated and maintenance periods could be extended, thus the initial investment and operating costs of the hydroelectric power plant are reduced.	Water lubricated guide bearing, PTFE, Tilting segment, Vertical shaft hydro turbine, Hydrodynamic lubrication
56-74	Analytical Study on Stall Stagnation Boundaries in Axial-Flow Compressor and Duct Systems	Nobuyuki Yamaguchi	Stall stagnations in the system of axial-flow compressors and ducts occur in transition from deep surge conditions to decayed or converged stall conditions. The present study is concerned with the boundaries between the deep surges and the stagnation stalls on the basis of analytical results by a code on surge transients analysis and simulation. The fundamental acoustical-geometrical stagnation boundaries were made clear from examinations of the results on a variety of duct configurations coupled with a nine-stage compressor and a single stage fan. The boundary was found to be formed by three parts, i.e., B- and A-boundaries, and an intermediate zone. The B-boundary occurs for the suction-duct having a length of about a quarter of the wave-length of the first resonance in the case of very short and fat plenum-type delivery duct. On the other hand, the A-boundary occurs for the long and narrow duct- type delivery flow-path having a length about a fifth of the wavelength and relatively small sectional area in the case of short and narrow suction ducts. In addition to this, the reduced surge-cycle frequencies with respect to the duct lengths are observed to have respective limiting values at the stagnation boundaries. The reduced frequency for the B-boundary is related with a limiting value of the Greitzer's B parameter. The tendency and the characteristic features of the related flow behaviors in the neighborhood of the boundaries were also made clearer.	Compressor, surge, stall, stall stagnation, unsteady flow, flow oscillation
75-86	Effects of Acoustic Resonance and Volute Geometry on Phase Resonance in a Centrifugal Fan	Yoshinobu Tsujimoto, Hiroshi Tanaka, Peter Doerfler, Koichi Yonezawa, Takayuki Suzuki, Keisuke Makikawa	The effects of acoustic resonance and volute geometry on phase resonance are studied theoretically and experimentally using a centrifugal fan. One dimensional theoretical model is developed taking account of the reflection from the discharge pipe end. It was found that the phase resonance occurs, even with the effects of acoustic resonance, when the rotational speed of rotor-stator interaction pattern agrees with the sound velocity. This was confirmed by experiments with and without a silencer at the discharge pipe exit. The pressure wave measurements showed that there are certain effects of the cross-sectional area change of the volute which is neglected in the one dimensional model. To clarify the effects of area change, experiments were carried out by using a ring volute with a constant area. It was demonstrated that the phase resonance occurs for both interaction modes travelling towards/away from the volute. The amplitude of travelling wave grows towards the volute exit for the modes rotating towards the volute exit, in the same direction as the impeller. However, a standing wave is developed in the volute for the modes rotating away from the volute exit in the opposite direction as the impeller, as a result of the interaction of a growing wave while travelling towards the tongue and a reflected wave away from the tongue.	Phase resonance, acoustic resonance, rotor-stator interaction, centrifugal fan

Page	Title	Author	Abstract	Keywords
87-93	Fluid-Structure Interaction Study on Diffuser Pump With a Two-Way Coupling Method	Xu Huan, Liu Houlin, Tan Minggao, Cui Jianbao	In order to study the effect of the fluid-structure interaction (FSI) on the simulation results, the external characteristics and internal flow features of a diffuser pump were analyzed with a two-way flow solid coupling method. And the static and dynamic structure analysis of the blade was also calculated with the FEA method. The steady flow field is based on Reynolds Averaged N-S equations with standard k-ε turbulent model, the unsteady flow field is based on the large eddy simulation, and the structure response is based on elastic transient structural dynamic equation. The results showed that the effect of FSI on the head prediction based on CFD really exists. At the same radius, the vanes stress on the nodes closed shroud and hub was larger than other nodes. A large deformation region existed near inlet side at the middle of blades. The strength of impeller satisfied the strength requirement with static stress analysis based on the fourth strength theory. The dynamic stress varied periodically with the impeller rotating. It was also found that the fundamental frequency of the dynamic stress is the rotating frequency and its harmonic frequency. The frequency of maximum stress amplitude at node 1626 was 7 times of the rotating frequency. The frequency of maximum stress amplitude at node 2328 was 14 times of the rotating frequency. No matter strength failure or fatigue failure, the root of blades near shroud is the key region to analyse.	Fluid-Structure Interaction, Diffuser Pump, Two-Way Coupling Method, head prediction
94-104	Design and Prediction of Three Dimensional Flows in a Low Speed Highly Loaded Axial Flow Fan	Xuejiao Liu, Liu Chen, Ren Dai, Ailing Yang	This paper describes the design to increase the blade loading factor of a low speed axial flow fan from normal 0.42 to highly loaded 0.55. A three-dimensional viscous solver is used to model the flows in the highly-loaded and normal loaded stages over its operation range. At the design point operation the static pressure rise can be increased by 20 percent with a deficit of efficiency by 0.3 percent. In the highly loaded fan stage, the rotor hub flow stalls, and separation vortex extends over the rotor hub region. The backflow, which occurs along the stator hub-suction surface, changes the exit flow from the prescribed axial direction. Results in this paper confirm that the limitation of the two dimensional diffusion does not affect primarily on the fan's performance. Highly loaded fan may have actually better performance than its two dimensional design. Three dimensional designing approaches may lead to better highly loaded fan with controlled rotor hub stall.	highly-loaded fan, numerical simulation, stage loading factor
105-112	Low Speed Design of Rear Rotor in Contra-Rotating Axial Flow Pump	Linlin Cao, Satoshi Watanabe, Simpei Momosaki, Toshiki Imanishi, Akinori Furukawa	The application of contra-rotating rotors for higher specific speed pump has been proposed in our studies, which is in principle effective for reducing the rotational speed and/or the pump size under the same specification of conventional axial flow pump. In the previous experiments of our prototype, the cavitation inception at the tip region of the rear rotor rather than that of the front rotor and the strong potential interaction from the suction surface of the rear rotor blade to the pressure surface of the front one were observed, indicating the possibility to further improve the pump performance by optimizing rotational speed combination between the two rotors. The present research aims at the design of rear rotor with lower rotational speed. Considering the fact that the incoming flow velocity defects at the tip region of the rear rotor, an integrated inflow model of 'forced vortex' and 'free vortex' is employed. The variation of maximum camber location from hub to tip as well as other related considerations are also taken into account for further performance improvement. The ideas cited above are separately or comprehensively applied in the design of three types of rear rotor, which are subsequently simulated in ANSYS CFX to evaluate the related pump performance and therefore the whole low speed design idea. Finally, the experimental validation is carried out on one type to offer further proofs for the availability of the whole design method.	contra-rotating rotors, blade design, cavitation, CFD
113-120	Effect of Inlet Clearance Gap on the Performance of an Industrial Centrifugal Blower with Parallel Wall Volute	Chinnasamy Hariharan, Mukka Govardhan	While performing numerical simulations, it is general industrial practice to neglect the clearance gap between the impeller and the inlet duct. In the present work, the effect of clearance gap on the performance of an industrial sized centrifugal blower is simulated for two volutes of width ratios and various flow coefficients. The results show that the clearance has a positive effect at low mass flow rates. This is observed in the pressure rise (1.3%) as well as in efficiency (0.7%). At higher mass flow rates, it has a negative effect with a drop in efficiency of 1% and pressure drop of about 1.4%. The effect of clearance gap on volute with higher width ratio is smaller when compared with the volute with smaller width ratio.	Parallel wall volute, clearance gap, centrifugal blower
121-136	Hexagonal reciprocating pump: advantages and weaknesses	Milan Stanko, Michael Golan	This paper reports the 1-D fluid transient simulation results of the discharge flow conditions in a 6-cylinder reciprocating slurry pump. Two discharge manifold configurations are studied comparatively, a case with a hexagon shaped discharge manifold where each cylinder discharges at a single vertex, and a case where all the cylinders discharges are lumped together into a tank shaped manifold. In addition, the study examines the effect of two pulsation mitigation measures in the case of hexagonal manifold; a single inline orifice in one of the hexagon sides and a volumetric dampener at the manifold outlet. The study establishes the pressure and flow fluctuation characteristics of each configuration and decouples the pulsation characteristics of the pump and the discharge manifold.	Hexagonal piston pump, pressure pulsation, reciprocating piston pump
137-143	Application of Constant Rate of Velocity or Pressure Change Method to Improve Annular Jet Pump Performance	Xuelong Yang, Xiping Long, Yong Kang, Longzhou Xiao	To improve annular jet pump (AJP) performance, new ways named constant rate of velocity/pressure change method (CRVC/CRPC) were adopted to design its diffuser. The design formulas were derived according to the assumption of linear velocity/pressure variation in the diffuser. Based on the two-dimensional numerical simulations, the effect of the diffuser profile and the included angle on the pump performance and the internal flow details has been analyzed. The predicted results of the RNG k-epsilon turbulence model show a better agreement with the experiment data than that of the standard and the realizable k-epsilon turbulence models. The AJP with the CRPC diffuser produces a linear pressure increase in the CRPC diffuser as expected. The AJP with CRPC/CRVC diffuser has better performance when the diffuser included angle is greater or the diffuser length is shorter. Therefore, the AJP with CRPC/CRVC diffuser is suitable for applications requiring space limitation and weight restriction.	Jet pump, turbulence model, annular jet pump, diffuser, numerical simulation, structure optimization
144-151	Influence of Blade Number on the Flow Characteristics in the Vertical Axis Propeller Hydro Turbine	Sun-Seok Byeon, Youn-Jea Kim	In this paper, the design method of a low-head propeller-type hydro turbine is studied for various numbers of blades on an axial propeller. We also investigate the relationship between geometrical parameters and internal performance parameters, such as angular velocities (100, 200, 300, 400 rpm) and 2.5-4m low heads through a three-dimensional numerical method with the SST turbulent model. The numerical results showed that the blade number had a more dominant influence than the change in heads and rotational speed on the flow characteristics of the turbine. The distributions of pressure and velocity in the streamwise direction of the propeller turbine were graphically depicted. Especially, the relationship among dimensionless parameters like specific speed (Ns), flow coefficient (φ) and power coefficient were investigated.	Propeller hydro turbine, Free vortex theory, Shaft power, Head, Blade number
152-159	A Study on Darrieus-type Hydroturbine toward Utilization of Extra-Low Head Natural Flow Streams	Kei Tanaka, Kotaro Hirowatari, Kai Shimokawa, Satoshi Watanabe, Daisuke Matsushita, Akinori Furukawa	A two-dimensional Darrieus-type hydroturbine system, installed with a wear for flow streams such as small rivers and waterways, has been developed for hydropower utilization of extra-low head less than 2m. There are several problems such as flow rate change and flowing wastes to be solved for its practical use in natural flow streams. In the present study, at first, a design guideline in the case of overflow or bypass flow is shown by using simple flow model. Next, in order to avoid the unexpected obstacles flowing into the hydroturbine, an installation of waste screening system is examined. It is confirmed that the screen is effective with some amount of bypass flow rate, however the output power is remarkably deteriorated.	low head hydropower, Darrieus turbine, bypass flow, waste screening system

	160-169	Study of Mechanism of Counter-rotating Turbine Increasing Two-Stage Turbine System Efficiency	Yanbin Liu, Weilin Zhuge, Xinqian Zheng, Yangjun Zhang, Shuyong Zhang, Junyue Zhang	Two-stage turbocharging is an important way to raise engine power density, to realize energy saving and emission reducing. At present, turbine matching of two-stage turbocharger is based on MAP of turbine. The matching method does not take the effect of turbines' interaction into consideration, assuming that flow at high pressure turbine outlet and low pressure turbine inlet is uniform. Actually, there is swirl flow at outlet of high pressure turbine, and the swirl flow will influence performance of low pressure turbine which influencing performance of engine further. Three-dimension models of turbines with two-stage turbocharger were built in this paper. Based on the turbine models, mechanism of swirl flow at high pressure turbine outlet influencing low pressure turbine performance was studied and a two-stage radial counter-rotation turbine system was raised. Mechanisms of the influence of counter-rotation turbine system acting on low-pressure turbine were studied using simulation method. The research result proved that in condition of small turbine flow rate corresponding to engine low-speed working condition, counter-rotation turbine system can effectively decrease the influence of swirl flow at high pressure turbine outlet imposing on low pressure turbine and increases efficiency of the low-pressure turbine, furthermore increases the low-speed performance of the engine.	Two-stage turbocharger, Swirl flow, Interaction, Radial counter-rotation turbine, Turbine efficiency, Three-dimension simulation
Vol. 6, No. 4, October-December, 2013	170-176	Effect of Axial Spacing between the Components on the Performance of a Counter Rotating Turbine	Rayapati Subbarao, Mukka Govardhan	Counter Rotating Turbine (CRT) is an axial turbine with a nozzle followed by a rotor and another rotor that rotates in the opposite direction of the first one. Axial spacing between blade rows plays major role in its performance. Present work involves computationally studying the performance and flow field of CRT with axial spacing of 10, 30 and 70% for different mass flow rates. The turbine components are modeled for all the three spacing. Velocity, pressure, entropy and Mach number distributions across turbine stage are analyzed. Effect of spacing on losses and performance in case of stage, Rotor1 and Rotor2 are elaborated. Results confirm that an optimum axial spacing between turbine components can be obtained for the improved performance of CRT.	Counter Rotating Turbine, Axial Spacing, Turbine Performance, Pressure Coefficient
	177-187	A Second Order Exact Scaling Method for Turbomachinery Performance Prediction	Peter Franz Pelz, Stefan Sebastian Stojek	A scaling method valid for most turbomachines based on first principles is derived. It accounts for axial and centri-fugal turbomachines with respect to relative gap width / tip clearance, relative roughness, Reynolds number and/or Mach number for design and off-design operation as well. The scaling method has been successfully validated by a variety of experimental data obtained at TU Darmstadt. The physically based, hence reliable and universal method is compared with previous, empirical scaling methods.	Scaling, Efficiency, Axial, Centrifugal, Turbine, Compressor, Fan, Reynolds number, Mach number, Size, Tip Clearance, Gap, Roughness
	189-199	Development of a Simulation Method of Surge Transient Flow Phenomena in a Multistage Axial Flow Compressor and Duct System	Nobuyuki Yamaguchi	A practical method of surge simulation in a system of a high-pressure-ratio multistage axial flow compressor and ducts, named SRGRAN, is described about the principal procedures and the details. The code is constructed on the basis of one-dimensional stage-by-stage modeling and application of fundamental equations of mass, momentum, and energy. An example of analytical result on surge behaviors is included as an experimental verification. It will enable to examine the transient flow phenomena caused by possible compressor surges and their influences on the system components in plant systems including high-pressure-ratio axial compressors or gas turbines.	Fluid Machine, Axial Flow Compressor, Surge, Fluid Dynamics, Analytical Simulation, Transient Flow
	200-205	The Numerical Simulation of Unsteady Flow in a Mixed flow Pump Guide Vane	Li Yi-bin, Li Ren-nian, Wang Xiu-yong	In order to investigate the characteristics of unsteady flow in a mixed flow pump guide vane under the small flow conditions, several indicator points in a mixed flow pump guide vane was set, the three-dimensional unsteady turbulence numerical value of the mixed flow pump which is in the whole flow field will be calculated by means of the large eddy simulation (LES), sub-grid scale model and sliding mesh technology. The experimental results suggest that the large eddy simulation can estimate the positive slope characteristic of head & capacity curve. And the calculation results show that the pressure fluctuation coefficients of the middle section in guide vane inlet will decrease firstly and then increase. In guide vane outlet, the pressure fluctuation coefficients of section will be approximately axially symmetrical distribution. The pressure fluctuation minimum of section in guide vane inlet is above the middle location of the guide vane suction surface, and the pressure fluctuation minimum of section in which located the middle and outlet of guide vane. When it is under the small flow operating condition, the eddy scale of guide vane is larger, and the pressure fluctuation of the channel in guide vane being cyclical fluctuations obviously which leads to the area of eddy expanding to the whole channel from the suction side. The middle of the guide vane suction surface of the minimum amplitude pressure fluctuation to which the vortex core of eddy scale whose direction of fluid's rotation is the same to impeller in the guide vane adhere.	Mixed-flow Pump, Guide Vane, Pressure fluctuation, Vortex, LES, CFD
	206-212	Study on the frequency of self-excited pulse jet	Wang Jian, Li Jiangyun, Guan Kai, Ma Tianyou	Self-excited pulse jet is a specific nozzle with a closed chamber which can change a continuous jet into a pulse one. Energy of the pulse jet can be output not only unevenly but also with multifrequency. With the peak pressure of pulse jet, the hitting power would be 2-2.5 times higher than that of continuous jet. In order to reveal the correlation between the self-excited pulse frequency and nozzle diameter ratio, nozzle spacing and operating pressure, the model of 3D unsteady cavitation model has been used. We found that with the same nozzle structure parameters and the different operating pressure, the self-excited frequency and the width of peak crest are different, but the wave profiles are similar. With FFT, we also found that the less bandwidth of amplitude in low frequency range will lead to the wider wave crest of outlet velocity in its time domain, and the larger force of the strike will be gained. By studying the St of self-excite nozzle, not only the frequency of a certain nozzle can be predicted, but also a nozzle structure with a certain frequency can be designed.	Self-excited, Pulse Jet, Frequency, Numerical Simulation
	213-221	Experimental Study on Performance of a Propulsive Nozzle with a Blower Piping System	Masahiko Sakamoto	The characteristics of the thrust for ship propulsion equipment directly driven by air compressed by pressure fluctuation in a blower piping system are investigated. The exhaust valve is positioned upon the air ejection hole in the discharge pipe in order to induce the large-scale pressure fluctuation, and the effects of the valve on the pressure in the pipes and the thrust for the propulsive nozzle are examined. The pressure in the pipes decreases immediately after the valve is opened, and it increases just before the valve is closed. The thrust for the propulsive nozzle monotonically increases with increasing number of revolutions and depth. The interfacial wave in the nozzle appears in the frequency of approximately 4Hz, and it is important for the increase of the thrust to synchronize the opening-closing cycle for the exhaust valve with the generation frequency of the interfacial wave. The finite difference lattice Boltzmann method is helpful to investigate the characteristics of the flow in the nozzle.	Ship propulsion, water jet, pressure fluctuation, blower, pipe line, fluid machinery
		Page	Title	Author	Abstract



Vol. 7, No. 1, January-March, 2014	1-6	Experiment Investigation on Fluid Transportation Performance of Propellant Acquisition Vanes in Microgravity Environment	Baotang Zhuang, Yong Li, Xianwu Luo, Halin Pan, Jingjing Ji	The propellant acquisition vane (PAV) is a key part of a vane type surface tension propellant management device (PMD), which can manage the propellant effectively. In the present paper, the fluid transportation behaviors for five PAVs with different sections were investigated by using microgravity drop tower test. Further, numerical simulation for the propellant flow in a PMD under microgravity condition was also carried out based on VOF model, and showed the similar flow pattern for PAVs to the experiment. It is noted that the section geometry of PAVs is one of the main factors affecting the fluid transportation behavior of PMD. PAVs with bottom length ratio of 5/6 and 1/2 have larger propellant transportation velocity. Based on the experiments, there were two stages during the process of propellant transportation under microgravity environment: liquid relocation and steady transportation stage. It is also recognized that there is a linear correlation between liquid transportation velocity and relative time's square root. Those results can not only provide a guideline for optimization of new vane type PMDs, but also are helpful for fluid control applications in space environment.	Microgravity, Propellant Acquisition Vanes, Fluid transportation, Experiment
	7-15	Numerical Investigation on the Characteristics of Flow-Induced Noise in a Centrifugal Blower	Chanyoung Lee, Taebin Jeong, Kyoung-Ku Ha, Shin-Hyung Kang	In the present study, a computational analysis of the flow in a centrifugal blower is carried out to predict a performance and to explain noise characteristics of the blower. Unsteady, 3D Navier-Stokes equations were solved with k-ε turbulence model using CFX software. CFD results were compared with the experimental data that is acquired from an experiment conducted with the same blower. The pressure fluctuation in the blower was transformed into the frequency domain by Fourier decomposition to find the relationship between flow behaviors and noise characteristics. Sound pressure level (SPL) which is obtained from wall pressure fluctuation at impeller outlet represents relative overall sound level of the blower well. Sound spectra show that there are some specific peak frequencies at each mass flow rate and it can be explained by flow pattern.	centrifugal blower, wall pressure fluctuation, flow-induced noise
	16-27	Predicting Double-Blade Vertical Axis Wind Turbine Performance by a Quadruple-Multiple Streamtube Model	Yutaka Hara, Takafumi Kawamura, Hiromichi Akimoto, Kenji Tanaka, Takuju Nakamura, Kentaro Mizumukai	Double-blade vertical axis wind turbines (DB-VAWTs) can improve the self-starting performance of lift-driven VAWTs. We here propose the quadruple-multiple streamtube model (QMS), based on the blade element momentum (BEM) theory, for simulating DB-VAWT performance. Model validity is investigated by comparison to computational fluid dynamics (CFD) prediction for two kinds of two-dimensional DB-VAWT rotors for two rotor scales with three inner-outer radius ratios: 0.25, 0.5, and 0.75. The BEM-QMS model does not consider the effects of an inner rotor on the flow speed in the upwind half of the rotor, so we introduce a correction factor for this flow speed. The maximum power coefficient predicted by the modified BEM-QMS model for a DB-VAWT is thus closer to the CFD prediction.	Wind turbine, Double-blade rotor, VAWT, BEM, CFD, QMS
	28-33	Numerical simulation Analysis of Tip Clearance Flow in a Centrifugal Compressor	Shuiqing Zhou, Jun Wang, Chuanghua Wang, Ye Li	In order to research the relationship between the tip clearance and leakage flow of centrifugal compressor, a high speed centrifugal compressor was investigated by using CFD. A numerical study on the effect of four different rotor tip clearance sizes of centrifugal compressor, which were 0.5times, 1 times, 1.5times and 2.0times of the design tip clearance, was carried out. Efficiency and pressure ratio curves were obtained under different mass flow. The reasons of the clearance vortex and the factors of vortex size were analyzed. The result indicated that with the increase of tip clearance size, the performance of the compressor changed obviously, the performance parameters such as efficiency and pressure ratio tended to decrease obviously. While, the leakage flow does not always lead to leak vortex. The strength of the vortex increased with the tip clearance. The size of leak vortex was affected by the pressure difference between the suction side and the pressure side of blade tip.	Leakage flow, Tip clearance, Pressure ratio, Pressure difference
	34-41	Flow Evaluation and Hemolysis Analysis of BVAD Centrifugal Blood Pump by Computational Fluids Dynamics	Jeerast Bumrungratch, Andy Chit Tan, Shu-Hong Liu, Xian-Wu Luo, Qing-Yu Wu, Jian-Ping Yuan, Ming-Kui Zhang	Computational fluid dynamics (CFD) and particle image velocimetry (PIV) are commonly used techniques to evaluate the flow characteristics in the development stage of blood pumps. CFD technique allows rapid change to pump parameters to optimize the pump performance without having to construct a costly prototype model. These techniques are used in the construction of a bi-ventricular assist device (BVAD) which combines the functions of LVAD and RVAD in a compact unit. The BVAD construction consists of two separate chambers with similar impellers, volutes, inlet and output sections. To achieve the required flow characteristics of an average flow rate of 5 l/min and different pressure heads (left – 100mmHg and right – 20mmHg), the impellers were set at different rotating speeds. From the CFD results, a six-blade impeller design was adopted for the development of the BVAD. It was also observed that the fluid can flow smoothly through the pump with minimum shear stress and area of stagnation which are related to haemolysis and thrombosis. Based on the compatible Reynolds number the flow through the model was calculated for the left and the right pumps. As it was not possible to have both the left and right chambers in the experimental model, the left and right pumps were tested separately.	Blood Pump, Ventricular Assist device, LVAD, BVAD, Computational fluid dynamics, artificial heart
Vol. 7, No.	42-53	Phase Resonance in Centrifugal Fluid Machinery -A Comparison between Pump Mode and Turbine Mode Operations and a Discussion of Mechanisms of Flow Rate Fluctuation through a Stator-	Koichi Yonezawa, Shingo Toyahara, Shingo Motoki, Hiroshi Tanaka, Peter Doerfler, Yoshinobu Tsujimoto	Phase resonance in Francis type hydraulic turbine is studied. The phase resonance is a phenomenon that the pressure fluctuation in the penstock of hydraulic turbine installation can become very large when the pressure waves from each guide vane caused by the interaction with the runner vane reach the penstock with the same phase. Experimental and numerical studies have been carried out using a centrifugal fan. In the present study, comparisons between the pump mode and the turbine mode operations are made. The experimental and numerical results show that the rotational direction of the rotor does not affect characteristics of the pressure fluctuation but the propagation direction of the rotor-stator interaction mode plays an important role. Flow rate fluctuations through the stator are examined numerically. It has been found that the blade passing flow rate fluctuation component can be evaluated by the difference of the fluctuating pressure at the inlet and the outlet of the stator. The amplitude of the blade passage component of the pressure fluctuation is greater at the stator inlet than the one at the stator outlet. The rotor-stator interaction mode component is almost identical at the inlet and the outlet of the stator. It was demonstrated that the pressure fluctuation in the volute and connecting pipe normalized by the flow rate fluctuation becomes the same for pump and turbine mode operations, and depends on the rotational direction on the interaction mode.	Phase resonance, rotor-stator interaction, centrifugal fluid machinery, pump, turbine, flow rate fluctuation
	54-59	Research of liquid-solid two phase flow in centrifugal pump with crystallization phenomenon	Liu Dong, Wang Ya-yun, Wang Ying-ze, Wang Chun-lin, Yang min-guan	Particle Image Velocimetry combined with developed image processing method is adopted to study the liquid-solid two phase flow in the centrifugal pump impeller with crystallization phenomenon. The tracer particle is used to follow the liquid phase, which has the diameter between 8 to 12μm. The crystal particle precipitates from the sodium sulfate solution does change the wavelength of the laser, and which has great laser scattering characteristics. The diameter of the crystal particle is larger than 20μm. Through calculating the diameter of the particles in the image, the tracer particle and the crystal particle can be distinguished. By analyzing the experimental result, the following conclusion has been obtained. During the delay period, there is not any crystal particle and the pump performance has not been changed. As the crystallization process begins, the crystal nuclei appears from the supersaturation solution and grows larger with temperature decreasing, which has the tendency of moving towards the pressure side. The characteristics of liquid-solid two phase flow with crystallization phenomenon in the pump are obtained according to analysis of experimental results, and some guiding advices are presented to mitigate the crystallization phenomenon in pump impeller.	Centrifugal pump, crystallization phenomenon, PIV, image processing method

2. April-June, 2014	60-67	Effect of Inlet Geometry on Fan Performance and Inlet Flow Fields in a Semi-opened Axial Fan	Pin Liu, Norimasa Shiomi, Yoichi Kinoue, Toshiaki Setoguchi, Ying-Zi Jin	In order to clarify the effect of inlet bellmouth size of semi-opened type axial fan on its performance and flow fields around rotor, fan test and flow field measurements using hotwire anemometer were carried out for 6 kinds of bellmouth size. As results of fan test, the shaft power curve hardly changed, even if the bellmouth size changed. On the other hand, the pressure-rise near best efficiency point became small with the bellmouth size decreasing. Therefore, the value of maximum efficiency became small as the bellmouth size decreased. As results of flow field measurements at fan inlet, the main flow region with large meridional velocity existed near blade tip when the bellmouth size was large. As bellmouth size became smaller, the meridional velocity at fan inlet became smaller and the one at outside of blade tip became larger. As results of flow field measurements at fan outlet, the main flow region existed near rotor hub side.	Fan, semi-opened, inlet geometry, measurement
	68-79	Design and Simulation of Very Low Head Axial Hydraulic Turbine with Variation of Swirl Velocity Criterion	Abdul Muis, Priyono Sutikno	The type of turbine developed is based on the very low head of water potential source for the electric power production. The area of research is focused for the axial water turbine that can be applied at the simple site open channel with has a very low cost and environmental impact compared to the conventional hydro installation. High efficiency of axial turbine which applied to the very low potential head will made this type of turbine can be used at wider potential site. Existing irrigation weir and river area will be the perfect site for this turbine. This paper will compare the effects of the variation of swirl velocity criterion during the design of the blade of guide vane and rotor of the turbine. Effects of the swirl velocity criterion is wider known as a vortex conditions (free vortex, force vortex and swirl velocity constant), and the free vortex is the very popular condition that applied by most of turbine designer, therefore will be interesting to do a comparison against other criterion. ANSYS Fluent will be used for simulation and to determine the predictive performance obtained by each of design criteria.	Axial turbine, very low head turbines, swirl velocity, vortex, ANSYS Fluent
	80-85	Visualization of Flow inside a Regenerative Turbomachinery	Yang Hyeonmo, Lee Kyoung-yong, Choi Youngseok, Jeong Kyungseok	In this study, we visualized the internal flow of a regenerative turbomachinery using the direct injection tracer method. For visualization, the working fluid was water and the tracer was oil colors (marbling colors). Droplets were injected at the inlet of the machinery and the streak were recorded using a high-speed camera with high-power light sources. While circulating inside the groove, the droplets were translated by the rotational motion of the impeller. When the droplets flow out of the impeller groove, relative to the impeller, they moved more slowly. And the droplets repeatedly reentered into the groove and circulated again. Then the droplets either flowed to the outlet or to the stripper. As a result, this experiment has confirmed the internal circulating flow of a regenerative turbomachinery.	Regenerative Turbomachinery, Internal Circulation Flow, Flow Visualization, Direct Injection Tracer Method
	Page	Title	Author	Abstract	Keywords
Vol. 7, No. 3, April-June, 2014	86-93	Performance Enhancement of 20kW Regenerative Blower Using Design Parameters	Choon-Man Jang, Hyun-Jun Jeon	This paper describes performance enhancement of a regenerative blower used for a 20 kW fuel cell system. Two design variables, bending angle of an impeller and blade thickness of an impeller tip, which are used to define an impeller shape, are introduced to enhance the blower performance. Internal flow of the regenerative blower has been analyzed with three-dimensional Navier-Stokes equations to obtain the blower performance. General analysis code, CFX, is introduced in the present work. SST turbulence model is employed to estimate the eddy viscosity. Throughout the numerical analysis, it is found that the thickness of impeller tip is effective to increase the blower efficiency in the present blower. Pressure is successfully increased up to 2.8% compared to the reference blower at the design flow condition. And efficiency is also enhanced up to 2.98 % compared to the reference one. It is noted that low velocity region disturbs to make strong recirculation flow inside the blade passages, thus increases local pressure loss. Detailed flow field inside the regenerative blower is also analyzed and compared.	Regenerative Blower, Design Parameter, Pressure, Blower Efficiency, Numerical Simulation
	94-100	Study of Cavitation Instabilities in Double- Suction Centrifugal Pump	Shinya Hatano, Donghyuk Kang, Shusaku Kagawa, Motohiko Nohmi, Kazuhiko Yokota	In double-suction centrifugal pumps, it was found that cavitation instabilities occur with vibration and a periodic chugging noise. The present study attempts to identify cavitation instabilities in the double-suction centrifugal pump by the experiment and Computational Fluid Dynamics (CFD). Cavitation instabilities in the tested pump were classified into three types of instabilities. The first one, in a range of cavitation number higher than breakdown cavitation number, is cavitation surge with a violent pressure oscillation. The second one, in a range of cavitation number higher than the cavitation number of cavitation surge, is considered to be rotating cavitation and causes the pressure oscillation due to the interaction of rotating cavitation with the impeller. Last one, in a range of cavitation number higher than the cavitation number of rotating cavitation, is considered to be a surge type instability.	Double-suction centrifugal pump, Cavitation instability, Interaction of rotating cavitation with the impeller
	101-109	Fully coupled FSI analysis of Francis turbines exposed to sediment erosion	Sailesh Chitrakar, Michel Cervantes, Biraj Singh Thapa	Sediment erosion is one of the key challenges in hydraulic turbines from a design and maintenance perspective in Himalayas. The present study focuses on choosing the best design in terms of blade angle distribution of a Francis turbine runner which has least erosion effect without influencing the efficiency and the structural integrity. A fully coupled Fluid-Structure-Interaction (FSI) analysis was performed through a multi-field solver, which showed that the maximum stress induced in the optimized design for better sediment handling is less than that induced in the reference design. Some numerical validation techniques have been shown for both CFD and FSI analysis.	CFD, FSI, Francis, Sediment, Erosion
110-124	Surge Phenomena Analytically Predicted in a Multi-stage Axial Flow Compressor System in the Reduced-Speed Zone	Nobuyuki Yamaguchi	Surge phenomena in the zone of reduced speeds in a system of a nine-stage axial flow compressor coupled with ducts were studied analytically by use of a surge transient simulation code. Main results are as follows. (1) Expansion of apparently stable, non-surge working area of the pressure vs. flow field beyond the initial stage-stall line was predicted by the code in the lower speed region. The area proved analytically to be caused by significantly mismatched stage- working conditions, particularly with the front stages deep in the rotating stall branch of the characteristics, as was already known in situ and in steady-state calculations also. (2) Surge frequencies were found to increase for decreasing compressor speeds as far as the particular compressor system was concerned. (3) The tendency was found to be explained by a newly introduced volumemodified reduced surge frequency. It suggests that the surge frequency is related intimately with the process of emptying and filling of air into the delivery volume. (4) The upstream range of movement of the fluid mass having once passed through the compressor in surge was found to reduce toward the lower speeds, which could have caused additionally the increase in surge frequency. (5) The concept of the volumemodified reduced surge frequency was able to explain, though qualitatively at present, the behaviors of the area- pressure ratio parameter for the stall stagnation boundary proposed earlier by the author.	Fluid Machine, Compressor, Surge, Analytical Simulation, Transient Flow, Fluid Dynamics	

	125-129	Paint Removal of Airplane & Water Jet Application	Xue Sheng-xiong, Chen Zheng-wen, Ren Qi-le, Su Ji-xin, Han Cai-hong, Pang lei	The paint removal and recoating are the very important process in airplane maintenance. The traditional technology is to use the chemical way corroding the paint with paint remover. For changing the defects, corrosion & pollution & manual working, of the traditional technology, the physical process which removes the paint of airplane with 250MPa/250kW ultra-high pressure rotary water jetting through the surface cleaner installed on the six axes robot is studied. The paint layer of airplane is very thin and close. The contradiction of water jetting paint removal is to remove the paint layer wholly and not damage the surface of airplane. In order to solve the contradiction, the best working condition must be reached through tests. The paint removal efficiency with ultra-high pressure and move speed of not damaged to the surface. The move speed of this test is about 2m/min, and the paint removal efficiency is about 30-40m <sup>2</sup> /h, and the paint removal active area is 85-90%. No-repeat and no-omit are the base requests of the robot program. The physical paint removal technology will be applied in airplane maintenance, and will face the safety detection of application permission.	Airplane surface, Paint removal, Ultra-high pressure, Water jetting technology, Six axes robot, Test research
	130-141	Experimental Investigation of Blade-To-Blade Pressure Distribution in Contra-Rotating Axial Flow Pump	Linlin Cao, Satoshi Watanabe, Hironori Honda, Hiroaki Yoshimura, Akinori Furukawa	As a high specific speed pump, the contra-rotating axial flow pump with two rotors rotating reversely has been proved with higher hydraulic and cavitation performance, while in our previous researches, the potential interaction between two blade rows was distinctly observed for our prototype rotors designed with equal rotational speed for both front and rear rotors. Based on the theoretical and experimental evidences, a rotational speed optimization methodology was proposed and applied in the design of a new combination of contra-rotating rotors, primarily in expectation of the optimized blade pressure distributions as well as pertinently improved hydraulic performances including cavitation performance. In the present study, given one stationary and two rotating frames in the contra-rotating rotors case, a pressure measurement concept taking account of the revolutions of both front and rear rotors simultaneously was adopted. The casing wall pressure data sampled in time domain was successfully transferred into space domain, by which the ensemble averaged blade-to-blade pressure distributions at the blade tip of two contra-rotating rotors under different operation conditions were studied. It could be seen that the rotor pair with the optimized rotational speed combination as well as work division, shows more reasonable blade-to-blade pressure distribution and well weakened potential interaction. Moreover, combining the loading curves estimated by the measured casing wall pressure, the cavitation performance of the rotor pairs with new rotational speed combination were proved to be superior to those of the prototype pairs.	Axial flow pump, contra-rotating rotors, blade rows interaction, pressure measurement
	142-150	Study on the Development of Two-Stage Centrifugal Blood Pump for Cardiopulmonary Support System	Hironori Horiguchi, Tomonori Tsukiya, Takeshi Nomoto, Toratarou Takemika, Yoshinobu Tsujimoto	In the cardiopulmonary support system with an ECMO (extracorporeal membrane oxygenation), a higher pump head is demanded for a blood pump. In order to realize a blood pump with higher pump head, higher anti-hemolysis and thrombosis performances, a study on the development of unprecedented multistage blood pump was conducted. In consideration of the application of the blood pump for pediatric patients, a miniature two-stage centrifugal blood pump with the impeller's diameter of 40mm was designed and the performance was examined in experiments and computations. Some useful knowledge for a design of the blood pump with higher anti-hemolysis and thrombosis performances was obtained.	Centrifugal Pump, Artificial Heart, Blood, Hemolysis, Thrombosis
Vol. 7, No. 4, October-December, 2014	151-159	Experimental and Numerical Investigations on Performances of Darrieus-type Hydro Turbine with Inlet Nozzle	Daisuke Matsushita, Kei Tanaka, Satoshi Watanabe, Kusuo Okuma, Akinori Furukawa	Low head hydropower is one of realistic renewable energies. The Darrieus-type hydro turbine with an inlet nozzle is available for such low head conditions because of its simple structure with easy maintenance. Experimental and numerical studies are carried out in order to examine the effects of gap distances between the runner pitch circle and two edges of inlet nozzle on turbine performances. By selecting narrower gaps of left and right edges, the performance could be improved. From the results of two dimensional numerical simulations, the relation between the performance and flow behaviors around the Darrieus blade are discussed to obtain the guideline of appropriate inlet nozzle design.	low head hydropower, Darrieus turbine, inlet nozzle edge gap, performance test
	160-173	A Study on the Fundamental Surge Frequencies in Multi-Stage Axial Flow Compressor Systems	Nobuyuki Yamaguchi	Surge phenomena in multi-stage axial flow compressors were studied with attention to the frequency behaviors. A new parameter "volume-modified reduced surge frequency" was introduced, which took into consideration the essential surge process, i.e., emptying and filling of the working gas in the delivery plenum. The behaviors of the relative surge frequencies at the stall stagnation boundaries, compared with the corresponding duct resonance frequencies, have demonstrated the existence of two types of surges; i.e., a near-resonant surge and a subharmonic surge. The former, which has fundamentally a near-resonance frequency, occurs predominantly at the stall stagnation boundary for the short-and-fat plenum delivery flow-path and the long-and-narrow delivery duct flow-path, and possibly in the intermediate conditions. The latter, which has a subharmonic frequency of the fundamental near-resonant one and occurs mainly in the inter-mediate zone, is considered to be caused by the reduced frequency restricted to a limited range. In relation with those dimensionless frequencies at the stall stagnation boundary, the surge frequency behaviors in more general situations away from the boundaries could be estimated, though very roughly.	Fluid Machine, Axial Flow Compressor, Surge, Analytical Simulation, Frequency, Fluid Dynamics
	174-182	The Flow Field of Undershot Cross-Flow Water Turbines Based on PIV Measurements and Numerical Analysis	Yasuyuki Nishi, Terumi Inagaki, Yanrong Li, Ryota Omiya, Kentaro Hatano	The ultimate objective of this study is to develop a water turbine appropriate for low-head open channels to effectively utilize the unused hydropower energy of rivers and agricultural waterways. The application of a cross-flow runner to open channels as an undershot water turbine has been considered and, to this end, a significant simplification was attained by removing the turbine casing. However, the flow field of an undershot cross-flow water turbine possesses free surfaces, and, as a result, the water depth around the runner changes with variation in the rotational speed such that the flow field itself is significantly altered. Thus, clear understanding of the flow fields observed with free surfaces to improve the performance of this turbine is necessary. In this study, the performance of this turbine and the flow field were evaluated through experiments and numerical analysis. The particle image velocimetry technique was used for flow measurements. The experimental results reflecting the performance of this turbine and the flow field were consistent with numerical analysis. In addition, the flow fields at the inlet and outlet regions at the first and second stages of this water turbine were clarified.	Water Turbine, Cross-Flow Turbine, Open Channel, Free Surface, Particle Image Velocimetry, Numerical Analysis
	Page	Title	Author	Abstract	Keywords
	1-12	Improvement of Two-Stage Centrifugal Blood Pump for Cardiopulmonary Support System and Evaluation of Anti-Hemolysis Performance	Hironori Horiguchi, Tomonori Tsukiya, Toratarou Takemika, Takeshi Nomoto, Yoshinobu Tsujimoto	In cardiopulmonary support systems with a membrane oxygenation such as a percutaneous cardiopulmonary support (PCPS) or an extracorporeal membrane oxygenation (ECMO), blood pumps need to generate the pressure rise of approximately 200mmHg or higher, due to the high hydraulic resistances of the membrane oxygenation and of the cannula tubing. In order to realize the blood pump with higher pressure rise, higher anti-hemolysis and thrombosis performances, the development of novel centrifugal blood pump composed of two-stage has been conducted by the authors. In the present paper, effective attempts to decrease the wall shear stress and to suppress the stagnation are introduced for the prevention of hemolysis and thrombosis in blood pumps. The hemolysis test was also carried out and it was clarified that the decrease of wall shear stress is effective as a guideline of design of blood pumps for improving the anti-hemolysis performance.	Centrifugal Pump, Artificial Heart, Blood, Hemolysis, Thrombosis

13-22	Performance Evaluation of a Variable Frequency Heat Pump Air Conditioning System for Electric Bus	Qinghong Peng, Qungui Du	This study presents a simulation model of a heat pump air conditioning system with a variable capacity compressor and variable speeds fans for electric bus. An experimental sample has been developed in order to check results from the model. Effects on system performance of such working conditions as compressor speed, evaporator fans speeds and the condenser fans speeds have been simulated by means of developed model. The results show that the three speeds can be adjusted simultaneously according to actual working condition so that the AC system can operate under the optimum state which the control objects want to achieve. It would be a good and simple solution to extend the driving ranges of EVs because of the highest efficiency and the lowest energy consumption of AC system	Air conditioning, Heat pump, Electric bus, EER, Variable frequency
23-35	Rotordynamic Performance Measurements of An Oil-Free Turbocharger Supported on Gas Foil Bearings and Their Comparisons to Floating Ring Bearings	Yong-Bok Lee, Dong-Jin Park, Kyuho Sim	This paper presents the rotordynamic performance measurement of oil-free turbocharger (TC) supported on gas foil bearings (GFBs) for 2 liter class diesel vehicles and comparison to floating ring bearings (FRBs). Oil-free TC was designed and developed via the rotordynamic analyses using dynamic force coefficients from GFB analyses. The rotordynamics and performance of the oil-free TC was measured up to 85 krpm while being driven by a diesel vehicle engine, and compared to a commercial oil-lubricated TC supported on FRBs. The test results showed that the GFBs increased the rotor speed by ~ 20% at engine speeds of 1,500 rpm and 1,750 rpm, yielding the reduction of turbine input energy by more than 400 W. Incidentally, an external shock test on the oil-free TC casing was conducted at the rotor speed of 60 krpm, and showed a good capability of vibration damping due to the well-known dry friction mechanism of the GFBs.	turbocharger, gas foil bearings, floating ring bearings, rotordynamic performance
36-45	Numerical Analysis of Centrifugal Impeller for Different Viscous Fluids	Sayed Ahmed Imran Bellary, Abdus Samad	Oil and gas industry pumps viscous fluids and investigation of flow physics is important to understand the machine behavior to deliver such fluids. 3D numerical flow simulation and analysis for different viscous fluids at different rotational speeds of a centrifugal impeller have been reported in this paper. Reynolds-averaged Navier Stokes (RANS) equations were solved and the performance analysis was made. Standard two equation k- $\epsilon$ model was used for the turbulence closure of steady incompressible flow. An inlet recirculation and reverse flow in impeller passage was observed at low impeller speeds. It was also found that the higher viscosity fluids have higher recirculation which hinders the impeller performance.	Centrifugal impeller, Off design condition, Rotational speed, Viscosity, Inlet recirculation
46-54	Hydrodynamic Design of Thrust Ring Pump for Large Hydro Turbine Generator Units	Xide Lai, Xiang Zhang, Xiaoming Chen, Shifu Yang	Thrust-ring-pump is a kind of extreme-low specific speed centrifugal pump with special structure as numerous restrictions from thrust bearing and operation conditions of hydro-generator units. Because the oil circulatory and cooling system with thrust-ring-pump has a lot of advantages in maintenance and compactness in structure, it has widely been used in large and medium-sized hydro-generator units. Since the diameter and the speed of the thrust ring is limited by the generator set, the matching relationship between the flow passage inside the thrust ring (equivalent to impeller) and oil bath (equivalent to volute) has great influence on hydrodynamic performance of thrust-ring-pump. On another hand, the head and flow rate are varying with the operation conditions of hydro-generator units and the oil circulatory and cooling system. As so far, the empirical calculation method is employed during the actual engineering design, in order to guarantee the operating performance of the oil circulatory and cooling system with thrust-ring-pump at different conditions, a collaborative hydrodynamic design and optimization is purposed in this paper. Firstly, the head and flow rate at different conditions are decided by 1D flow numerical simulation of the oil circulatory and cooling system. Secondly, the flow passages of thrust-ring-pump are empirically designed under the restrictions of diameter and the speed of the thrust ring according to the head and flow rate from the simulation. Thirdly, the flow passage geometry matching optimization between thrust ring and oil bath is implemented by means of 3D flow simulation and performance prediction. Then, the pumps and the oil circulatory and cooling system are collaborative hydrodynamic optimized with predicted head-flow rate curve and the efficiency-flow rate curve of thrust-ring-pump. The presented methodology has been adopted by DFEM in design process of thrust-ring-pump and it shown can effectively improve the performance of whole system.	Thrust-ring-pump, Hydrodynamic design, Thrust bearing, Lubricating and cooling system, Hydro turbine generator, Low specific speed centrifugal pump
55-62	Case studies for solving the Saint-Venant equations using the method of characteristics: pipeline hydraulic transients and discharge propagation	Regina Mambeli Barros, Geraldo Lúcio Tiago Filho, Ivan Felipe Silva dos Santos	This study aims to present a hydraulic transitory study as MOC applications for solving the Saint-Venant equations in two case studies: 1) in a penstock of a small hydropower system as a simple pipeline in the case of valve-closure in the downstream boundary with a reservoir in the upstream boundary; and 2) for discharge propagation into a channel by velocity and depth of the flow channel along space evaluation. The proposed methodology by Chaudry [5] concerning the development of hydrodynamic models was used. The obtained results for first and second case study has been confirmed that MOC numerical approach is useful for several engineering purposes, including cases of hydraulic transients and discharge propagation in hydraulic systems.	Method of characteristics, Hydraulic transient, Discharge propagation, Saint-Venant equations Numerical Approach, Hydraulic Systems
36-72	Improved prediction of Pump Turbine Dynamic Behavior using a Thoma number dependent Hill Chart and Site Measurements	Maximilian Manderla, Karl N. Kiniger, Jiri Koutnik	Water hammer phenomena are important issues for the design and the operation of hydro power plants. Especially, if several reversible pump-turbines are coupled hydraulically there may be strong unit interactions. The precise prediction of all relevant transients is challenging. Regarding a recent pump-storage project, dynamic measurements motivate an improved turbine modeling approach making use of a Thoma number dependency. The proposed method is validated for several transient scenarios and turns out to improve correlation between measurement and simulation results significantly. Starting from simple scenarios, this allows better prediction of more complex transients. By applying a fully automated simulation procedure broad operating ranges of the highly nonlinear system can be covered providing a consistent insight into the plant dynamics. This finally allows the optimization of the closing strategy and hence the overall power plant performance.	pump turbine, hydraulic transients, turbine modelling, draft tube pressure, Thoma number, automated simulation
73-83	Bubble size characteristics in the wake of ventilated hydrofoils with two aeration configurations	Ashish Karn, Christopher R Ellis, Christopher Milliren, Jiarong Hong, David Scott, Roger E A Arndt, John S Gulliver	Aerating hydroturbines have recently been proposed as an effective way to mitigate the problem of low dissolved oxygen in the discharge of hydroelectric power plants. The design of such a hydroturbine requires a precise understanding of the dependence of the generated bubble size distribution upon the operating conditions (viz. liquid velocity, air ventilation rate, hydrofoil configuration, etc.) and the consequent rise in dissolved oxygen in the downstream water. The purpose of the current research is to investigate the effect of location of air injection on the resulting bubble size distribution, thus leading to a quantitative analysis of aeration statistics and capabilities for two turbine blade hydrofoil designs. The two blade designs differed in their location of air injection. Extensive sets of experiments were conducted by varying the liquid velocity, aeration rate and the hydrofoil angle of attack, to characterize the resulting bubble size distribution. Using a shadow imaging technique to capture the bubble images in the wake and an in-house developed image analysis algorithm, it was found that the hydrofoil with leading edge ventilation produced smaller size bubbles as compared to the hydrofoil being ventilated at the trailing edge.	Ventilated hydrofoil, Hydroturbine aeration, Shadow Image Velocimetry, Bubble size distribution, Auto-venting turbines
84-93	Accelerating CFD-DEM simulation of dilute pneumatic conveying with bends	Jun Du, Guoming Hu, Ziqiang Fang, Wenjie Gui	The computational cost is expensive for CFD-DEM simulation, a larger time step and a simplified CFD-DEM model can be used to accelerate the simulation. The relationship between stiffness and overlap in non-linear Hertzian model is examined, and a reasonable time step is determined by a new single particle test. The simplified model is used to simulate dilute pneumatic conveying with different types of bends, and its applicability is verified by compared with the traditional model. They are good agreement in horizontal-vertical case and vertical-horizontal case, and show a significant differences in horizontal-horizontal case. But the key features of particle rope formed in different types of bends can be obtained by both models.	CFD-DEM, dilute pneumatic conveying, time step

Vol. 8, No. 2, April-June, 2015	94-101	Numerical studies on cavitation behavior in impeller of centrifugal pump with different blade profiles	Pengfei Song, Yongxue Zhang, Cong Xu, Xin Zhou, Jinya Zhang	To investigate the influence of blade profiles on cavitation behavior in impeller of centrifugal pump, a centrifugal pump with five different blade profiles impellers are studied numerically. The impellers with five different blade profiles (single arc, double arcs, triple arcs, logarithmic spiral and linear - variable angle spiral) were designed by the in-house hydraulic design code using geometric parameters of IS 150-125-125 centrifugal pump. The experiments of the centrifugal pump have been conducted to verify numerical simulation model. The numerical results show that the blade profile lines has a weak effect on cavitation inception near blade inlet edge position, however it has the key effect on the development of sheet cavitation in impeller, and also influences the distribution of sheet cavitation in impeller channels. A slight changing of blade setting angle will induce significant difference of cavitation in impeller. The sharp changing of impeller blade setting angle causes obvious cavitation region separation near the impeller inlet close to blade suction surface and much more flow loss. The centrifugal pump with blade profile of setting angle gently changing (logarithmic spiral) has the super cavitation performance, which means smaller critical cavitation number and lower vapor cavity volume fraction at the same conditions.	Centrifugal pump, Blade profile, Cavitation behavior, Numerical simulation
	102-112	Simulations of the Dynamic Load in a Francis Runner based on measurements of Grid Frequency Variations	Rakel Ellingsen, Pål-Tore Storli	In the Nordic grid, a trend observed the recent years is the increase in grid frequency variations, which means the frequency is outside the normal range (49.9-50.1 Hz) more often. Variations in the grid frequency leads to changes in the speed of rotation of all the turbines connected to the grid, since the speed of rotation is closely related to the grid frequency for synchronous generators. When the speed of rotation changes, this implies that the net torque acting on the rotating masses are changed, and the material of the turbine runners must withstand these changes in torque. Frequency variations thus leads to torque oscillations in the turbine, which become dynamical loads that the runner must be able to withstand. Several new Francis runners have recently experienced cracks in the runner blades due to fatigue, obviously due to the runner design not taking into account the actual loads on the runner. In this paper, the torque oscillations and dynamic loads due to the variations in grid frequency are simulated in a 1D MATLAB program, and measured grid frequency is used as input to the simulation program. The maximum increase and decrease in the grid frequency over a 440 seconds interval have been investigated, in addition to an extreme event where the frequency decreased far below the normal range within a few seconds. The dynamic loading originating from grid frequency variations is qualitatively found by a constructed variable Tstress, and for the simulations presented here the variations in Tstress are found to be around 3 % of the mean value, which is a relatively small dynamic load. The important thing to remember is that these dynamic loads come in addition to all other dynamic loads, like rotor-stator interaction and draft tube surges, and should be included in the design process, if not found to be negligible.	Grid Frequency Variations, Torque Oscillations, 1D Simulations, Francis turbine, Dynamic loading
	113-123	CFD-based Design and Analysis of the Ventilation of an Electric Generator Model, Validated with Experiments	Hamed Jamshidi, Håkan Nilsson, Valery Chernoray	The efficiency of the ventilation system is a key point for durable and reliable electric generators. The design of such system requires a detailed understanding of the air flow in the generator. Computational fluid dynamics (CFD) has the potential to resolve the lack of information in this field. The present work analyses the air flow inside a generator model. The model is designed using a CFD-based approach, and manufactured by taking into consideration the experimental and numerical requirements and limitations. The emphasis is on the possibility to accurately predict and experimentally measure the flow distribution inside the stator channels. A major part of the work is focused on the design of an intake and a fan that gives an evenly distributed flow with a high flow rate. The intake also serves as an accurate flowmeter. Experimental results are presented, of the total volume flow rate, the total pressure and velocity distributions. Steady-state CFD simulations are performed using the FOAM-extend CFD toolbox. The simulations are based on the multiple rotating reference frames method. The results from the frozen rotor and mixing plane rotor-stator coupling approaches are compared. It is shown that the fan design provides a sufficient flow rate for the stator channels, which is not the case without the fan or with a previous fan design. The detailed experimental and numerical results show an excellent agreement, proving that the results reliable.	Experiments, CFD, Electric Generator and Ventilation
	Page	Title	Author	Abstract	Keywords
	124-131	Numerical Study of Important Factors for a Vortex Shedder using Automated Design Cycle	Su Myat Nyein, He Xu	The good performance of a vortex shedder is defined by strong and stable vortex generated under the condition of most linearity in Strouhal number and low power loss. In this paper, the flow past a bluff body of circular cylinder with a slit normal to the flow has been analyzed focusing on drag coefficient, linearity of Strouhal number and flow resistance (K-factor). The ANSYS/FLUENT package is used for flow simulation and the integration method of computational code to iSIGHT platform is employed for automated design cycle. This study results the design with (0.20~0.267) blockage ratio and 0.10 slit ratio as the best shedder for vortex flowmeter and this results are in well agreement with the experiment. As the combination of GAMBIT, FLUENT, and iSIGHT substitutes the design parameters automatically according to the input data, this method designs effectively the vortex shedder with less design cycle time and low manufacturing cost eliminating the human intervention bottleneck.	drag coefficient, linearity of Strouhal number, K-factor, integration of GAMBIT, FLUENT, iSIGHT
	132-141	CFD Analysis for Aligned and Misaligned Guide Vane Torque Prediction and Validation with Experimental Data	Christophe Devals, Thi C. Vu, François Guibault	This paper presents a CFD-based methodology for the prediction of guide vane torque in hydraulic turbine distributor for aligned and misaligned configurations. A misaligned or desynchronized configuration occurs when the opening angle of one guide vane differs from the opening angle of all other guide vanes, which may lead to a torque increase on neighbouring guide vanes. A fully automated numerical procedure is presented, that automates computations for a complete range of operation of a 2D or 3D distributor. Results are validated against laboratory measurements.	Torque, Guide Vane, CFD, Casing, Distributor
	142-154	3D Casing-Distributor Analysis for Hydraulic Design Application	Christophe Devals, Ying Zhang, Julien Dompierre, Thi C. Vu, Luca Mangani, François Guibault	Nowadays, computational fluid dynamics is commonly used by design engineers to evaluate and compare losses in hydraulic components as it is less expensive and less time consuming than model tests. For that purpose, an automatic tool for casing and distributor analysis will be presented in this paper. An in-house mesh generator and a Reynolds Averaged Navier-Stokes equation solver using the standard k- $\omega$ shear stress transport (SST) turbulence model will be used to perform all computations. Two solvers based on the C++ OpenFOAM library will be used and compared to a commercial solver. The performance of the new fully coupled block solver developed by the University of Lucerne and Andritz will be compared to the standard 1.6ext segregated simpleFoam solver and to a commercial solver. In this study, relative comparisons of different geometries of casing and distributor will be performed. The present study is thus aimed at validating the block solver and the tool chain and providing design engineers with a faster and more reliable analysis tool that can be integrated into their design process.	CFD, Casing-Distributor Analysis, OpenFOAM, Block Coupled Solver

155-168	Comparison of steady and unsteady simulation methodologies for predicting no-load speed in Francis turbines	Hossein Hosseinimanes, Christophe Devals, Bernd Nennemann, François Guibault	No-load speed is an important performance factor for the safe operation of hydropower systems. In turbine design, the manufacturers must conduct several model tests to calculate the accurate value of no-load speed for the complete range of operating conditions, which are expensive and time-consuming. The present study presents steady and unsteady methods for calculating no-load speed of a Francis turbine. The steady simulations are implemented using a commercial flow solver and an iterative algorithm that relies on a smooth relation between turbine torque and speed factor. The unsteady method uses unsteady RANS simulations that have been integrated with a user subroutine to retrieve the runner speed, time step and friction torque. The main goal of this research is to evaluate and compare the two methods by calculating turbine dynamic parameters for three test cases consisting of high and medium head Francis turbines. Overall, the numerical results agreed well with experimental data. The unsteady method provided more accurate results in the opening angle range from 20 to 26 degrees. Nevertheless, the steady results showed more consistency than unsteady results for the three different test cases at different operating conditions.	No-load speed, runaway speed, Francis turbine, steady-state simulation, unsteady state simulation	
169-182	Simulation model for Francis and Reversible Pump Turbines	Torbjørn K. Nielsen	When simulating the dynamic behaviour of a hydro power plant, it is essential to have a good representation of the turbine behaviour. The pressure transients in the system occurs because the flow changes, which the turbine defines. The flow through the turbine is a function of the pressure, the speed of rotation and the wicket gate opening and is, most often described in a performance diagram or Hill diagram. In the Hill diagram, the efficiency is drawn like contour lines, hence the name. A turbines Hill diagram is obtained by performance tests on scaled model in a laboratory. However, system dynamic simulations have to be performed in the early stage of a project, before the turbine manufacturer has been chosen and the Hill diagram is known. Therefore one have to rely on diagrams for a turbine with similar speed number. The Hill diagram is drawn through measured points, so for using the diagram in a simulation program, one have to iterate in the diagram based on curve fitting of the measured points. This paper describes an alternative method. By means of the Euler turbine equation, it is possible to set up two differential equations which represents the turbine performance with good enough accuracy for the dynamic simulations. The only input is the turbine's main geometry, the runner blade in- and outlet angle and the guide vane angle at best efficiency point of operation (BEP). In the paper, simulated turbine characteristics for a high head Francis turbine, and for a reversible pump turbine are compared with laboratory measured characteristics.	Hydropower, Turbines, Characteristics, Simulation	
183-192	Comparative study of sediment erosion on alternative designs of Francis runner blade	Bidhan Rajkarnikar, Dr. Hari P. Neopane, Biraj S. Thapa	The aim of this study was comparative analysis of sediment-induced erosion on optimized design and traditional design of Francis runner blade. The analysis was conducted through laboratory experiments in a test rig called Rotating Disc Apparatus. The results showed that the extent of erosion was significantly less in the optimized design when compared based on the material loss. It was observed that the optimized design could reduce sediment erosion by about 14.4% if it was used in place of the reference design for entire duration of the experiment. Based on the observations and results obtained, it has been concluded that the optimization of hydraulic design of blade profile of Francis runner can significantly reduce the effect of sediment-induced erosion.	Francis turbine, optimized design, rotating disc apparatus, runner blade, sediment erosion, wear pattern	
193-201	Leakage Flow Influence on SHF pump model performances	Patrick Dupont, Annie-Claude Bayeul-Lainé, Antoine Dazin, Gérard Bois, Olivier Rousselet, Qiaorui Si	This paper deals with the influence of leakage flow existing in SHF pump model on the analysis of internal flow behaviour inside the vane diffuser of the pump model performance using both experiments and calculations. PIV measurements have been performed at different hub to shroud planes inside one diffuser channel passage for a given speed of rotation and various flow rates. For each operating condition, the PIV measurements have been triggered with different angular impeller positions. The performances and the static pressure rise of the diffuser were also measured using a three-hole probe. The numerical simulations were carried out with Star CCM+ 9.06 code (RANS frozen and unsteady calculations). Some results were already presented at the XXth IAHR Symposium for three flowrates for RANS frozen and URANS calculations. In the present paper, comparisons between URANS calculations with and without leakages and experimental results are presented and discussed for these flow rates. The performances of the diffuser obtained by numerical calculations are compared to those obtained by the three-holes probe measurements. The comparisons show the influence of fluid leakages on global performances and a real improvement concerning the efficiency of the diffuser, the pump and the velocity distributions. These results show that leakage is an important parameter that has to be taken into account in order to make improved comparisons between numerical approaches and experiments in such a specific model set up.	pump, numerical calculations, performances, unsteady flow, vanned diffuser, radial flow pump	
202-208	Numerical Investigation of Pressure Fluctuation Reducing in Draft Tube of Francis Turbines	W F Li, J J Feng, H Wu, J L Lu, W L Liao, X Q Luo	For a prototype turbine operating under part load conditions, the turbine output is fluctuating strongly, leading to the power station incapable of connecting to the grid. The field test of the prototype turbine shows that the main reason is the resonance between the draft tube vortex frequency and the generator natural vibration frequency. In order to reduce the fluctuation of power output, different measures including the air admission, water admission and adding flow deflectors in the draft tube are put forward. CFD method is adopted to simulate the three-dimensional unsteady flow in the Francis turbine, to calculate pressure fluctuations in draft tube under three schemes and to compare with the field test result of the prototype turbine. Calculation results show that all the three measures can reduce the pressure pulsation amplitude in the draft tube. The method of water supply and adding flow deflector both can effectively change the frequency and avoid resonance, thus solving the output fluctuation problem. However, the method of air admission could not change the pressure fluctuation frequency.	Francis turbine, draft tube vortex, air supply, water supply, flow deflector	
209-219	Physics-based Surrogate Optimization of Francis Turbine Runner Blades, Using Mesh Adaptive Direct Search and Evolutionary Algorithms	Salman Bahrami, Christophe Tribes, Sven von Fellenberg, Thi C. Vu, François Guibault	A robust multi-fidelity optimization methodology has been developed, focusing on efficiently handling industrial runner design of hydraulic Francis turbines. The computational task is split between low- and high-fidelity phases in order to properly balance the CFD cost and required accuracy in different design stages. In the low-fidelity phase, a physics-based surrogate optimization loop manages a large number of iterative optimization evaluations. Two derivative-free optimization methods use an inviscid flow solver as a physics-based surrogate to obtain the main characteristics of a good design in a relatively fast iterative process. The case study of a runner design for a low-head Francis turbine indicates advantages of integrating two derivative-free optimization algorithms with different local- and global search capabilities.	Physics-based surrogate optimization, Francis turbine runner blade, multi-fidelity algorithm	
220	Erratum to: Numerical Investigation of Pressure Fluctuation Reducing in Draft Tube of Francis Turbines	W F Li, J J Feng, H Wu, J L Lu, W L Liao, X Q Luo	An error has been found in the footnote in page 202.		
	Page	Title	Author	Abstract	Keywords



221-229	The Performance Analysis Method with New Pressure Loss and Leakage Flow Models of Regenerative Blower	Chan Lee, Hyun Gwon Kil, Kwang Yeong Kim	For efficient design process of regenerative blower, the present study provides new generalized pressure and leakage flow loss models, which can be used in the performance analysis method of regenerative blower. The present performance analysis on designed blower is made by incorporating momentum exchange theory between impellers and side channel with mean line analysis method, and its pressure loss and leakage flow models are generalized from the related fluid mechanics correlations which can be expressed in terms of blower design variables. The present performance analysis method is applied to four existing models for verifying its prediction accuracy, and the prediction and the test results agreed well within a few percentage of relative error. Furthermore, the present performance analysis method is also applied in developing a new blower used for fuel cell application, and the newly designed blower is manufactured and tested through chamber-type test facility. The performance prediction by the present method agreed well with the test result and also with the CFD simulation results. From the comparison results, the present performance analysis method is shown to be suitable for the actual design practice of regenerative blower.	Regenerative Blower, Performance Analysis, Pressure Loss, Leakage Flow, CFD
230-239	Study on Flow Instability and Countermeasure in a Draft tube with Swirling flow	Takahiro Nakashima, Ryo Matsuzaka, Kazuyoshi Miyagawa, Koichi Yonezawa, Yoshinobu Tsujimoto	The swirling flow in the draft tube of a Francis turbine can cause the flow instability and the cavitation surge and has a larger influence on hydraulic power operating system. In this paper, the cavitating flow with swirling flow in the diffuser was studied by the draft tube component experiment, the model Francis turbine experiment and the numerical simulation. In the component experiment, several types of fluctuations were observed, including the cavitation surge and the vortex rope behaviour by the swirling flow. While the cavitation surge and the vortex rope behaviour were suppressed by the aeration into the diffuser, the loss coefficient in the diffuser increased by the aeration. In the model turbine test the aeration decreased the efficiency of the model turbine by several percent. In the numerical simulation, the cavitating flow was studied using Scale-Adaptive Simulation (SAS) with particular emphasis on understanding the unsteady characteristics of the vortex rope structure. The generation and evolution of the vortex rope structures have been investigated throughout the diffuser using the iso-surface of vapor volume fraction. The pressure fluctuation in the diffuser by numerical simulation confirmed the cavitation surge observed in the experiment. Finally, this pressure fluctuation of the cavitation surge was examined and interpreted by CFD.	Draft tube, Cavitation, Hydro turbine, Vortex rope, Aeration
240-253	Investigation of Cavitation Models for Steady and Unsteady Cavitating Flow Simulation	Tan Dung Tran, Bernd Nennemann, Thi Cong Vu, François Guibault	The objective of this paper is to evaluate the applicability of mass transfer cavitation models and determine appropriate numerical parameters for cavitating flow simulations. CFD simulations were performed for a NACA66 hydrofoil at cavitation numbers of 1.49 and 1.00, corresponding to steady sheet and unsteady sheet/cloud cavitating regimes using the Kubota and Merkle cavitation models. The Merkle model was implemented into CFX by User Fortran code. The Merkle cavitation model is found to give some improvements for cavitating flow simulation results for these cases. Turbulence modeling is also found to have an important contribution to the prediction quality of the simulations. The relationship between the turbulence viscosity modification, in order to take into account the local compressibility at the vapor/liquid interfaces, and the predicted numerical results is clarified. The limitations of current cavitating flow simulation techniques are discussed throughout the paper.	Cavitation, CFD, steady, unsteady flow simulation, turbulence, cavitation local compressibility
254-263	The effect of materials properties on the reliability of hydraulic turbine runners	Denis Thibault, Martin Gagnon, Stéphane Godin	The failure of hydraulic turbine runners is a rare event. So in order to assess the reliability of these components one cannot rely solely on the number of observed failures in a given population. However, as there is a limited number of degradation mechanisms involved, it is possible to use physically-based reliability models. Such models are often more complicated but are able to account for physical parameters in the degradation process. They can therefore help provide solutions to improve reliability. With such models, the effect of materials properties on runner reliability can be highlighted. This paper presents a brief review of the Kitagawa-Takahashi diagram which links the damage tolerance approach, based on fracture mechanics, to the stress or strain-life approaches. Using simplified response spectra based on runner stress measurements, we will show how fatigue reliability is sensitive to materials fatigue properties, namely fatigue crack propagation behaviour and fatigue limit obtained on S-N curves. Furthermore, we will review the influence of the main microstructural features observed in 13%Cr-4%Ni stainless steels commonly used for runner manufacturing. The goal is ultimately to identify the most influential microstructural features and to quantify their effect on fatigue reliability of runners.	fatigue, reliability, stainless steel, hydraulic turbine, turbine runner, microstructure
264-273	The detection of cavitation in hydraulic machines by use of ultrasonic signal analysis	P. Gruber, M. Farhat, P. Odermatt, M. Etterlin, T. Lerch, M. Frei	This presentation describes an experimental approach for the detection of cavitation in hydraulic machines by use of ultrasonic signal analysis. Instead of using the high frequency pulses (typically 1MHz) only for transit time measurement different other signal characteristics are extracted from the individual signals and its correlation function with reference signals in order to gain knowledge of the water conditions. As the pulse repetition rate is high (typically 100Hz), statistical parameters can be extracted of the signals. The idea is to find patterns in the parameters by a classifier that can distinguish between the different water states. This classification scheme has been applied to different cavitation sections: a sphere in a water flow in circular tube at the HSLU in Lucerne, a NACA profile in a cavitation tunnel and two Francis model test turbines all at LMH in Lausanne. From the signal raw data several statistical parameters in the time and frequency domain as well as from the correlation function with reference signals have been determined. As classifiers two methods were used: neural feed forward networks and decision trees. For both classification methods realizations with lowest complexity as possible are of special interest. It is shown that two to three signal characteristics, two from the signal itself and one from the correlation function are in many cases sufficient for the detection capability. The final goal is to combine these results with operating point, vibration, acoustic emission and dynamic pressure information such that a distinction between dangerous and not dangerous cavitation is possible.	ultrasonic signals, cavitation, turbine, neural network, decision tree

4, October-December, 2015	274-282	Leakage Flow Influence on SHF pump model performances	Patrick Dupont, Annie-Claude Bayeul-Lainé, Antoine Dazin, Gérard Bois, Olivier Roussette, Qiaorui Si	This paper deals with the influence of leakage flow existing in SHF pump model on the analysis of internal flow behaviour inside the vane diffuser of the pump model performance using both experiments and calculations. PIV measurements have been performed at different hub to shroud planes inside one diffuser channel passage for a given speed of rotation and various flow rates. For each operating condition, the PIV measurements have been triggered with different angular impeller positions. The performances and the static pressure rise of the diffuser were also measured using a three-hole probe. The numerical simulations were carried out with Star CCM+ 9.06 code (RANS frozen and unsteady calculations). Some results were already presented at the XXth IAHR Symposium for three flowrates for RANS frozen and URANS calculations. In the present paper, comparisons between URANS calculations with and without leakages and experimental results are presented and discussed for these flow rates. The performances of the diffuser obtained by numerical calculations are compared to those obtained by the three-holes probe measurements. The comparisons show the influence of fluid leakages on global performances and a real improvement concerning the efficiency of the diffuser, the pump and the velocity distributions. These results show that leakage is an important parameter that has to be taken into account in order to make improved comparisons between numerical approaches and experiments in such a specific model set up.	pump, numerical calculations, performances, unsteady flow, vaned diffuser, radial flow pump
	283-293	Quantitative and qualitative analysis of the flow field development through T99 draft tube caused by optimized inlet velocity profiles.	Sergio Galván, Marcelo Reggio, François Guibault, Gildardo Solorio	The effect of the inlet swirling flow in a hydraulic turbine draft tube is a very complex phenomenon, which has been extensively investigated both theoretically and experimentally. In fact, the finding of the optimal flow distribution at the draft tube inlet in order to get the best performance has remained a challenge. Thus, attempting to answer this question, it was assumed that through an automatic optimization process a Genetic Algorithm would be able to manage a parameterized inlet velocity profile in order to achieve the best flow field for a particular draft tube. As a result of the optimization process, it was possible to obtain different draft-tube flow structures generated by the automatic manipulation of parameterized inlet velocity profiles. Thus, this work develops a qualitative and quantitative analysis of these new draft tube flow field structures provoked by the redesigned inlet velocity profiles. The comparisons among the different flow fields obtained clearly illustrate the importance of the flow uniformity at the end of the conduit. Another important aspect has been the elimination of the re-circulating flow area which used to promote an adverse pressure gradient in the cone, deteriorating the pressure recovery effect. Thanks to the evolutionary optimization strategy, it has been possible to demonstrate that the optimized inlet velocity profile can suppress or mitigate, at least numerically, the undesirable draft tube flow characteristics. Finally, since there is only a single swirl number for which the objective function has been minimized, the energy loss factor might be slightly affected by the flow rate if the same relation of the axial-tangential velocity components is maintained, which makes it possible to scale the inlet velocity field to different operating points.	Numerical optimization, Draft Tubes, Hydraulic Turbines, Genetic Algorithm
	294-303	Selection of Optimal Number of Francis Runner Blades for a Sediment Laden Micro Hydropower Plant in Nepal	Binaya Baidar, Sailesh Chitrakar, Ravi Koirala, Hari Prasad Neopane	The present study is conducted to identify a better design and optimal number of Francis runner blades for sediment laden high head micro hydropower site, Tara Khola in the Baglung district of Nepal. The runner is designed with in-house code and Computational Fluid Dynamics (CFD) analysis is performed to evaluate the performance with three configurations; 11, 13 and 17 numbers of runner blades. The three sets of runners were also investigated for the sediment erosion tendency. The runner with 13 blades shows better performance at design as well as in variable discharge conditions. 96.2% efficiency is obtained from the runner with 13 blades at the design point, and the runners with 17 and 11 blades have 88.25% and 76.63% efficiencies respectively. Further, the runner with 13 blades has better manufacturability than the runner with 17 blades as it has long and highly curved blade with small gaps between the blades, but it comes with 65% more erosion tendency than in the runner with 17 blades.	CFD, Francis turbine, Sediment erosion, Efficiency, Performance, Optimum number of blades
	304-310	Numerical simulation of slit wall effect on the Taylor vortex flow with radial temperature gradient	Dong Liu, Chang-qing Chao, Fang-neng Zhu, Xi-qiang Han, Cheng Tang	Numerical simulation was applied to investigate the Taylor vortex flow inside the concentric cylinders with a constant radial temperature gradient. The reliability of numerical simulation method was verified by the experimental results of PIV. The radial velocity and temperature distribution in plain and 12-slit model at different axial locations were compared, and the heat flux distributions along the inner cylinder wall at different work conditions were obtained. In the plain model, the average surface heat flux of inner cylinder increased with the inner cylinder rotation speed. In slit model, the slit wall significantly changed the distribution of flow field and temperature in the annulus gap, and the radial flow was strengthened obviously, which promoted the heat transfer process at the same working condition.	Taylor vortex flow, Slit wall, temperature gradient, heat transfer, numerical simulation
	311-317	Correction and Experimental Verification of Velocity Circulation in a Double-blade Pump Impeller Outlet	Wang Kai, Liu Qiong	It is difficult to calculate velocity circulation in centrifugal pump impeller outlet accurately. Velocity circulations of a double-blade pump impeller outlet were calculated with Stodola formula, Weisner formula and Stechkin formula. Simultaneously, the internal flow of impeller for the double-blade pump were measured with PIV technology and average velocity circulations at the 0.8, 1.0 and 1.2 times of design flow were obtained. All the experimental values were compared with the above calculation values at the three conditions. The results show that calculation values of velocity circulations with Weisner formula is close to the experimental values. On the basis of the above, velocity circulations of impeller outlet were corrected. The results of experimental verification show that the corrected calculation errors, whose maximum error is 3.65%, are greatly reduced than the uncorrected calculation errors. The research results could provide good references for establishment of theoretical head and multi-condition hydraulic optimization of double-blade pumps.	Double-blade pump, Impeller, Velocity circulation, PIV, Correction
	318-326	Modeling and testing for hydraulic shock regarding a valve-less electro-hydraulic servo steering device for ships	Liao Jian, He Lin, Xu Rongwu	A valve-less electro-hydraulic servo steering device (short: VSSD) for ships was chosen as a study object, and its mathematic model of hydraulic shock was established on the basis of flow properties and force balance of each component. The influence of system structure parameters, changing rate of motor speed and external load on hydraulic shock strength was simulated by the method of numerical simulation. Experiment was designed to test the hydraulic shock mathematic model of VSSD. Experiment results verified the correctness of the model, and the model provided a correct theoretical method for the calculation and control of hydraulic shock of valve-less electro-hydraulic servo steering device.	valve-less, steering device, hydraulic shock, changing rate, accumulator, external load
	Page	Title	Author	Abstract	Keywords

1-16	Analytical Surge Behaviors in Systems of a Single-stage Axial Flow Compressor and Flow-paths	Nobuyuki Yamaguchi	Behaviors of surges appearing near the stall stagnation boundaries in various fashions in systems of a single-stage compressor and flow-path systems were studied analytically and were tried to put to order. Deep surges, which enclose the stall point in the pressure-mass flow plane, tend to have either near-resonant surge frequencies or subharmonic ones. The subharmonic surge is a multiple-loop one containing, for example, in a (1/2) subharmonic one, a deep surge loop and a mild surge loop, the latter of which does not enclose the stall point, staying only within the stalled zone. Both loops have nearly equal time periods, respectively, resulting in a (1/2) subharmonic surge frequency as a whole. The subharmonic surges are found to appear in a narrow zone neighboring the stall stagnation boundary. In other words, they tend to appear in the final stage of the stall stagnation process. It should be emphasized further that the stall stagnation initiates fundamentally at the situation where a volume-modified reduced resonant-surge frequency becomes coincident with that for the stagnation boundary conditions, where the reduced frequency is defined by the acoustical resonance frequency in the flow-path system, the delivery flow-path length and the compressor tip speed, modified by the sectional area ratio and the effect of the stalling pressure ratio. The real surge frequency turns from the resonant frequency to either near-resonant one or subharmonic one, and finally to stagnation condition, for the large-amplitude conditions, caused by the non-linear self-excitation mechanism of the surge.	Fluid Machine, Axial Flow Compressor, Surge, Analytical Simulation, Frequency, Fluid Dynamics
17-27	Genesis of Researches on Surges in Pumping Systems in Japan	Nobuyuki Yamaguchi, Yoshinobu Tsujimoto	Researches on the mechanism of surging and the surge behaviors in the systems of pumps, or fans or compressors, and the effects of flow-paths had been initiated and had made a great progress in Japan in the decades from the nineteen-forties to the nineteen-sixties. In 1947, the essential cause of the surges, i.e., self-excited oscillation nature of the flow-system, was discovered analytically by Professor Sumiji Fujii of Tokyo University, and most of the characteristic behaviors of the phenomena had been explained clearly. Successive studies by many other Japanese researchers continued to prove experimentally the mechanism, to extend the analytical studies, and to attempt preventing surge occurrence, etc. in the following two decades. The historical information on the early surge studies could be helpful to some concerned people. At the same time, the basic and plain ways of discussions and reasoning about the phenomena in the pioneering researches could give us much to be learned even in the present time of high-power computing systems. Regrettably, many of the original research works have been published only in Japanese. The present review introduces very briefly the situations in memories of the pioneering researchers and engineers.	Surge, pumps, compressors and fans, self-excited oscillation, pressure-flow characteristics
28-38	Robust Design of Main Control Valve for Hydraulic Pile Hammer Flexible Control System	Guo Yong, Hu Jun Ping, Zhang Long Yan	The flexible control system for hydraulic pile hammer using main control valve is present to the requirement of rapidly reversing with high frequency. To ensure the working reliability of hydraulic pile hammer, the reversing performance of the main control valve should commute robustness to various interfere factors. Through simulation model built in Simulink/Stateflow and experiment, the effects of relative parameters to reverse performance of main control are analyzed and the main interfere factors for reversing performance are acquired. Treating reverse required time as design objects, some structure parameters as control factors, control pressure, input flow and gaps between spool and valve body as interfere factors, the robust design of the main control valve is done. The combination of factors with the strongest anti-jamming capability is acquired which ensured the reliability and anti-jamming capability of the main control valve. It also provides guidance on design and application of the main control valve used in large flow control with interferences.	hydraulic pile hammer, main control valve, reversing performance, pile driving, Stateflow, robust design
39-46	Impact performance for high frequency hydraulic rock drill drifter with sleeve valve	Guo Yong, Yang Shu Yi, Liu De Shun, Zhang Long Yan, Chen Jian Wen	A high frequency hydraulic rock drill drifter with sleeve valve is developed to use on arm of excavator. In order to ensure optimal working parameters of impact system for the new hydraulic rock drill drifter controlled by sleeve valve, the performance test system is built using the arm and the hydraulic source of excavator. The evaluation indexes are gained through measurement of working pressure, supply oil flow and stress wave. The relations of working parameters to impact system performance are analyzed. The result demonstrates that the maximum impact energy of the drill drifter is 98.34J with impact frequency of 71HZ. Optimal pressure of YZ45 rock drill is 12.8 MPa-13.6MPa, in which the energy efficiency reaches above 58.6%, and feature moment of energy distribution is more than 0.650.	rock drill drift, sleeve valve, energy efficiency, impact energy, feature moment of energy distribution
47-55	Study on Performance Improvement of an Axial Flow Hydraulic Turbine with a Collection Device	Yasuyuki Nishi, Terumi Inagaki, Yanrong Li, Sou Hiram, Norio Kikuchi	The portable hydraulic turbine we previously developed for open channels comprises an axial flow runner with an appended collection device and a diffuser section. The output power of this hydraulic turbine was improved by catching and accelerating an open-channel water flow using the kinetic energy of the water. This study aimed to further improve the performance of the hydraulic turbine. Using numerical analysis, we examined the performances and flow fields of a single runner and a composite body consisting of the runner and collection device by varying the airfoil and number of blades. Consequently, the maximum values of input power coefficient of the Runner D composite body with two blades (which adopts the MEL031 airfoil and alters the blade angle) are equivalent to those of the composite body with two blades (MEL021 airfoil). We found that the Runner D composite body has the highest turbine efficiency and thus the largest power coefficient. Furthermore, the performance of the Runner D composite body calculated from the numerical analysis was verified experimentally in an open-channel water flow test.	Hydraulic Turbine, Runner, Collection Device, Airfoil, Performance, Flow Field
56	Erratum: The Flow Field of Undershot Cross-Flow Water Turbines Based on PIV Measurements and Numerical Analysis	Yasuyuki Nishi, Terumi Inagaki, Yanrong Li, Ryota Omiya, Kentaro Hatano	Errors have been found in Vol. 7 (2014) No. 4 (October-December) pp. 174-182.	
57-65	Numerical Analysis of Damping Effect of Liquid Film on Material in High Speed Liquid Droplet Impingement	Hirotohi Sasaki, Naoya Ochiai, Yuka Iga	By high speed Liquid Droplet Impingement (LDI) on material, fluid systems are seriously damaged, therefore, it is important for the solution of the erosion problem of fluid systems to consider the effect of material in LDI. In this study, by using an in-house fluid/material two-way coupled method which considers reflection and transmission of pressure, stress and velocity on the fluid/material interface, high-speed LDI on wet/dry material surface is simulated. As a result, in the case of LDI on wet surface, maximum equivalent stress are less than those of dry surface due to damping effect of liquid film. Empirical formula of the damping effect function is formulated with the fluid factors of LDI, which are impingement velocity, droplet diameter and thickness of liquid film on material surface.	Liquid droplet, Liquid film, Fluid/material coupled numerical method, Homogeneous model, Elastic body material

66-74	Numerical Simulation of Unsteady Cavitation in a High-speed Water Jet	Guoyi Peng, Kunihiro Okada, Congxin Yang, Yasuyuki Oguma, Seiji Shimizu	Concerning the numerical simulation of high-speed water jet with intensive cavitation this paper presents a practical compressible mixture flow method by coupling a simplified estimation of bubble cavitation and a compressible mixture flow computation. The mean flow of two-phase mixture is calculated by URANS for compressible fluid. The intensity of cavitation in a local field is evaluated by the volume fraction of gas phase varying with the mean flow, and the effect of cavitation on the flow turbulence is considered by applying a density correction to the evaluation of eddy viscosity. High-speed submerged water jets issuing from a sheathed sharp-edge orifice nozzle are treated with the cavitation number, $\sigma = 0.1$ , and the computation result is compared with experimental data. The result reveals that cavitation occurs initially at the entrance of orifice and bubble cloud develops gradually while flowing downstream along the shear layer. Developed bubble cloud breaks up and then sheds downstream periodically near the sheath exit. The pattern of cavitation cloud shedding evaluated by simulation agrees experimental one, and the possibility to capture the unsteadily shedding of cavitation clouds is demonstrated. The decay of core velocity in cavitating jet is delayed greatly compared to that in no-activation jet, and the effect of the nozzle sheath is demonstrated.	Submerged water jet, cavitation, bubble dynamics, two-phase flow, computational fluid dynamics	
75-84	Historical Perspective on Fluid Machinery Flow Optimization in an Industry	Akira Goto	Fluid-dynamic design of fluid machinery had heavily relied on empiricism and experimental observations for many years. Since 1980s, thanks to the advancements in Computational Fluid Dynamics (CFD), a variety of flow physics have been revealed. The contribution by CFD is indispensable; however, the challenge is required not only on the advancements in CFD technologies but also innovation of "design (optimization) technologies" because of the complex interactions between 3-D flow fields and the complex 3-D flow passage configurations, etc. This paper presents historical perspective on fluid machinery flow optimization in an industry with some messages for the future.	Design technology, Physical insight, Inverse design, Numerical optimization, Multi-objective, Adjoint	
85-94	Unsteady Wet Steam Flow Measurements in a Low-Pressure Test Steam Turbine	Chongfei Duan, Koji Ishibashi, Shigeki Senoo, Ilias Bosdas, Michel Mansour, Anestis I. Kalfas, Reza S. Abhari	An experimental study is conducted for unsteady wet steam flow in a four-stage low-pressure test steam turbine. The measurements are carried out at outlets of the last two stages by using a newly developed fast response aerodynamic probe. This FRAP-HTH probe (Fast Response Aerodynamic Probe - High Temperature Heated) has a miniature high-power cartridge heater with an active control system to heat the probe tip, allowing it to be applied to wet steam measurements. The phase-locked average results obtained with a sampling frequency of 200 kHz clarify the flow characteristics, such as the blade wakes and secondary vortices, downstream from the individual rotational blades in the wet steam environment.	Steam turbine, Wet steam, Unsteady flow measurement, Low-pressure stage	
95-106	Optimization of Blade Profile of a Plenum Fan	Lin Wu, Hua-Shu Dou, Yikun Wei, Yongning Chen, Wenbin Cao, Cunlie Ying	A method of optimization design for the blade profile of a centrifugal impeller by controlling velocity distribution is presented, and a plenum fan is successfully designed. This method is based on the inner flow calculation inside the centrifugal impeller, and is related to the distribution of relative velocity. The results show that after optimization, the boundary layer separation on the suction surface has been inhibited and the stability of plenum fan is improved. The flow at the impeller outlet is also studied, and the jet-wake pattern at the impeller outlet is improved obviously by optimization. The calculation result shows that the static pressure and static pressure efficiency can be increased by 15.4% and 21.4% respectively.	velocity distribution, blade profile, static pressure, static pressure efficiency	
	Page	Title	Author	Abstract	Keywords
107-118	Numerical And Experimental Study Of Single stage And Multistage Centrifugal Mixed Flow Submersible Borewell Pumps	C. Murugesan, Dr. R. Rudramoorthy	This paper focuses on the single stage and multistage performance characteristics of centrifugal mixed flow submersible borewell pump. This study reveals that the performance of single stage pump is higher than that of multistage pumps. The head, input power and efficiency of single stage pump are higher than the per stage head, per stage input power and efficiency of multistage pumps. This study is divided into three parts. In the first part, five prototype pumps were made in single stage and multistage construction and the performance tests were conducted. In the second part, numerical validation has been done for different turbulence models and grid sizes. k- $\Omega$ SST model has been selected for the performance simulation and was validated with the performance of the test pump with static pressure tappings. In the third part, single and three stage pump performance were simulated numerically and compared with experimental results. The detailed analysis of pressure and velocity distributions reveals the difference in performance of single and three stage pump, due to non-uniform flow and difference in averaged flow velocities at the subsequent impeller inlets except the 1st stage impeller inlet.	velocity distribution, blade profile, static pressure, static pressure efficiency	
119-128	Numerical Analysis of Unsteady Cavitating Flow around Balancing Drum of Multistage Pump	Milan Sedlár, Tomáš Krátký, Patrik Zima	This work presents the numerical investigation of an unsteady cavitating flow around a balancing drum of a multistage pump. The main attention is focused on the cavitation phenomena, which occur in the rear part of the drum clearance, cause the erosion of the drum material and influence the pressure losses and the flow rate through the clearance. The one-way coupling of the URANS equations and the full Rayleigh-Plesset equation is employed to analyse the flow field as well as the dynamics of cavitating bubbles. The numerical simulations show that the erosion processes are highly influenced by shaft vibrations, namely by periodic deformations of the annular clearance in time. The calculated results are verified by erosion tests on a real pump.	cavitation erosion, numerical simulation, multistage pump, balancing drum	
129-136	Performance Research of Counter-rotating Tidal Stream Power Unit	Xuesong Wei, Bin Huang, Pin Liu, Toshiaki Kanemoto	An experimental investigation was carried out to improve the performance of a counter-rotating type horizontal-axis tidal stream power unit. Front and rear blades were designed separately based on modified blade element momentum (BEM) theory, and their performances at different conditions of blade tip speed ratio were measured in a wind tunnel. Three different groups of blades were designed successively, and the results showed that Group3 possessed the highest power coefficient of 0.44 and was the most satisfactory model. This experiment shows that properly increasing diameter and reducing chord length will benefit the performance of the blade.	counter-rotating, tidal turbine, power coefficient, wind tunnel, BEM	
137-142	Numerical Analysis of Flow in Radial Turbine (Effects of Nozzle Vane Angle on Internal Flow)	Kenta OTSUKA, Tomoya KOMATSU, Hoshio TSUJITA, Satoshi YAMAGUCHI, Akihiro YAMAGATA	Variable Geometry System (VGS) is widely applied to the nozzle vane for the radial inflow turbine constituting automotive turbochargers for the purpose of optimizing the power output at each operating condition. In order to improve the performance of radial turbines with VGS, it is necessary to clarify the influences of the setting angle of nozzle vane on the internal flow of radial turbine. However, the experimental measurements are considered to be difficult for the flow in radial turbines because of the small size and the high rotational speed. In the present study, the numerical calculations were carried out for the flow in the radial turbine at three operating conditions by applying the corresponding nozzle vane exit angles, which were set up in the experimental study, as the inlet boundary condition. The numerical results revealed the characteristic flow behaviors at each operating condition.	Radial Turbine, Numerical Analysis, Turbocharger, Secondary Flow	

143-149	A Study on the Multi-Objective Optimization of Impeller for High-Power Centrifugal Compressor	Hyun-Su Kang, Youn-Jea Kim	In this study, a method for the multi-objective optimization of an impeller for a centrifugal compressor using fluid-structure interaction (FSI) and response surface method (RSM) was proposed. Numerical simulation was conducted using ANSYS CFX and Mechanical with various configurations of impeller geometry. Each design parameter was divided into 3 levels. A total of 15 design points were planned using Box-Behnken design, which is one of the design of experiment (DOE) techniques. Response surfaces based on the results of the DOE were used to find the optimal shape of the impeller. Two objective functions, isentropic efficiency and equivalent stress were selected. Each objective function is an important factor of aerodynamic performance and structural safety. The entire process of optimization was conducted using the ANSYS Design Explorer (DX). The trade-off between the two objectives was analyzed in the light of Pareto-optimal solutions. Through the optimization, the structural safety and aerodynamic performance of the centrifugal compressor were increased.	Centrifugal compressor, Shape optimization, Response surface method, Isentropic efficiency, Fluid-structure interaction	
150-158	Parametric Optimization of Vortex Shedder based on Combination of Gambit, Fluent and iSGHT	Su Myat Nyein, He Xu, Hongpeng YU	In this paper, a new framework that works the automatic execution with less design cycle time and human intervention bottlenecks is introduced to optimize the vortex shedder design by numerical integration method. This framework is based on iSGHT combined with the pre-processor GAMBIT, and flow analysis software FLUENT. Two vortex shedders, circular with slit and triangular- semi circular cylinder, are employed as the designed models to be optimized, and DOE driver is used for optimization. According to the essential properties of a vortex shedder, it has found that the best diameters are 30mm for circular cylinder with slit and 30 to 35 mm for tri-semi cylinder. For slit ratio, 0.1 and 0.15 are the optimized values for circular with slit and tri-semi cylinder respectively. And it is found that these optimal results generated by DOE automated design cycle are in well agreement with the experiment.	Optimization, vortex shedder, how to integrate the GAMBIT and FLUENT to iSGHT	
159	Erratum: Numerical Analysis of Damping Effect of Liquid Film on Material in High Speed Liquid Droplet Impingement	Hirotohi Sasaki, Naoya Ochiai, Yuka Iga	An error has been found in the footnote in Vol. 9 (2016) No. 1 (January-March) p. 57		
160-168	Dynamic Response of Blade Surface Cavitation	Masakazu Toyoshima, Kimiya Sakaguchi, Kota Tsubouchi, Hironori Horiguchi, Kazuyasu Sugiyama	In high speed turbopumps, cavitation occurs and often causes the flow instabilities such as cavitation surge and rotating cavitation. The occurrence of these cavitation instabilities is considered to relate to dynamic characteristics of the cavitation, which are modelled using a cavitation compliance and a mass flow gain factor. Various types of cavitation such as a blade surface cavitation, a tip leakage vortex cavitation, and a backflow vortex cavitation occur at the same time in the inducer and the dynamic characteristics of each cavitation have not been clarified yet in experiments. Focusing on the blade surface cavitation as one of fundamental cavitation, we investigated the dynamic characteristics of the blade surface cavitation on a flat plate hydrofoil in experiments in the present study.	Cavitation, Hydrofoil, Dynamic Characteristics, Experiment	
169-174	Performance and Flow Condition of Cross-Flow Wind Turbine with a Symmetrical Casing Having Side Boards	Toru Shigemitsu, Junichiro Fukutomi, Masaaki Toyohara	A cross-flow wind turbine has a high torque coefficient at a low tip speed ratio. Therefore, it is a good candidate for use as a self-starting turbine. Furthermore, it has low noise and excellent stability; therefore, it has attracted attention from the viewpoint of applications as a small wind turbine for an urban district. However, its maximum power coefficient is extremely low (10 %) as compared to that of other small wind turbines. In order to improve the performance and flow condition of the cross-flow rotor, the symmetrical casing with a nozzle and a diffuser are proposed and the experimental research with the symmetrical casing is conducted. The maximum power coefficient is obtained as $C_{pmax} = 0.17$ in the case with the casing and $C_{pmax} = 0.098$ in the case without the casing. In the present study, the power characteristics of the cross-flow rotor and those of the symmetrical casing with the nozzle and diffuser are investigated. Then, the performance and internal flow patterns of the cross-flow wind turbine with the symmetrical casings are clarified. After that, the effect of the side boards set on the symmetrical casing is discussed on the basis of the analysis results.	Cross-flow wind turbine, Power coefficient, Torque coefficient, Symmetrical casing, Numerical analysis	
175-181	PIV Measurement of Inlet and Outlet Flow of Contra-Rotating Small-Sized Cooling Fan	Toru Shigemitsu, Hiroaki Fukuda, Junichiro Fukutomi	Contra-rotating rotors have been adopted for some of the cooling fans to meet the demand for the high pressure and large flow rate. Therefore, it is important to clarify its inlet and outlet flows by experiments for the high performance and stable operation. PIV measurements were conducted at the design and partial flow rates. In the present paper, the inlet and outlet flow conditions of the contra-rotating small-sized cooling fan with a 40mm square casing are studied by using PIV measurement. Furthermore, improvements of the flow condition and design guideline to increase the performance were discussed based on the experimental results.	Cooling fan, PIV measurement, Internal flow, Contra-rotating rotor	
182-193	Effects of load variation on a Kaplan turbine runner	K. Amiri, B. Mulu, M.J. Cervantes, M. Raisee	Introduction of intermittent electricity production systems like wind and solar power to electricity market together with the deregulation of electricity markets resulted in numerous start/stops, load variations and off-design operation of water turbines. Hydraulic turbines suffer from the varying loads exerted on their stationary and rotating parts during load variations since they are not designed for such operating conditions. Investigations on part load operation of single regulated turbines, i.e., Francis and propeller, proved the formation of a rotating vortex rope (RVR) in the draft tube. The RVR induces pressure pulsations in the axial and rotating directions called plunging and rotating modes, respectively. This results in oscillating forces with two different frequencies on the runner blades, bearings and other rotating parts of the turbine. This study investigates the effect of transient operations on the pressure fluctuations exerted on the runner and mechanism of the RVR formation/mitigation. Draft tube and runner blades of the Porjus U9 model, a Kaplan turbine, were equipped with pressure sensors for this purpose. The model was run in off-cam mode during different load variations. The results showed that the transients between the best efficiency point and the high load occurs in a smooth way. However, during transitions to the part load a RVR forms in the draft tube which induces high level of fluctuations with two frequencies on the runner; plunging and rotating mode. Formation of the RVR during the load rejections coincides with sudden pressure change on the runner while its mitigation occurs in a smooth way.	Kaplan turbine, Runner pressure measurement, Load variations, Rotating vortex rope formation, Rotating vortex rope mitigation	
194-204	Rotordynamic Characteristics of Floating Ring Seals in Rocket Turbopumps	Yuichiro Tokunaga, Hideyuki Inoue, Jun Hiromatsu, Tetsuya Iguchi, Yasuhiro Kuroki, Masaharu Uchiomi	Floating ring seals offer an opportunity to reduce leakage flows significantly in rotating machinery. Accordingly, they have been applied successfully to rotating machinery within the last several decades. For rocket turbopump applications, fundamental behavior and design philosophy have been revealed. However, further work is needed to explore the rotordynamic characteristics associated with rotor vibrations. In this study, rotordynamic forces for floating ring seals under rotor's whirling motions are calculated to elucidate rotordynamic characteristics. Comparisons between numerical simulation results and experiments demonstrated in our previous report are carried out. The three-dimensional Reynolds equation is solved by the finite-difference method to calculate hydrodynamic pressure distributions and the leakage flow rate. The entrance loss at the upstream inlet of the seal ring is calculated to estimate the Lomakin effect. The friction force at the secondary seal surface is also considered. Numerical simulation results showed that the rotordynamic forces of this type of floating ring seal are determined mainly by the friction force at the secondary seal surface. The seal ring is positioned almost concentrically relative to the rotor by the Lomakin effect. Numerical simulations agree quite well with the experimental results.	cryogenics, shaft-seal system, FRS, floating ring seal, dynamic characteristics, rotordynamic coefficient	
	Page	Title	Author	Abstract	Keywords

205-212	Efficiency Increase and Input Power Decrease of Converted Prototype Pump Performance	Masao Oshima	The performance of a prototype pump converted from that of its model pump shows an increase in efficiency brought about by a decrease in friction loss. As the friction force working on impeller blades causes partial peripheral motion on the outlet flow from the impeller, the increase in the prototype's efficiency causes also a decrease in its input power. This paper discusses results of analyses on the behavior of the theoretical head or input power of a prototype pump. The equation of friction-drag coefficient for a flat plate was applied for the analysis of hydraulic loss in impeller blade passages. It was revealed that the friction-drag of a flat plate could be, to a certain degree, substituted for the friction drag of impeller blades, i.e. as a means for analyzing the relationship between a prototype pump's efficiency increase and input power decrease.	Performance conversion, Model pump, Efficiency increase, Input power decrease, Scale effect
213-221	CFD and surrogates-based inducer optimization	Tomáš Krátký, Lukáš Zavadil, Vit Doubrava	Due to the nature of cavitation numerical analyses, computational optimization of a pump with respect to the cavitation properties is extremely demanding. In this paper it is shown how a combination of Transient Blade Row (TBR) method and some simplifications can be used for making the optimization process more efficient and thus possible on current generation of hardware. The aim of the paper is not the theory of hydraulic design. Instead, the practical aspects of numerical optimization are shown. This is done on an example of a radial pump and a combination of ANSYS CFX, ANSYS software tools and custom scripts is used. First, a comparison of TBR and fully-transient simulation is made. Based on the results, the TBR method is chosen and a parametric model assembled. Design of Experiment (DOE) table is computed and the results are used for sensitivity analysis. As the last step, the final design is created and computed as fully-transient. In conclusion, the results are discussed.	Design Optimization, CFD, Cavitation, Inducer, TBR, ANSYS CFX
222-228	Hydraulic Model Test of a Floating Wave Energy Converter with a Cross-flow Turbine	Sangyoon Kim, Byungha Kim, Joji Wata, Young-Ho Lee	Almost 70% of the earth is covered by the ocean. Extracting the power available in the ocean using a wave energy converter has been seen to be eco-friendly and renewable. This study focuses on developing a method for analyzing a wave energy device that uses a cross-flow turbine. The motion of the ocean wave causes an internal bi-directional flow of water and the cross-flow turbine is able to rotate in one direction. This device is considered of double-hull structure, and because of this structure, sea water does not come into contact with the turbine. Due to this, the problem of fouling on the turbine is avoided. This study shows specific relationship for wave length and several motions.	Ocean energy, Floating wave energy converter(WEC), Cross-flow turbine, Power take-off system(PTO), Double-hull structure, Hydraulic model test
229-236	Energetic analysis for optimization of a rotating equilateral triangular cooling channel with staggered square ribs	Mi-Ae Moon, Kwang-Yong Kim	Energetic analysis was introduced in optimization of a rotating equilateral triangular internal cooling channel with staggered square ribs to maximize the net exergy gain. The objective function was defined as the net exergy gain considering the exergy gain by heat transfer and exergy losses by friction and heat transfer process. The flow field and heat transfer in the channel were analysed using three-dimensional Reynolds-averaged Navier-Stokes equations under the uniform temperature condition. Shear stress transport turbulence model has been selected as a turbulence closure through the turbulence model test. Computational results for the area-averaged Nusselt number were validated compared to the experimental data. Three design variables, i.e., the angle of rib, the rib pitch-to-hydraulic diameter ratio and the rib width-to-hydraulic diameter ratio, were selected for the optimization. The optimization was performed at Reynolds number, 20,000. Twenty-two design points were selected by Latin hypercube sampling, and the values of the objective function were evaluated by the RANS analysis at these points. Through optimization, the objective function value was improved by 22.6% compared to that of the reference geometry. Effects of the Reynolds number, rotation number, and buoyancy parameter on the heat transfer performance of the optimum design were also discussed.	Energetic analysis, Optimization, Cooling channel, Heat transfer enhancement, Ribs
237-244	Analysis on Characteristic of Pressure Fluctuation in Hydraulic Turbine with Guide Vane	FengXia Shi, JunHu Yang, XiaoHui Wang	An unsteady three-dimensional simulation based on Reynolds time-averaged governing equation and RNG k- $\epsilon$ turbulence model, was presented for pump-as-turbine, the pressure fluctuation characteristic of hydraulic turbine with guide vane was obtained. The results show that the time domains of pressure fluctuation in volute change periodically and have identical cycles. In volute tongue and inlet pressure fluctuations are light, while in dynamic and static coupling interface pressure fluctuations are serious; In impeller blade region the pressure fluctuation of pressure surface are lighter than that of suction surface. The dominant frequencies of pressure fluctuation concentrate in low frequency region, and concentrate within 2 times of the blade passing frequency.	Hydraulic turbine with guide vane, Volute tongue, Dynamic and static coupling interface, Pressure fluctuation
245-255	Study on the Alternating Flow Hydraulics and Its New Potential Application in the Geotechnical Testing Field	Yong Sang, Ying Han, Fuhai Duan	The alternating flow hydraulics (AFH) had demonstrated the unique features in the past. One of the most well-known inventions was the hydraulic machine-gun synchronizer, which had become the standard equipment of airplane during World War I. The studies on the AFH between 1960 and 1980 had triggered many researchers' interests and reached the summit. The disadvantages of the AFH like low efficiency and cooling difficulty had prevented the further development. Few people are engaged in studying the AFH at present. However, the unique characteristics of the AFH inspire the researchers to continuously explore the new special suitable applications. The overviews of the AFH and the new potential application in the geotechnical testing field have been discussed in this paper. First, the research results of the AFH in the past have been summarized. Then, the classifications of the AFH have been introduced in detail according to the working principle, the number of hydraulic transmission pipelines and the mode of input energy. The advantages and the disadvantages of the AFH have been discussed. A novel potential suitable application in the soil test field has been presented at last. The detailed designing ideas of a new dynamic triaxial instrument have been given, which will be a more innovational and energy-saving plan according to the current studies. A series of simulation experiments have been done. The simulation results show that the proposed scheme for the new dynamic triaxial instrument is feasible. The paper work will also give some inspirations in the reciprocating motion control system.	alternating flow hydraulics, hydraulic transmission, dynamic triaxial instrument
256-264	Design and performance research of a mixed-flow submersible deep well pump	Qihua Zhang, Yuanhui Xu, Li Cao, Weidong Shi, Weigang Lu	To meet the demand of higher handling capacity, a mixed-flow submersible deep well pump was designed and tested. The main hydraulic components are made of plastics, which is free of erosion, light-weight, and environment-friendly. To simplify plastic molding process, and to improve productivity, an axial-radial guide vane was proposed. To clarify its effect on the performance, a radial guide vane and a space guide vane are developed as well. By comparison, the efficiency of the pump equipped with the axial-radial guide vane is higher than the radial guide vane and is lower than the space guide vane, and its high efficiency range is wide. The static pressure recovery of the axial guide vane is a bit lower than the space guide vane, but it is much larger than the radial guide vane. Taking the cost and molding complexity into consideration, the axial-radial guide vane is much economic, promoting its popularity for the moderate and high specific speed submersible deep well pumps.	submersible deep well pump, mixed-flow, axial-radial guide vane, design



265-276	Energy Efficient Design of a Jet Pump by Ensemble of Surrogates and Evolutionary Approach	Afzal Husain, Arihant Sonawat, Sarath Mohan, Abdus Samad	Energy systems working coherently in different conditions may not have a specific design which can provide optimal performance. A system working for a longer period at lower efficiency implies higher energy consumption. In this effort, a methodology demonstrated by a jet pump design and optimization via numerical modeling for fluid dynamics and implementation of an evolutionary algorithm for the optimization shows a reduction in computational costs. The jet pump inherently has a low efficiency because of improper mixing of primary and secondary fluids, and multiple momentum and energy transfer phenomena associated with it. The high fidelity solutions were obtained through a validated numerical model to construct an approximate function through surrogate analysis. Pareto-optimal solutions for two objective functions, i.e., secondary fluid pressure head and primary fluid pressure-drop, were generated through a multi-objective genetic algorithm. For the jet pump geometry, a design space of several design variables was discretized using the Latin hypercube sampling method for the optimization. The performance analysis of the surrogate models shows that the combined surrogates perform better than a single surrogate and the optimized jet pump shows a higher performance. The approach can be implemented in other energy systems to find a better design.	Ensemble of surrogates, Multi-objective optimization, Pareto-optimal designs, Evolutionary computation, Efficiency enhancement, Jet pump	
277-286	Numerical Prediction of Inlet Recirculation in Pumps	Andrej Lipej, Duško Mitruševski	The development of heavy-duty process pumps, usually based on various design criteria, depends on the pump's application. The most important criteria are Q-H, efficiency and NPSH characteristics. Cavitation due to inlet recirculation is not often one of the design criteria, although many problems in pump operation appear because of inlet recirculation, when the operation range is from 0.5-0.8 Qopt. The present paper shows that steady state CFD analysis of inlet recirculation can give quite good results for the design of new hydraulic shapes, which have been developed to expand operating range and to minimize the harmful influence of recirculation at part load. In this paper, the structures of inlet recirculation are presented, as well as detailed shapes of vortices between the blades for various operating regimes, axial velocity distribution at the impeller inlet, the experimental results of NPSH and efficiency characteristics of an existing and newly designed pump.	process pump, CFD, cavitation, flow recirculation	
	Page	Title	Author	Abstract	Keywords
287-299	Numerical Simulation of Erosive Wear on an Impact Sprinkler Nozzle Using a Remeshing Algorithm	Yuncheng Xu, Haijun Yan	In China, agricultural irrigation water often contains a lot of suspended sediment which may cause the nozzle wear. In this study, a new numerical simulation combining the Discrete Phase Model and a remeshing algorithm was conducted. The geometric boundary deformation caused by the erosion wear, was considered. The weight loss of the nozzle, the node displacement and the flow field were investigated and discussed. The timestep sensitivity analysis showed that the timestep is very critical in the erosion modeling due to the randomness and the discreteness of the erosion behavior. Based on the simulation results, the major deformation of the boundary wall due to the erosion was found at the corners between outlet portion and contraction portion. Based on this remeshing algorithm, the simulated erosion weight loss of the nozzle is 4.62% less compared with the case without boundary deformation. The boundary deformation changes the pressure and velocity distribution, and eventually changes the sediment distribution inside the nozzle. The average turbulence kinetic energy at the outlet orifice is found to decrease with the erosion time, which is believed to change the nozzle's spray performance eventually.	Remeshing algorithm, erosion wear, impact sprinkler nozzle, CFD, discrete phase model	
300-306	A Twin Impulse Turbine for Wave Energy Conversion -The Performance under Unsteady Airflow-	M M Ashrafal Alam, Hideki Sato, Manabu Takao, Shinya Okuhara, Toshiaki Setoguchi	A twin unidirectional impulse turbine for wave energy conversion has been suggested in our previous study, and the performance under unsteady flow has been investigated by quasi-steady analysis. In the present study, the performance of twin impulse turbine under unsteady flow condition has been investigated by unsteady analysis of Computational fluid dynamics. As a result, the mean efficiency of twin unidirectional impulse turbine under unsteady flow is lower than the maximum efficiency of unidirectional impulse turbine. Moreover, it is verified that airflow goes backward in the reverse turbine in low flow rates.	CFD, Fluid machinery, Oscillating water column, Twin-impulse turbine, Wave energy conversion	
307-312	Wells Turbine for Wave Energy Conversion -Effect of Trailing Edge Shape-	Katsuya Takasaki, Tomohiro Tsunematsu, Manabu Takao, M M Ashrafal Alam, Toshiaki Setoguchi	The present study reported of the use of special shaped blade to reduce the difference in pressure across the Wells turbine for wave energy conversion. The blade profile was composed of NACA0020 airfoils and trailing edge was notched like chevron. Experiments were performed investigating the influence of trailing edge shape on the turbine performance. Four notch depths were used to investigate the effect of depth of cut on the turbine performance. As results, by placing a notch-cut at the trailing edge of the blade, it was possible to reduce the pressure difference across the turbine without lowering the efficiency. In addition, the pressure difference substantially reduced at a constant rate with the increase of the cut ratio.	Chevron, Fluid machinery, Oscillating water column, Wave energy conversion, Wells turbine	
313-324	Centrifugal Impeller Blade Shape Optimization Through Numerical Modeling	Sayed Ahmed Imran Bellary, Abdus Samad	Surrogate model based shape optimization methodology to enhance performance of a centrifugal pump has been implemented in this work. Design variables, such as blade number and blade angles defining the pump impeller blade shape were selected and a three-level full factorial design approach was used for efficiency enhancement. A three-dimensional simulation using Reynolds-averaged Navier Stokes (RANS) equations for the performance analysis was carried out after designing the geometries of the impellers at the design points. Standard k-ε turbulence model was used for steady incompressible flow simulations. The optimized impeller incurred lower losses by shifting the trailing edge towards the impeller pressure side. It is observed that the surrogates are problem dependent and most accurate surrogate does not deliver the best design always.	Centrifugal impeller, Optimal design, Blade number, Blade angles, Multiple Surrogate models, Hydraulic efficiency	
325-331	Free Surface Vortex in a Rotating Barrel with Rods of Different Heights	Zhang Xiaoyue, Zhang Min, Chen Wanyu, Yang Fan, Guo Xueyan	A bathtub vortex above the outlet of a rotating barrel is simulated. By analyzing the Ekman layer theory, it can be found that the main flow circulation is inversely proportional to the thickness of Ekman layer. The thicker the Ekman boundary layer, the weaker the rotational strength and the shorter of the length of gas core is. According to this law, models of barriers with rods of different heights are established. The reduction of air-core length in this air entrainment vortex and weakening the strength of rotation field were achieved.	bathtub vortex, Ekman layer, tangential velocity, air core, circulation	
332-337	Experimental study on the performance of a turbocompound diesel engine with variable geometry turbocharger	Yong Yin, Zhengbai Liu, Weilin Zhuge, Rongchao Zhao, Yanting Zhao, Zhen Chen, Jiao Mi	Turbocompounding is a key technology to satisfy the future requirements of diesel engine's fuel economy and emission reduction. A turbocompound diesel engine was developed based on a conventional 11-Liter heavy-duty diesel engine. The turbocompound system includes a power turbine, which is installed downstream of a Variable Geometry Turbocharger (VGT) turbine. The impacts of the VGT rack position on the turbocompound engine performance were studied. An optimal VGT control strategy was determined. Experimental results show that the turbocompound engine using the optimal VGT control strategy achieves better performance than the original engine under all full load operation conditions. The averaged and maximum reductions of the brake specific fuel consumption (BSFC) are 3% and 8% respectively.	Diesel engine, Turbocompounding, Variable Geometry Turbocharger (VGT), Fuel economy	

4, October-December, 2016	338-353	A Comparison of Surge Behaviors in Multi-Stage and Single-Stage Axial Flow Compressors	Nobuyuki Yamaguchi	Information on the surge behaviors and stall stagnation boundaries for a nine-stage axial flow compressor are summarized on the basis of analytical data in comparison with those for a single-stage one, with attention to the pressure ratio effect. The general trends of the surge loop behaviors of the pressure-mass flow are similar for both compressors including the fact that the subharmonic surges tend to appear very near the stall stagnation boundaries. With respect to the nine-stage compressor, however, the mild loops in the subharmonic surges tend to be very small in size relative to the deep loops, and at the same time, insufficient surge recovery phenomenon, which is a kind of subharmonic surge, appears also far from the stagnation boundary for relatively short delivery flow-paths. The latter is found to be a rear-stage surge caused by unstalling and re-stalling of the rear stages with the front-stages kept in stall in the stalled condition of the whole compressor, which situation is caused by stage-wise mismatching in the bottom pressure levels of the in-stall multi-stage compressor. The fundamental information on the stall stagnation boundaries is given by a group of normalized geometrical parameters including relative delivery flow-path length, relative suction flow-path length, and sectional area-pressure ratio, and by another group of normalized frequency parameters including relative surge frequencies, modified reduced resonance frequencies, and modified reduced surge frequencies. Respective groups of the normalized parameters show very similar tendency of behaviors for the nine-stage compressor and the single-stage compressor. <i>The modified reduced resonance frequency could be the more reasonable parameter.</i>	Fluid Machine, Axial Flow Compressor, Surge, Analytical Simulation, Frequency, Fluid Dynamics
	354-361	Performance Study of Thrust Control Unit with the Various Geometric Shapes	Kyoung-Ryun Kim, Jong-Ho Park	This study aims to identify aerodynamic characteristics of the ramp tab, a mechanical deflector, by conducting a non-combustive experiment using compressed air and supersonic flow test equipment. With the ramp tabs installed symmetrically and asymmetrically on the outlet of the supersonic nozzle, the structure of the flow field, the thrust spoilage, the thrust deviation angle, and the lift/drag coefficients were derived and analyzed. The results show that the asymmetrically-installed ramp tabs are advantageous relative to the symmetrically-installed tabs in terms of the performance of thrust vector control, thrust deviation angle, and lift coefficient.	Mechanical deflector, Drag coefficients, Flow field, Pressure distribution, Fluid flow characteristics
	362-369	Optimal Design of Two-Dimensional Cascade with Shock-Free Inflow Criterion	Abdul Muis, Priyono Sutikno, Aryadi Soewono, Firmans Hartono	The shock-free inflow criterion applied in the development of two-dimensional cascade for turbomachinery design. The developed cascade analysis with potential flow calculation through a panel method has been used to determine the shock-free inflow condition. The combination between cascade analysis and PSO (particle swarm optimization) algorithm provides an opportunity to develop a diagram of a two-dimensional parameter cascade at various airfoil shapes. Analytical equations have been derived from the diagram that will facilitate the turbomachinery designer in applying the shock-free inflow criterion on their developed cascade. This method has been used to develop the very low head axial hydraulic turbine and provides excellent results of numerical and actual prototype performances.	Shock-free inflow, cascade, particle swarm optimization, surface vorticity, panel method
	370-381	Optimization of a Single-Channel Pump Impeller for Wastewater Treatment	Joon-Hyung Kim, Bo-Min Cho, Youn-Sung Kim, Young-Seok Choi, Kwang-Yong Kim, Jin-Hyuk Kim, Yong Cho	As a single-channel pump is used for wastewater treatment, this particular pump type can prevent performance reduction or damage caused by foreign substances. However, the design methods for single-channel pumps are different and more difficult than those for general pumps. In this study, a design optimization method to improve the hydrodynamic performance of a single-channel pump impeller is implemented. Numerical analysis was carried out by solving three-dimensional steady-state incompressible Reynolds-averaged Navier-Stokes equations using the shear stress transport turbulence model. As a state-of-the-art impeller design method, two design variables related to controlling the internal cross-sectional flow area of a single-channel pump impeller were selected for optimization. Efficiency was used as the objective function and was numerically assessed at twelve design points selected by Latin hypercube sampling in the design space. An optimization process based on a radial basis neural network model was conducted systematically, and the performance of the optimum model was finally evaluated through an experimental test. Consequently, the optimum model showed improved performance compared with the base model, and the unstable flow components previously observed in the base model were suppressed remarkably well.	Single-channel pump, Wastewater treatment, Design optimization, Computational Fluid Dynamics, Cross-sectional area control, Bezier curve
	382-393	Numerical Investigation of CuO-Water Nanofluid Flow and Heat Transfer across a Heated Square Cylinder	Lotfi Bouazizi, Said Turki	Flow over a bluff body is an attractive research field in thermal engineering. In the present study, laminar flow over a confined heated square cylinder using CuO-Water nanofluid is considered. Unsteady two-dimensional Navier-Stokes and energy equations are solved numerically using finite volume method (FVM). Recent correlations for the thermal conductivity and viscosity of nanofluids, which are function of nanoparticle volume fraction, temperature and nanoparticle diameter, have been employed. The results of numerical solution are obtained for Richardson number, nanoparticle volume fractions and nanoparticle diameters ranges of 0-1, 1-5% and 30-100 nm respectively for a fixed Reynolds number of $Re = 150$ . At a given volume concentration, the investigations reveal that the decreasing in size of nanoparticles produces an increase in heat transfer rates from the square cylinder and a decrease in amplitude of the lift coefficient. Also, the increment of Nusselt number is more pronounced at higher concentrations and higher Richardson numbers.	CuO-Water nanofluid, heated square cylinder, thermophysical properties, buoyancy force, heat transfer enhancement
	Page	Title	Author	Abstract	Keywords
	1-8	Computational Design of Bifurcation: A Case Study of Darundi Khola Hydropower Project	Ravi Koirala, Sailesh Chitrakar, Hari Prasad Neopane, Balendra Chhetri, Bhola Thapa	Bifurcation refers to wye division of penstock to divide the flow symmetrically or unsymmetrically into two units of turbine for maintaining economical, technical and geological substrates. Particularly, water shows irrelevant behavior when there is a sudden change in flow direction, which results into the transition of the static and dynamic behavior of the flow. Hence, special care and design considerations are required both hydraulically and structurally. The transition induced losses and extra stresses are major features to be examined. The research on design and analysis of bifurcation is one of the oldest topics related to R&D of hydro-mechanical components for hydropower plants. As far as the earlier approaches are concerned, the hydraulic designs were performed based on graphical data sheet, head loss considerations and the mechanical analysis through simplified beam approach. In this paper, the multi prospect approach for design of Bifurcation, incorporating the modern day's tools and technology is identified. The hydraulic design of bifurcation is a major function of dynamic characteristics of the flow, which is performed with CFD analysis for minimum losses and better hydraulic performances. Additionally, for the mechanical design, a simplified conventional design method as pre-estimation and Finite Element Method for a relevant result projections were used.	Vertical Axis Wind Turbine, Duct Flow Power Generation, Wind Energy, Butterfly Wind Turbine, Computational Fluid Dynamics
	9-18	Employing rotating vaneless diffuser to enhance the performance of plenum fan	Hua-Shu Dou, Lin Wu, Yikun Wei, Yongning Chen, Wenbin Cao, Cunlie Ying	Numerical simulation is carried out for flow characteristics in a plenum fan and the influence of the diameter ratio of the rotating vaneless diffuser on the performance of plenum fan is analyzed. The diameter ratio of the rotating vaneless diffuser employed is from 1.03 to 1.3. The research results show that the rotating vaneless diffuser is able to enhance the performance of plenum fan. It is found that there is significant improvement in static pressure and efficiency at the diameter ratio of 1.05 at high flow coefficients, while the optimal diameter ratio is 1.2 at rated and low flow coefficient.	Plenum fan, rotating vaneless diffuser, static pressure, efficiency
	19-29	Experimental and Numerical Studies on the Possibility of Duct Flow Low-power Generation Using a Butterfly Wind Turbine	Yutaka Hara, Shohei Kogo, Katsuhiro Takagaki, Makoto Kawanishi, Takahiro Sumi, Shigeo Yoshida	An objective of this study is to demonstrate the validity of using a small wind turbine to recover the fluid energy flowing out of an exhaust duct for the generation of power. In these experiments, a butterfly wind turbine of a vertical axis type ( $D = 0.4$ m) is used. The output performance is measured at various locations relative to the exit of a small wind tunnel ( $W = 0.65$ m), representing the performance expected in an exhaust duct flow. Two-dimensional numerical analysis qualitatively agrees with the experimental results for the wind turbine power coefficient and rate of energy recovery. When the turbine is far from the duct exit (more than 2.5 D), an energy recovery rate of approximately 1.3% is obtained.	Vertical Axis Wind Turbine, Duct Flow Power Generation, Wind Energy, Butterfly Wind Turbine, Computational Fluid Dynamics

30-39	Effects of Volute Throat Enlargement and Fluid Viscosity on the Performance of an Over Hung Centrifugal Pump	Davoud Khoeiini, Alireza Riasi, Ali Shahmoradi	In the current study, identifying regimes and behaviors of the various viscous fluids in a typical horizontal singlestage centrifugal pump and improving its performance by enhancing volute throat area have been surveyed numerically and experimentally. Indeed the initial pump had insufficient head at BEP (Best Efficient Point) in relevant applications. In order to solve this problem, the method of increasing the volute throat area on the prototype was used in steps and eventually the increased head values have been achieved. Then modified centrifugal pump, that has been constructed based on the modified control volume from numerical results, has been tested thoroughly. The maximum head and efficiency discrepancy between numerical and experimental results in BEP were 1.4 and 2.6% respectively. The effects of viscous fluids, from 1 cSt to 500 cSt, on the performance curves of centrifugal pump have been investigated as well and results showed that viscous fluids has significant effect on them. Indeed the highest head and efficiency in the same conditions at BEP has been obtained in viscosity 1 cSt which was by 19.2% and 44% greater than the viscosity 500 cSt. It is also found that the highest viscous fluid had the highest energy consumption as the absorbed power of highest viscous fluid, 500 cSt, increased up to approximately 55% above the lowest viscous fluid, 1 cSt, values.	Centrifugal pump, Numerical and Experimental Study, Performance curve, Volute Throat Area, Increased Head, Viscosity Effects
40-46	Development of Life Test Equipment with Real Time Monitoring System for Butterfly Valves	Gi-Chun Lee, Byung-Oh Choi, Young-Bum Lee, Jong-Won Park, Tae-Yeon Nam, Keun-Won Song	Small valves including ball valves, gate valves and butterfly valves have been adopted in the fields of steam power generation, petrochemical industry, carriers, and oil tankers. Butterfly valves have normally been applied to fields where in narrow places installing the existing valves such as gate valves and ball valves have proven difficult due to the surrounding area and the heavier of these valves. Butterfly valves are used to control the mass flow of the piping system under low pressure by rotating the circular disk installed inside. The butterfly valve is benefitted by having simpler structure in which the flow is controlled by rotating the disc circular plate along the center axis, whereas the weight of the valve is light compared to the gate valve and ball valve above-mentioned, as there is no additional bracket supporting the valve body. The manufacturing company needs to acquire the performance and life test equipment, in the case of adopting the improving factors to detect leakage and damage on the seat of the valve disc. However, small companies, which are manufacturing the industrial valves, normally sell their products without the life test, which is the reliability test and environment test, because of financial and manpower problems. Furthermore, the failure mode analysis of the products failed in the field is likewise problematic as there is no system collecting the failure data on sites for analyzing the failures of valves. The analyzing and researching process is not arranged systematically because of the financial problem. Therefore this study firstly tried to obtain information about the failure data from the sites, analyzed the failure mode based on the field data collected from the customers, and then obtained field data using measuring equipment. Secondly, we designed and manufactured the performance and life test equipment which also have the real time monitoring system with the naked eye for the butterfly valves. The concept of this equipment can also be adopted by other valves, such as the ball valve, gate valve, and various others. It can be applied to variously sized valves, ranging from 25 mm to large sized valves exceeding 3000 mm. Finally, this study carries out the life test with square wave pressure, using performance and life test equipment. The	Butterfly valve, Life test, Real time monitoring system, Rated pressure, Leakage, Valve seat, Disc
47-53	Thrust Characteristics and Nozzle Role of Water Jet Propulsion	Yongyan Ni, Weimin Liu, Zhanhao Shen, Xiwei Pan	Surface pressure integration and momentum method were respectively performed to evaluate the impeller thrust and the system thrust of a contra-rotating axial flow water jet propulsion, and an interesting phenomenon so-called thrust paradox was revealed. To explain the paradox, the impeller thrust and the system thrust were physically and theoretically analyzed, the results show that the impeller thrust is head involved and is determined by the hydraulic parameters upstream and downstream the impeller, while the momentum method depicted by a classic equation is valid simply under the best efficiency point. Consequently, the role of a water jet propulsion nozzle was deduced that the nozzle is mainly to limit the flow rate that crosses the impeller and to assure the system working under the best efficiency condition apart from its ability to produce momentum difference. Related mathematical formula expressed the nozzle diameter is the dominant variable used to calculate the working condition of the water jet propulsion. Therefore the nozzle diameter can be steadily estimated by the former expression. The system thrust scaling characteristics under various speeds were displayed lastly	Water jet propulsion, Thrust paradox, Pump system, Nozzle
54-62	Performance Analysis of a Combined Blade Savonius Wind Turbines	Arifin Sanusi, Sudjito Soeparman, Slamet Wahyudi, Lilis Yulianti	The Savonius wind turbine has a lower performance than other types of wind turbines which may attract more study focus on this turbine. This study aimed to improve wind turbine performance by combining a conventional blade with an elliptical blade into a combined blade rotor. The analysis was performed on three blade models in computational fluid dynamics (CFD) using ANSYS Fluent Release 14.5. Then the results were verified experimentally using an open wind tunnel system. The results of the numerical simulation were similar to the experimental and showed that the combined blade rotor has better dragging flow and overlap flow than the conventional and elliptical blade. Experimental verification showed that the combined blade was to increase the maximum coefficient of power (Cpmax) by 11% of the conventional blade and to 5.5% of the elliptical blade.	Savonius turbine, combined blade, performance, CFD, experimental
63-75	A Design Method for Cascades Consisting of Circular Arc Blades with Constant Thickness	Tao Bian, Qianpeng Han, Martin Böhle	Many axial fans have circular arc blades with constant thickness. It is still a challenging task to calculate their performance, i.e. to predict how large their pressure rise and pressure losses are. For this task a need for cascade data exists. Therefore, the designer needs a method which works quickly for design purposes. In the present contribution a design method for such cascades consisting of circular arc blades with constant thickness is described. It is based on a singularity method which is combined with a CFD-data-based flow loss model. The flow loss model uses CFD-data to predict the total pressure losses. An interpolation method for the CFD-data are applied and described in detail. Data of measurements are used to validate the CFD-data and parameter variations are conducted. The parameter variations include the variation of the camber angle, pitch chord ratio and the Reynolds number. Additionally, flow patterns of two dimensional cascades consisting of circular arc blades with constant thickness are shown.	Cascade, Circular Arc Blades, CFD-data-based, loss model
76-85	Effect of deflected inflow on flows in a strongly-curved 90 degree elbow	Yukiharu Iwamoto, Ryo Kusuzaki, Motosuke Sogo, Kazunori Yasuda, Hidemasa Yamano, Masaaki Tanaka	Wall pressure measurements and flow visualization were conducted for a 90 degree elbow with an axis curvature radius the same as its inner diameter (125 mm). Reynolds numbers 320,000 and 500,000, based on the inner diameter and bulk velocity, were examined. A deflected inflow, having an almost constant velocity slope and a faster velocity at the inside, was introduced. Ensemble averaged pressure distributions showed that no difference of normalized pressure could be found in both the Reynolds number cases. Power spectral density functions of pressures exhibited that the fluctuation having the Strouhal number (based on the inner diameter and bulk velocity) of 0.6 existed in the downstream region of the elbow, which was 0.1 larger than that of the uniform inflow case [1]. Results of numerical calculations qualitatively coincided with the experimental ones.	90 degree elbow, deflected inflow, pressure measurements, power spectral density functions, flow visualization
86-98	Numerical Simulation on the Performance of Axial Vane Type Gas-Liquid Separator with Different Guide Vane Structure	Yang Fan, Liu Ailan, Guo Xueyan	In order to obtain high efficiency and low resistance droplet separation apparatus, axial vane type gas-liquid separators with different guide vanes were designed, and the RNG k-ε model as well as discrete phase model (DPM) were used to investigate the flow pattern inside the separators. It was shown that the tangential velocity distribution under different guide vanes have Rankine vortex characteristics, pressure distribution exhibits a high similarity which value becomes big as the increase of the blade outlet angle and the decrease of the guide vane numbers. The increase of the guide vane numbers and the decrease of the blade outlet angle could make separation improve significantly. The separation efficiency is almost 100% when the droplet diameter is bigger than 40μm.	axial vane type gas-liquid separator, numerical simulation, guide vane, separation efficiency

Page	Title	Author	Abstract	Keywords
------	-------	--------	----------	----------

99-118	An Outlook on Rotordynamic Pump Theory Development	Yongyan Ni, Zhongyong Pan	ECHO progress was defined to depict the rotordynamic pump theory development. Experience (E) era for pumps lasted nearly one and a half hundred years before the Industrial Revolution due to the low rotation speed of motor and undeveloped manufacture ability. Classic (C) theory referring to quasi-static performance as well as the items those were not able to be steadily resolved under the level were briefly and sophisticated outlined. Since 1962, flow instabilities and the dynamic responses had come into main attention with the development of the modern technologies such as ballistic missile, rocket and space shuttle main engine, and were finally heuristically (H) elucidated by talented scholars and researchers. Recently, new applications for the pumps open (O) to the surrounding fluid and diversity of the medium such as multiphase flow need more studies and some examples were briefly introduced to display the potential problems lastly.	rotordynamic pump, ECHO progress, quasi-static, flow instabilities, multiphase flow
119-137	A Study on the Fundamental Cause of Stall Stagnation Phenomena in Surges in Compressor Systems	Nobuyuki Yamaguchi	Although the stall stagnation phenomena have often been experienced in site and also analytically in numerical experiments in surges in systems of compressors and flow paths, the fundamental causes have not been identified yet. In order to clarify the situations, behaviours of infinitesimal disturbance waves superposed on a main flow were studied in a simplified one-dimensional flow model. A ratio of the amplifying rate of the system instability to the characteristic slope of the compressor element was surveyed as the instability enhancement factor. Numerical calculations have shown the following tendency of the factor. In the situation where both the sectional area ratio and the length ratio of the delivery flow-path to the suction duct are sufficiently large, the enhancement factors are greater in magnitude, which means occurrence of ordinary deep surges. However, in the situation where the area ratio and/or the length ratio is relatively smaller, the enhancement factor tends to lessen significantly, which situation tends to suppress deep surges for the same value of the characteristic slope. It could result in the stall stagnation condition. In the domain of area ratio vs. length ratio of the delivery duct to the suction duct, contour-lines of the enhancement factor behave qualitatively similar to those of the stall stagnation boundaries of a fan analytically obtained, suggesting that a certain range of the enhancement factor values could specify the stagnation occurrence. The significant decreases in the factors are observed to accompany appearances of phase lags and travelling waves in the wave motions, which macroscopically suggests breaking down of the complete surge actions of filling and emptying of the air in the delivery duct. The strength of the action is deeply related with acoustic interferences and is evaluated in terms of the volume-modified reduced resonance frequency proposed by the author. These observations	Compressor system, Surge, Stall, Stall stagnation, System instability, Acoustic waves
138-145	The Effect of Different Inflows on the Unsteady Hydrodynamic Characteristics of a Mixed Flow Pump	Long Yun, Wang Dezhong, Yin Junlian, Cai Youlin, Feng Chao	waterjet pump and, the channel head of steam generator which is directly connect with reactor coolant pump. Generally, pumps are identical designs and are selected based on performance under uniform inflow with the straight pipe, but actually non-uniform suction flow is induced by upstream equipment. In this paper, CFD approach was employed to analyze unsteady hydrodynamic characteristics of reactor coolant pumps with different inflows. The Reynolds-averaged Navier-Stokes equations with the k-ε turbulence model were solved by the computational fluid dynamics software CFX to conduct the steady and unsteady numerical simulation. The numerical results of the straight pipe and channel head were validated with experimental data for the heads at different flow coefficients. In the nominal flow rate, the head of the pump with the channel head decreases by 1.19% when compared to the straight pipe. The complicated structure of channel head induces the inlet flow non-uniform. The non-uniformity of the inflow induces the difference of vorticity distribution at the outlet of the pump. The variation law of blade to blade velocity at different flow rate and the difference of blade to blade velocity with different inflow are researched. The effects of non-uniform inflow on radial forces are absolutely different from the uniform inflow. For the radial forces at the frequency 1R, the corresponding amplitude of channel head are higher than the straight pipe at 1.0Φd and 1.2Φd flow rates, and the corresponding amplitude of channel head are lower than the straight pipe at 0.8Φd flow rates.	hydrodynamic, radial force, mixed flow pump, channel head, non-uniform
146-153	Sediment monitoring for hydro-abrasive erosion: A field study from Himalayas, India	Anant Kr. Rai, Arun Kumar	Sediment flow through hydropower components causes hydro-abrasive erosion resulting in loss of efficiency, interruptions in power production and downtime for repair/maintenance. Online instruments are required to measure/capture the variations in sediment parameters along with collecting samples manually to analyse in laboratory for verification. In this paper, various sediment parameters viz. size, concentration (TSS), shape and mineral composition relevant to hydro-abrasive erosion were measured and discussed with respect to a hydropower plant in Himalayan region, India. A multi-frequency acoustic instrument was installed at a desilting chamber to continuously monitor particle size distribution (PSD) and TSS entering the turbine during 27 May to 6 August 2015. The sediment parameters viz. TSS, size distribution, mineral composition and shape entering the turbine were also measured and analysed, using manual samples collected twice daily from hydropower plant, in laboratory with instruments based on laser diffraction, dynamic digital image processing, gravimetric method, conductivity, scanning electron microscope, X-ray diffraction and turbidity. The acoustic instrument was able to capture the variation in TSS; however, significant deviations were found between measured mean sediment sizes compared to values found in the laboratory. A good relation was found for turbidity ( $R^2 = 0.86$ ) and laser diffraction ( $R^2 = 0.93$ ) with TSS, which indicated that turbidimeter and laser diffraction instrument can be used for continuous monitoring of TSS at the plant. Total sediment load passed through penstock during study period was estimated to be 15,500 ton. This study shall be useful for researchers and hydropower managers in measuring/monitoring sediment for hydro-abrasive erosion study in hydropower plants.	Hydro-abrasive, erosion, sediment, turbidity, LISST, hydropower
154-163	Influence of Guide Vane Setting in Pump Mode on Performance Characteristics of a Pump-Turbine	Deyou Li, Hongjie Wang, Torbjørn K. Nielsen, Ruzhi Gong, Xianzhu Wei, Daqing Qin	Performance characteristics in pump mode of pump-turbines are vital for the safe and effective operation of pumped storage power plants. However, the head characteristics are different under different guide vane openings. In this paper, 3-D steady simulations were performed under 13mm, 19mm and 25mm guide vane openings. Three groups of operating points under the three GVOs were chosen based on experimental validation to investigate the influence of guide vane setting on flow patterns upstream and downstream. The results reveal that, the guide vane setting will obviously change the flow pattern downstream, which in turn influences the flow upstream. It shows a strong effect on hydraulic loss (power dissipation) in the guide and stay vanes. It is also found that the hydraulic loss mainly comes from the flow separation and vortices. In addition, in some operating conditions, the change of guide vane opening will change the flow angle at the runner inlet and outlet, which will change the Euler momentum (power input). The joint action of Euler momentum and hydraulic loss results in the change of the head characteristics	Pump-turbine, Guide vane setting, Performance characteristic, Euler theory, Hydraulic loss, Vortices

164-175	Characteristics of Synchronous and Asynchronous modes of fluctuations in Francis turbine draft tube during load variation	Rahul Goyal, Michel J. Cervantes, Bhupendra K. Gandhi	Francis turbines are often operated over a wide load range due to high flexibility in electricity demand and penetration of other renewable energies. This has raised significant concerns about the existing designing criteria. Hydraulic turbines are not designed to withstand large dynamic pressure loadings on the stationary and rotating parts during such conditions. Previous investigations on transient operating conditions of turbine were mainly focused on the pressure fluctuations due to the rotor-stator interaction. This study characterizes the synchronous and asynchronous pressure and velocity fluctuations due to rotor-stator interaction and rotating vortex rope during load variation, i.e. best efficiency point to part load and vice versa. The measurements were performed on the Francis-99 test case. The repeatability of the measurements was estimated by providing similar movement to guide vanes twenty times for both load rejection and load acceptance operations. Synchronized two dimensional particle image velocimetry and pressure measurements were performed to investigate the dominant frequencies of fluctuations, vortex rope formation, and modes (rotating and plunging) of the rotating vortex rope. The time of appearance and disappearance of rotating and plunging modes of vortex rope was investigated simultaneously in the pressure and velocity data. The asynchronous mode was observed to dominate over the synchronous mode in both velocity and pressure measurements.	Francis turbine, transient, particle image velocimetry, pressure measurements, load rejection, load acceptance, rotating vortex rope formation, synchronous and asynchronous modes	
176-187	Numerical Simulation and Experimental Research of the Flow Coefficient of the Nozzle-Flapper Valve Considering Cavitation	Lei Li, Changchun Li, Hengxuan Zhang	The nozzle-flapper valves are widely applied as a pilot stage in aerospace and military system. A subject of the analysis presented in this work is to find out a reasonable range of null clearance between the nozzle and flapper. This paper has presented a numerical flow coefficient simulation. In every design point, a parameterized model is created for flow coefficient simulation and cavitation under different conditions with varying gap width and inlet pressure. Moreover, a new test device has been designed to measure the flow coefficient and for visualized cavitation. The numerical simulation and test results both indicate that cavitation intensity gets fierce initially and shrinks finally as the gap width varies from small to large. From the curve, the flow coefficient mostly has experienced three stages: linear throttle section, transition section and saturation section. The appropriate deflection of flapper is recommended to make the gap width drop into the linear throttle section. The flapper-nozzle null clearance is optionally recommended near the range of DN/16. Finally through simulation it is also concluded that the inlet pressure plays a little role in the influence on the flow	Nozzle-flapper valve, Flow coefficient, Servo valve, Cavitation, Numerical simulation	
	Page	Title	Author	Abstract	Keywords
188-196	Physical and Numerical Investigation of Cavitating Flow-Induced Vibration of a Flexible Hydrofoil	Qin Wu, Guoyu Wang, Biao Huang		The objective of this paper is to investigate the flow-induced vibration of a flexible hydrofoil in cavitating flows via combined experimental and numerical studies. The experiments are presented for the modified NACA66 hydrofoil made of POM Polyacetate in the closed-loop cavitation tunnel and the numerical investigations are performed using a hybrid coupled fluid structure interaction model. The results showed that with the decreasing of cavitation number, the vibration magnitude increases dramatically for the cloud cavitation and declines for the supercavitation. The cloud cavitation development strongly affects the vibration response, with the main frequency of the vibration being accordance with the cavity shedding frequency and other two frequencies corresponding to the bending and twisting frequencies.	Cloud cavitation, Cavitating flow-induced vibration, Flexible hydrofoil, Cavitation-excited pressure fluctuation, One-dimensional model
197-208	Cavitation Compliance in 1D Part-load Vortex Models	Peter K Dörfler		When Francis turbines operate at partial load, residual swirl in the draft tube causes low-frequency pulsation of pressure and power output. Scale effects and system response may bias the prediction of prototype behavior based on laboratory tests, but could be overcome by means of a 1D analytical model. This paper deals with the two most important features of such a model, the compliance and the source of excitation. In a distributed-parameter version, compliance should be represented as an exponential function of local pressure. Lack of similarity due to different Froude number can thus be compensated. The normally unknown gas content in the vortex cavity has significant influence on the pulsation, and should therefore be measured and considered as a test parameter.	Francis turbine, model test, draft tube vortex, scale effect, cavitation, similarity
209-217	Novel methods of increasing the storage volume at Pumped Storage Power plants	Pål-Tore Storli		The paper presents two novel concepts of increasing the energy storage capacity at pumped storage power plants, both existing and new projects. The concepts utilize compressed air as a working medium to displace water from a volume originally not available for storage. The concepts are likely to give additional storage volume at a low cost, however, much development and many investigations are needed before the concepts can be shown to be technical and economical feasible solutions for energy storage. The concepts are disclosed so that researchers and utilities can start those investigations, hopefully helping the green transition by providing highly valuable energy storage for a future renewable energy having a much higher share of renewable energies than the current systems.	Pumped Storage Power plants, Energy storage, Novel concepts, Renewable Energy
218-226	Numerical Investigation on Hydrodynamic Characteristics of a Centrifugal Pump with a Double Volute at Off-Design Conditions	Hyeon-Seok Shim, Kwang-Yong Kim		Severe radial thrust under off-design operating conditions can be a harmful factor for centrifugal pumps. In the present work, effects of geometry of a double volute casing on the hydrodynamic performance of a centrifugal pump have been investigated focusing on off-design conditions. Three-dimensional steady Reynolds-averaged Navier-Stokes analysis was carried out by using shear stress transport turbulence model. Numerical results for the hydrodynamic performance of the centrifugal pump were validated compared with experimental data. The hydraulic efficiency and radial thrust coefficient were used as performance parameters to evaluate the hydrodynamic characteristics of the centrifugal pump. The cross-sectional area ratio of the volute casing, the expansion coefficient of the rib structure, the distance between the rib starting point and volute entrance, and radius and width of the volute entrance, and length of the rib structure, were selected as geometric parameters. Results of the parametric study show that the performance parameters are significantly affected by the geometric variables and operating conditions. Optimal configurations of the double volute casing based on the design of experiments technique show outstanding performance in terms of the efficiency and radial thrust coefficient.	Centrifugal pump, Double volute casing, Radial thrust, Hydraulic efficiency, L18 orthogonal array
227-239	Evaluation of turbulence models in rough wall-boundary layers for hydroelectric applications	Rabjit Dutta, Jonathan Nicolle, Anne-Marie Giroux, Ugo Piomelli		The accuracy of turbulence models for the Reynolds-Averaged Navier-Stokes (RANS) equations in rough-wall flows is evaluated using data from large-eddy simulations (LES) of boundary layers with favourable and adverse pressure gradients. Some features of the flow (such as flow reversal in the roughness sublayer) cannot be captured accurately by any model, due to the fundamental model formulation. In mild pressure gradients most RANS models are sufficiently accurate for engineering applications, but if strong favourable or adverse pressure gradients	turbulence modelling, large-eddy simulations, roughness, boundary layers, separation

240-253	Effects of Latin hypercube sampling on surrogate modeling and optimization	Arshad Afzal, Kwang-Yong Kim, Jae-won Seo	Latin hypercube sampling is widely used design-of-experiment technique to select design points for simulation which are then used to construct a surrogate model. The exploration/exploitation properties of surrogate models depend on the size and distribution of design points in the chosen design space. The present study aimed at evaluating the performance characteristics of various surrogate models depending on the Latin hypercube sampling (LHS) procedure (sample size and spatial distribution) for a diverse set of optimization problems. The analysis was carried out for two types of problems: (1) thermal-fluid design problems (optimizations of convergent-divergent micromixer coupled with pulsatile flow and boot-shaped ribs), and (2) analytical test functions (six-hump camel back, Branin-Hoo, Hartman 3, and Hartman 6 functions). The three surrogate models, namely, response surface approximation, Kriging, and radial basis neural networks were tested. The important findings are illustrated using Box-plots. The surrogate models were analyzed in terms of global exploration (accuracy over the domain space) and local exploitation (ease of finding the global optimum point). Radial basis neural networks showed the best overall performance in global exploration characteristics as well as tendency to find the approximate optimal solution for the majority of tested problems. To build a surrogate model, it is recommended to use an initial sample size equal to 15 times the number of design variables. The study will provide useful guidelines on the effect of initial sample size and distribution on surrogate construction and subsequent optimization using LHS sampling plan.	Latin hypercube sampling, Optimization, Surrogate model, Cross-validation, Global Optimization	
254-263	Numerical Cavitation Intensity on a Hydrofoil for 3D Homogeneous Unsteady Viscous Flows	Christophe Leclercq, Antoine Archer, Regiane Fortes-Patella, Fabien Cerru	The cavitation erosion remains an industrial issue for many applications. This paper deals with the cavitation intensity, which can be described as the fluid mechanical loading leading to cavitation damage. The estimation of this quantity is a challenging problem both in terms of modeling the cavitating flow and predicting the erosion due to cavitation. For this purpose, a numerical methodology was proposed to estimate cavitation intensity from 3D unsteady cavitating flow simulations. CFD calculations were carried out using Code_Saturne, which enables U-RANS equations resolution for a homogeneous fluid mixture using the Merkle's model, coupled to a k-ε turbulence model with the Reboud's correction. A post-process cavitation intensity prediction model was developed based on pressure and void fraction derivatives. This model is applied on a flow around a hydrofoil using different physical (inlet velocities) and numerical (meshes and time steps) parameters. The article presents the cavitation intensity model as well as the comparison of this model with experimental results. The numerical predictions of cavitation damage are in good agreement with experimental results obtained by pitting test.	cavitation, cavitation intensity prediction, erosion	
264-273	Annual Energy Production Maximization for Tidal Power Plants with Evolutionary Algorithms	Evgenia Kontoleonos, Simon Weissenberger	In order to be able to predict the maximum Annual Energy Production (AEP) for tidal power plants, an AEP optimization tool based on Evolutionary Algorithms was developed by ANDRITZ HYDRO. This tool can simulate all operating modes of the units (bi-directional turbine, pump and sluicing mode) and provide the optimal plant operation that maximizes the AEP to the control system. For the Swansea Bay Tidal Power Plant, the AEP optimization evaluated all different hydraulic and operating concepts and defined the optimal concept that led to a significant AEP increase. A comparison between the optimal plant operation provided by the AEP optimization and the full load operating strategy is presented in the paper, highlighting the advantage of the method in providing the maximum AEP.	Annual Energy Production, tidal power plants, optimization, Evolutionary Algorithms, Swansea Bay	
287-295	Pressure Pulsation Characteristics of a Model Pump-turbine Operating in the S-shaped Region: CFD Simulations	Linsheng Xia, Yongguang Cheng, Fang Cai	The most detrimental pressure pulsations in high-head pump-turbines is caused by the rotor-stator interaction (RSI) between the guide vanes and runner blades. When the pump-turbine operates in the S-shaped region of the characteristic curves, the deteriorative flow structures may significantly strengthen RSI, causing larger pressure pulsations and stronger vibration with an increased risk of mechanical failure. CFD simulations were carried out to analyze the impacts of flow evolution on the pressure pulsations in the S-shaped region of a model pump-turbine. The results show that the reverse flow vortex structures (RFVS) at the runner inlet have regular development and transition patterns when discharge reduces from the best efficiency point (BEP). The RFVS first occur at the hub side, and then shift to the midspan near the no-load point, which cause the strongest pressure pulsations. The locally distributed RFVS at hub side enhance the local RSI and makes the pressure fluctuations at the corresponding sections stronger than those at the rest sections along the spanwise direction. Under the condition of RFVS at the mid-span, the smaller flow rate make the smaller difference of pressure pulsation amplitudes in the spanwise direction. Moreover, the rotating stall, rotating at 35.7%-62.5% of the runner rotational frequency, make the low frequency components of pressure pulsations distribute unevenly along the circumference in the vaneless space. However, it have little influence on the distributions of high components.	Rotor-stator interaction, Pressure pulsations, rotating stall, Pump-turbine, S-shaped characteristics	
296-306	How to Avoid Severe Incidents at Hydropower Plants	Masashi Yasuda, Satoshi Watanabe	Hydropower is now changing its role from the energy generator into the most powerful and reliable tool for stabilizing the electrical network, especially under the increase of intermittent power sources like wind-power and solar-power. Although the hydropower plants are the most robust generating facilities, they are not immune from unexpected severe incidents having long downtime, considerable restoration cost and sometimes fatalities. The present paper provides some study results about severe incidents in the conventional hydropower plants, mainly about the flood, fire and electromechanical troubles, except for the incidents of civil facilities. It also provides some possible scenarios which may lead some measures how to avoid such incidents. Finally, it provides some comprehensible recommendations to avoid severe incidents based on experiences.	hydropower plant, severe incident, flood, fire, turbine failure, generator failure	
307-317	Measurement of Dynamic Characteristics of an Inducer in Cavitating Conditions	Takuya Ashida, Keita Yamamoto, Koichi Yonezawa, Hironori Horiguchi, Yutaka Kawata, Yoshinobu Tsujimoto	In liquid-propellant rockets, POGO instability can occur, in which a fluctuation of propellant supply to the engine, a thrust fluctuation, and a structural vibration are coupled. For the prediction of this instability, it is required to provide dynamic characteristics of the pump represented as the transfer matrix correlating the upstream and downstream pressure and flow rate fluctuations. In the present study, the flow rate fluctuation is evaluated from the fluctuation of pressure difference at the different locations assuming that the fluctuation is caused by the inertia of the flow rate fluctuation. The experiments were performed in some flow conditions, and it was shown that the tendencies of dynamic characteristics are related to excitation frequencies, cavitation numbers and flow rate coefficients.	Inducer, Cavitation, Dynamic transfer matrix, Dynamic characteristics, Experiment, CFD	
	Page	Title	Author	Abstract	Keywords
318-327	Study on Influence of Blade Number on Aerodynamic Noise of Half-ducted Propeller Fans for Packaged Air-conditioners	Taku Iwase, Tetsushi Kishitani, Masato Furukawa	Flow fields in 2-blade and 4-blade half-ducted propeller fans for the outdoor units of air-conditioners were calculated with large eddy simulation based on finite element method with the aim of investigating the influence of blade number on aerodynamic noise. We confirmed that the tip vortex had a great influence on aerodynamic noise in half-ducted propeller fans. The length of the tip vortex trajectory and the blade pitch for the 2-blade propeller fan were longer than those for the 4-blade propeller fan. These were suppressed the interaction between the tip vortex and the adjacent blade in the 2-blade propeller fan. The 2-blade propeller fan was therefore more silent than the 4-blade propeller fan.	Half-ducted propeller fan, Tip vortex, Aerodynamic noise, Large eddy simulation	



328-335	Research on Temperature Current Drift Characteristics Test of Force Feedback Hydraulic Servo Valve	Lei Li, Hao Yan, Changchun Li	The servo valve is the key component in the electrical-hydraulic servo system. Status for the servo valve temperature drift characteristics of the test system is studied in this paper firstly. Since Research on temperature servo valve drift characteristics has not formed systematic theory. As for Position closed-loop test method, the theory and calculation algorithms of servo valve temperature drift are analyzed. Simulation and experiments both show the feasibility of the theory test, and by a large number of experiments, the servo valve drift results are put into classification. Finally, corresponding phenomenon is elaborated, pointing out its causes and appropriate preventive measures, hoping to reduce drift value influenced by the temperature on the servo valve.	fluid power transmission and control, servo valve, temperature drift characteristics, electrohydraulic control equipment
336-344	Feasibility Analysis of Existing Compressor Design for Different Refrigerants	Arihant Sonawat, Young-Seok Choi, Kyoung-Yong Lee, Hyeon-Mo Yang, Jin-Hyuk Kim	The destruction of ozone layer due to the use of refrigerants is a matter of concern in the present world. This has been addressed through various platforms and several guidelines have been framed for their usage to prevent its further degradation. Since the existing chlorofluorocarbon (CFC) and hydrofluorocarbon (HFC) refrigerants are at the verge of phasing out because of their higher environmental impacts, this has raised various questions about the systems which employ these refrigerants. One such question is: Will the existing systems work efficiently when the existing refrigerant is replaced by a viable environment friendly substitute? The present work makes an effort to answer this question.	Computational fluid dynamics, turbo chiller, refrigerant, global warming, centrifugal compressor
345-354	Blade passage modeling strategy for hydraulic turbine	Ruzhi Gong, Chirag Trivedi, Ole G. Dahlhaug, Torbjørn K. Nielsen	CFD has played a significant role to investigate the performance and improve the design of hydraulic turbines; however, the improvement of CFD method demands powerful computer resources including time, CPU, memory, and commercial licenses. In present work, both global and local parameters of a high head Francis turbine were studied using several geometrical and interface modelling approaches. The aim of the work is to find suitable strategy for designers to simulate the hydraulic turbines to balance the numerical accuracy and the requirement of computational resources. The geometrical modelling approaches include combinations of turbine components such as, spiral casing, distributor, runner and draft tube. The interface modelling approaches includes, stage, Frozen rotor and transient rotorstator types. The study showed that the proper combinations of both approaches can effectively reduce the numerical error.	Hydraulic turbine, modeling strategy, pressure fluctuation, rotor-stator interaction
355-362	Investigation of Pressure Pulsations and Flow Instabilities in a Centrifugal Pump at Part-load Conditions	Denghao Wu, Yun Ren, Jiegang Mou, Yunqing Gu	The flow instabilities and pressure pulsations can be generated during the pump's operation. However, it is even more serious under the part-load condition. Currently, the links between flow instabilities and pressure pulsations were still not fully understood. In the present study, the experimental investigation was performed on pressure pulsations by utilizing the dynamic pressure transducers and the internal unsteady flow structures were measured by PIV. The pressure pulsations were extracted at 6 different locations around the volute under different flow rates conditions ranging from 10% to 120% of the nominal flow rate. The study allowed relating the pressure pulsations and unsteady flow structures in a pump. It was noted that higher intensive broadband pressure pulsations can be found at the small flow rate together with the fully developed flow instabilities. This led to the preliminary conclusion that the broadband pressure pulsations are exclusively induced by the flow instabilities, especially the vortices in the flow passages of impeller.	centrifugal pump, pressure pulsations, flow instabilities, vortices
363-377	A Study on the Deep-Surge Frequencies in Various Conditions of Axial Flow Compressors and Flow-paths	Nobuyuki Yamaguchi	Frequencies of deep surges and their behaviors in axial flow compressors were surveyed numerically. Relative surge frequencies, normalized by the basic acoustical resonance frequencies, are seen to tend to lower together with increases in the stalling pressure ratios, i.e. increases in the number of stages and the compressor tip speed, and also together with increases in the flow-path sectional area ratios. However, it appears difficult to express simply the general behaviors of the relative frequencies affected by the various factors. In order to know the essential behaviors, a modified reduced surge frequency is proposed, which is a dimensionless number comparing the mass flow filling and emptying the plenum volume in surge and the mass flow provided by the compressor. The modified reduced surge frequencies are found to have or approach a definite and nearly constant value in conditions of deep surges. The parameter suggests the fundamental mechanism of deep surges and could be used to determine approximate frequencies of deep-surges in various conditions of compressors and flow-paths.	Fluid Machine, Axial Flow Compressor, Surge, Analytical Simulation, Frequency, Fluid Dynamics
378-384	CFD-DEM Simulation for Distribution and Motion Feature of Crystal Particles in Centrifugal Pump	Dong Liu, Cheng Tang, Shicheng Ding, Binhao Fu	In consideration of the particle features and behaviors, the Computational Fluid Dynamics (CFD)-Discrete Element Method (DEM) coupled method has been applied to simulate the liquid-solid flows in the centrifugal pump with crystallization phenomenon. The crystal particles tend to distribute more uniformly in the inlet section and enter the impeller along the pressure sides of the blades with a moderate rise in velocity. Particle number density is different at different regions in the impeller passages with the characteristics of small density near suction sides and large density near pressure sides. In addition, large crystal particles are mainly located near the pressure sides and small crystal particles predominantly appear in the region near suction sides. The relative velocity magnitude of flow near the impeller inlet tends to be higher than that of crystal particles, while the velocities of the solid particles are	Centrifugal pump, crystallization phenomenon, CFD-DEM, particle feature and behavior
385-393	Internal Flow and Performance with Foreign Vegetable Materials in a Contra-Rotating Small Hydro-Turbine	Ding Nan, Toru Shigemitsu, Shengdun Zhao, Yasutoshi Takeshima	Small hydropower generation is one of important alternative energy, and potential of small hydropower is great. Efficiency of small hydro-turbine is lower than that of large one, and these small hydro-turbine's common problems are out of operation by foreign materials. Therefore, we adopted contra-rotating rotors, which can be expected to achieve high performance and enable to use low-solidity rotors to achieve stable operation. Experimental apparatus of the contra-rotating small hydro-turbine with 60mm casing diameter was manufactured and some experiments were conducted. In this research, the internal flow with foreign vegetable materials, i.e. leaves and grasses, were investigated by a high speed camera, and the performance experiments when two pieces of cudweeds were attached on the blade of front rotor were conducted. As the results shown, the passing rate of small leaves is 41.2% and all of the middle leaves can't pass through the contra-rotating small hydro-turbine. The highest efficiency also decreased about 13%, reaching to 51.2% at 1.4Qd, because of the foreign vegetable materials.	Small hydro-turbine, Contra-rotating rotors, Internal flow, Performance, Flow visualization
394-403	Comparative Performance Analysis of an Electric Motor Cooling Fan with Various Inlet Vent and Blade Shapes	Jae-Min Park, Kwang-Yong Kim, Man-Woong Heo	Electric motors are used as the main power sources in many industrial equipments and household appliances. The miniaturization and weight reduction of electric motors generally increase the internal heat generation. Therefore, it is important to understand the flow characteristics of motor cooling fans and to improve their performance. The present study aimed at systematically investigating the effects of the inlet vent and blade shapes on the aerodynamic performance of a low-voltage electric motor cooling fan. The flow characteristics of the low-voltage electric motor cooling fan was numerically analyzed using three-dimensional Reynolds-averaged Navier-Stokes equations. The k-ε turbulence model was selected for the analysis of turbulence using a turbulence model test. An optimal grid system in the computational domain was selected through a grid dependency test. The mass flow coefficient and torque coefficient were considered as the performance parameters of the cooling fan. Eleven inlet vent shapes and eleven blade shapes of the cooling fan were tested by evaluating the aerodynamic performance of the cooling fan. The mass flow coefficient and torque coefficient were considered as the performance parameters of the motor cooling fan. Eleven inlet vent shapes on the fan cover and eleven blade shapes were tested to evaluate their effects on the mass flow coefficient and torque coefficient. The maximum mass flow coefficient of 0.0908 and the minimum torque coefficient of 0.0089, were achieved using different combinations of vent and blade shapes.	Motor cooling fan, Reynolds-averaged Navier-Stokes equations, Inlet vent, Bi-directional blade, Mass flow coefficient, Torque coefficient

404-411	Dynamic evolutions between the draft tube pressure pulsations and vortex ropes of a Francis turbine during runaway	Xiaoxi Zhang, Qijhua Chen, Jie Liao	To analyze the dynamic evolutions between the draft tube pressure pulsations and vortex ropes of a Francis turbine, the runaway transient process of a hydropower system is simulated by coupling a one-dimensional model of the water conveyance system and a three-dimensional model of the Francis turbine. The results show that the annular-distributed pressure pattern at the entrance of the draft tube breaks and induces small vortex ropes, which then merge into an eccentric-distributed helical one with the transient operating point moving away from the rating region. In this process, low frequency pressure pulsations form and continue to strengthen. When the operating point moves to the runaway point, the vortex ropes keep dividing and merging irregularly, causing random-like pressure pulsations.	Francis turbine, runaway, 1D-3D coupling, numerical simulation, pressure pulsations, flow patterns
412-420	Effect of Solid Particles on Cavitation and Lubrication Characteristics of Upstream Pumping Mechanical Seal Liquid Membrane	Huilong Chen, Dongdong Sun, Yuanzheng Wu, Miaomiao Chen, Peilin Zhang	In order to investigate the effect of solid particles on the cavitation characteristics and lubricating properties of the micro-gap liquid film in Upstream pumping mechanical seals, the Eulerian multiphase flow model was used to simulate the liquid film with different diameter and volume fraction of solid particles to analyze the influence of the particles on the distribution of vacuole, opening force and friction torque of the film under different working conditions. The results showed that the particles have an inhibitory effect on the cavitation, and the cavitation area and the volume fraction of the bubbles were both decreased. The cavitation area increased with the increase of particle diameter, which indicated that the inhibition of cavitation was weakened with the increase of particle diameter. The cavitation area decreased with the increase of the particle volume fraction, and the volume fraction increased the cavitation inhibition effect. The presence of particles improved the opening force of liquid film to a certain extent and increased with the increase of particle volume fraction, but the effect of particle diameter on opening force was different under different rotating speed and different medium pressure. The friction torque did not change obviously with the particle diameter, and decreased only slightly with the increase of the particle volume fraction. In the working condition range, the cavitation degree is not related to the pressure of the medium, but increases with the increase of the rotational speed, and the cavitation area and volume fraction of bubbles were significantly decreased when there were solid particles.	Upstream pumping mechanical seal, Particles, Cavitation, Lubrication, Opening force, Friction torque
421-431	Unstructured Grid Smoothing for Turbomachinery Applications	Mehdi Falsafioon, Sina Arabi, Ricardo Camarero, Francois Guibault	In the present study, two mesh smoothing techniques, Laplace and Winslow smoothing techniques, for unstructured grids on turbomachinery application are investigated. These operators are based on the solution of elliptic equations. In the first case, Laplace's equations are solved using a barycentric averaging procedure. Solution of Winslow's equations has been a challenging work for unstructured grids because of existence of cross derivative terms in the equations. This issue is addressed devising a local control volume. Both methods are compared using different grid quality criteria. Finally, these operators have been applied to turbomachinery configurations and the advantages and disadvantages are discussed.	Grid Smoothing, Unstructured Grid, Winslow's Equation, Grid quality
432-438	Unsteady Flow Condition of Centrifugal Pump for Low Viscous Fluid Food	Toru Shigemitsu, Junichiro Fukutomi, Takumi Matsubara, Masahiro Sakaguchi	Fluid machineries for fluid food have been used in wide variety of field i.e. transportation, filling, and improvement of quality of fluid food. The flow condition of these fluid machineries is quite complicated because the fluid food is different from water. Therefore, a design method based on the internal flow condition is not conducted. In this research, a turbo-pump having small number of blades was used to decrease shear loss and keep wide flow passage. In previous studies, it was found that internal flow condition was complex in the test pump, but those flow phenomena were not clarified in detail. In order to investigate the complex internal flow condition, the unsteady numerical analysis using low viscous fluid was conducted. In this paper, the relation between the blade geometry and the performance was investigated. In addition to that, the internal flow condition of the centrifugal pump using low viscous fluid was clarified by the numerical analysis results.	Centrifugal pump, Numerical analysis, Unsteady flow, Low viscous fluid
439-446	Effects of Surface Roughness on Thermal and Hydrodynamic Behaviors in Microchannel Using Lattice Boltzmann Method	M. A. Taher, M. K. Dey, Y. W. Lee	The aim of the present study is to use the artificial roughness geometries in order to investigate the thermal and hydrodynamic behaviors in a rough microchannel using the most effective alternative method in micro-flows namely Lattice-Boltzmann Method (LBM). The rough surfaces are configured with triangular, trapezoidal and rectangular roughness elements with a relative roughness height 0-8% of the channel height. To analyze the roughness effects, the friction coefficients in terms of Poiseuille number (Pn), the rate of heat transfer in terms of Nusselt number (Nu) and the mass flow rate have been discussed at the slip flow regime, $0.01 \leq Kn \leq 0.10$ , where Kn is the Knudsen number. Finally, the overall performance has been studied numerically. All of these numerical results for all kind of roughness geometries are compared with smooth microchannel.	Lattice-Boltzmann, Friction factor, Nusselt number, Surface roughness, Microchannel
447-456	Design and Performance Analysis of Axial Hydro Turbine with Criteria of Tangential Velocity and Constant Diffusion Factor	Priyono Sutikno, Nono Suprayetno, Nathanael P. Tandian, Firman Hartono	In reality, the flow condition inside meridional channel may vary from hub to tip, therefore meridional analysis will affect to the blade geometry whether is twisted or tapered. The prediction of maximum surface velocity is not easy, but Lieblein was postulated the alternative form of diffusion factor (DF) on airfoil which can be applied for the cascade analysis. Diffusion factor on cascade is an important design parameter to avoid cavitation in turbomachinery due to the pressure distribution and velocity variation on the surface. The purpose of present study is to know the influence of tangential velocity distribution and constant DF on the blade shape and performance of the axial hydro turbine. There are four criteria of tangential velocity distribution or usually is called as swirl velocity that discussed in this paper, they are mixed vortex, free vortex, forced vortex and constant vortex. Constant DF was applied to the all tangential velocity distribution to obtain pitch chord ratio calculation. This will affect to the pitch chord ratio value and the blade shape of rotor. Performance prediction of axial turbine is performed through Computational Fluid Dynamic (CFD) to validate global parameter design. The result of numerical simulation can be used as basic consideration for manufacturing and experimental testing.	tangential velocity, cascade, diffusion factor, meridional analysis, pitch chord ratio, axial hydro turbine
457-464	Numerical Simulation of Vortex Induced Vibration of A Flexible Cylinder	Mojtaba Hosseini, Hamid reza Ashrafi, Peyman Beiranvand, Ahmad Dehestani, Kioumars Dehestani	Numerical simulations of vortex-induced vibration of a three-dimensional flexible Cylinder under uniform turbulent flow are calculated when Reynolds number is $1.35 \times 10^4$ . In order to achieve the vortex-induced vibration, the three-dimensional unsteady, viscous, incompressible Navier-Stokes equation and LES turbulence model are solved with the finite volume approach, the Cylinder is discretized according to the finite element theory, and its dynamic equilibrium equations are solved by the Newmark method. The fluid-Cylinder interaction is realized by utilizing the diffusion-based smooth dynamic mesh method. Considering VIV system, the variety trends of lift coefficient, drag coefficient, displacement, vortex shedding frequency, phase difference angle of Cylinder are analyzed under different frequency ratios. The nonlinear phenomena of locked-in, phase-switch are captured successfully. Meanwhile, the limit cycle and bifurcation of lift coefficient and displacement are analyzed using trajectory, phase portrait and Poincare sections. The results reveal that: when drag coefficient reaches its minimum value, the transverse amplitude reaches its maximum and the "lock-in" begins simultaneously. In the range of "lock-in", amplitude decreases gradually with increasing of frequency ratio. When lift coefficient reaches its minimum value, the phase difference, between lift coefficient and lateral displacement, undergoes a suddenly change from the "out-of-phase" to the "in-phase" mode. There is no bifurcation of lift coefficient and lateral displacement occurred in three dimensional flexible Cylinder submitted to uniform turbulent flow.	Vortex-induced, dimensional, finite element, lift coefficient