Computational Investigation of Turbulent Separating Flows in a Gas Turbine Combustor

A. C. Benin, M. P. Eckardt, P. J. Stopford, E. Buchanen, K. J. Syrd

In the first part of the paper, Computational Fluid Dynamics analysis of the combusting 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 lean-pretired 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 water test rigs. It has been observed that turbulent swirling flows downstream 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 DES procedures as well as LES and DES approaches.

Flows over Concave Surfaces: Development of Pre-set Wavelength Görtler Vortices

S. H. Wonko, Tadonko, O. A. Shah, H. Mitsubishi

The development of pre-set wavelength Görtler vortices are studied in the boundary-layer flows on concave surfaces of 1.0 and 2.0 in radius of curvature. The wavelengths of the vortices were pre-set by a wire of 0.5 mm diameter placed 30 mm upstream and perpendicular to the concave surface leading edge. Velocity contours were obtained from velocity measurements using a single hot-wire anemometer. The most amplified or dominant wavelength is found to be 15 mm for free-stream velocity of 2.1 m/s and 1.0 m/s on the concave surface of R = 2 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 on the basis of 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.

Control of Shock/Wave/Bound-Layer Interactions by Inlet Bleed

T.L.P. Sesh

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.

The Development of Protocols for Equitable Testing and Evaluation in Ocean Energy - A Three-Year Strategy

David M. Oprean, Jose Luis Villate, Cyllie Abumen, Cameron Johnstone

Equitable (Shareable) 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 EquMar project aims to define a suite of protocols for the evaluation of both wave and tidal converters, harmonising 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 EquMar protocols will cover site selection, initial design, scaling up of designs, the deployment of arrays and environmental impact assessment as well as economic issues. Equitable will build on existing protocols, e.g., U.K. DTI Marine Renewables Development Fund (MRDF) protocols for wave and tidal energy, and engage with international standards setting activities, e.g., IEC, TCIU.

Cavitation Instabilities of Hydrofoils and Cascades

Toshinobu Tsujimoto, Satoshi Watanabe, Hironori Horiguchi

Studies on cavitation instabilities of hydrofoils and cascades are reviewed to fundamental understandings of the instabilities observed in the pump-inducer. 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 mainly on the mean cavity length. For a hydrofoil, the characteristic length is the chord length and partial/transitional cavity cavitation occurs with shorter/faster cavity than 15% of the chord length. For cascades, the characteristic length in 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.

Study on Flow Fields in Variable Area Nozzles for Radial Turbines

Hideaki Tamaki, Masaru Urno

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 vane is dominant.

Study on the Cavitation Correlation-test: Control of Surge and Vortex Core Transition in a High Power Gas Turbine

Makpuri Miyani, Toshiro Kanemoto, Daisuke Kawashima, Akhiro Wada, Takashi Hara, Kanayuki Sakata

To optimize the stationary components in the multistage centrifugal pump, the effects of the return vane profile on the performance of the multistage centrifugal pump were investigated experimentally, taking account of the inlet flow conditions for the next stage impeller. The return vane, whose leading edge is at the outer wall position of the annular channel downstream of the vanes and which discharges the swirl-les flow, gives better pump performances. By equipping such return vane with the swirl step set from the trailing edge to the main shaft position, the unstable head characteristics can be 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.

Study on Flow Fields in Variable Area Nozzles for Radial Turbines

Toshinobu Watanabe, Dohyoung Kang, Angelo Cenone, Yuko Kawai, Toshinobu Tsujimoto

During an experimental investigation on a 3-bladed and a 4-bladed 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 vane 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 period 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.
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, 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, it is newly proposed a 4-blade helical type rotor which can reduce the pressure pulsation drastically and the performance prediction of new PDT is determined.

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 cranking. In order to resolve such problems, preventive coating and protective measures against internal and external fatigue and corrosive environments are introduced. 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.

An experiment setup was introduced to study dynamic behavior 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 trailing distance/angle gave better performance under pressure transient condition than check valves without these features. Air entrainment was found to shift 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.

The Wells turbine rotor consists of several symmetric aerofoil 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.

The present study is to reveal a variable geometry (VG) mixed flow turbine with a novel, purposely designed plugging nozzle vane ring. The nozzle vane ring was matched to the 3-dimensional aspect of the mixed flow rotor leading edge with lean steering. It was found that for a nozzle vane ring in a volute, the wall 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 experimentally studied in steady and unsteady flow conditions. The VG mixed flow turbine shows higher peak efficiency and swallowing capacity at various nozzle 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 be 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 is higher possibility of choking during a pulse cycle. 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 rate, 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.

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 rate, 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.

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. Kurrus 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.

An experiment 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 trailing distance/angle gave better performance under pressure transient condition than check valves without these features. Air entrainment was found to shift 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.

The Wells turbine rotor consists of several symmetric aerofoil 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.

The present study is to reveal a variable geometry (VG) mixed flow turbine with a novel, purposely designed plugging nozzle vane ring. The nozzle vane ring was matched to the 3-dimensional aspect of the mixed flow rotor leading edge with lean steering. It was found that for a nozzle vane ring in a volute, the wall 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 experimentally studied in steady and unsteady flow conditions. The VG mixed flow turbine shows higher peak efficiency and swallowing capacity at various nozzle 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 be 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 is higher possibility of choking during a pulse cycle. 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 rate, 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.

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. Kurrus 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.

Steam Turbine, Corrosion Fatigue, Wilson Zone, Thermodynamics, Numerical Analysis, ODE, Surface Treatment, Hydraulic Design, Deposits, Corrosion, Erosion, On-line wash

The Wells turbine rotor consists of several symmetric aerofoil 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.

The present study is to reveal a variable geometry (VG) mixed flow turbine with a novel, purposely designed plugging nozzle vane ring. The nozzle vane ring was matched to the 3-dimensional aspect of the mixed flow rotor leading edge with lean steering. It was found that for a nozzle vane ring in a volute, the wall 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 experimentally studied in steady and unsteady flow conditions. The VG mixed flow turbine shows higher peak efficiency and swallowing capacity at various nozzle 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 be 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 is high possibility of choking during a pulse cycle. The present study is to reveal a variable geometry (VG) mixed flow turbine with a novel, purposely designed plugging nozzle vane ring. The nozzle vane ring was matched to the 3-dimensional aspect of the mixed flow rotor leading edge with lean steering. It was found that for a nozzle vane ring in a volute, the wall 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 experimentally studied in steady and unsteady flow conditions. The VG mixed flow turbine shows higher peak efficiency and swallowing capacity at various nozzle 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 be 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 is high possibility of choking during a pulse cycle. The present study is to reveal a variable geometry (VG) mixed flow turbine with a novel, purposely designed plugging nozzle vane ring. The nozzle vane ring was matched to the 3-dimensional aspect of the mixed flow rotor leading edge with lean steering. It was found that for a nozzle vane ring in a volute, the wall 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 experimentally studied in steady and unsteady flow conditions. The VG mixed flow turbine shows higher peak efficiency and swallowing capacity at various nozzle 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 be 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 is high possibility of choking during a pulse cycle.
Analysis of Aerodynamic Performance in an Annular Compressor Bowed Cascade with Large Camber Angle

Shooshen Chen, Yu Chen

The effect 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 surface, 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 larger 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, 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

Leakage Flow, Geometry

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 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 correspond approximately with the results of slurry wear tests. This technique is useful for industrial application.

Pump, Impeller, Numerical analysis, Slurry wear, Life prediction

A Behavior of the Diffuser Rotating Stall in a Low Specific Speed Mixed-Flow Pump


The flow instability in a low specific speed mixed-flow pump, having a positive slope of head-characteristics was investigated. Based on the static pressure measurements, it was found that a rotating stall in the varied diffuser occurs at about 65% flow rate of best efficiency point (BEP). A Dynamic Particle Image Velocimetry (DPIV) measurement and the numerical simulations were conducted to investigate the flow fields. As a result, the diffuser rotating stall was simulated even by Computational Fluid Dynamics (CFD) and the phenomena were confirmed. It is clarified that a periodical large scaled backflow is generated at the leading edge of the suction surface of the diffuser vane, cause the instability. Furthermore, the growth of the strong vortices at the leading edge of the diffuser vane induce the strong backflow from the diffuser outlet to the inlet. The scale of one stall cell cover over four passages in total thirteen vane-passage.

Mixed flow pump, pump instability, Varied diffusion, Rotating stall, CFD

A Backflow Vortex: Cavitation and Its Effects on Cavitation Instabilities

Kazuyoshi Yamamoto, Yoshikazu Tasumoto

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. The importance of the phase delay of backflow vortex cavitation is clarified. Fundamental characteristics of backflow vortex structure is shown by followed details: on the energy transfer under cavitation surge in the centrifugal pump. Then, the dynamics of backflow vortex is discussed to explain a large phase lag observed in the experiments with the centrifugal pump.

Cavitation, Instability, Backflow, Cavitation Surge, Induced, Centrifugal pump, Vortex

A Thermal Analysis of a Film Cooling System with Normal Injection Holes Using Experimental Data

Eung Min Kim, Dong Hyun Lee, Hyung Rae 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 corrected from the Stanton number 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 10^2 °C in the y-axis (spanwise) direction and about 10^3 °C 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

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 developed extensively. This paper 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

Compressor Blade, Optimization, Surrogate Modeling, Stacking Line, Thickness of Blade, Efficiency

Effect of Internal Flow in Symmetric and Asymmetric Micro Regenerative Pump Impellers on Their Pressure Performance

Hironori Horiguchi, Shinya Matsumoto, Yoshinobu Tasumoto, Masaaki Sakagami, Shigey Tanaka

The effect of symmetric and asymmetric micro regenerative pump impellers on their pressure performance was studied. The effect of 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 unsteady velocity in the casing was attributed to the smaller local flow rate in the casing.

Regenerative Pump, Pump performance, Internal flow, Leakage flow, Geometry
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 position 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 tip factor and pressure loss caused by the vortical 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 deviation in the relative velocity contributed to the improvement in the overall performance.

The purpose of this paper is to gain a better understanding of the performance of practical wind turbine generating systems with different 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.

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 zone on the suction surface and on the front shroud. Hence several experimental attempts were earlier made to assess the efficiency of using boundary layer fences to trip the flow in the regions of separation and to make the flow align 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 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 fan 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 improve the static pressure recovery across the fan, fences provided at the radial mid-span on the pressure side of the diffuser and near the leading edge and trailing edge of the suction side of diffuser vanes also improve the static pressure recovery across the fan.

A conventional centrifugal pump causes a drastic deterioration of air-water two-phase flow performance even at an air-water-two-phase flow condition of inlet void fraction less than 10% in the range of high void 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 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 an air-water two-phase flow performance indicate a little difference from the similarity relations.

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 computers) was selected for further examination. The influence on aerodynamic performance and noise of such frame design parameters on blade tip clearance results in a decrease of discrete frequency noise and an increase of broad-spectrum noise. As most suitable design refinement in terms of fan efficiency, we found that the treatment of outlet corner roundness and altering spoke slots to the direction counter to that of fan rotation was effective.

Dynamic characteristics of the clearance flow between an axisymmetric rotating 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 contraction flows had an effect to stabilize the 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.

The 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 identified. Wave energy conversion in Malaysia is reviewed.
189-196

Study for the Increase of Micro Regenerative Pump Head
Hiroshi Horikuchi, Keisuke Wakiya, Yoshinobu Tsuji moto, Masaki Sekigami, Shigen 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.

Keywords: Micro Regenerative Pump, Internal Flow, Blade Angle, Clearance, Leakage Flow

197-205

Fluid-Oscillation Coupled Analysis for HAWT Rotor Blade (Degree of Freedom Weak Coupling Analysis with Hinge-Spring Model)
Kohei Imamura, Yutaka Hasegawa, Jason Muzari, Sho Chihara, Daisuke Takezaki, Naoto 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 vibration. In this paper, fluid-structure coupled analysis for the HAWT rotor blade was 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 data of the NOE uncommon aerodynamic experiment. In the analysis the unsteady flapwise moments in yawed inflow conditions are compared for the blade with different flapwise eigen frequencies.

Keywords: Wind Turbine, Free Wake Panel Method, Weak Coupling Analysis, Hinge-Spring Model, Yawed Inflow

206-214

Cause of Cavitation Instabilities in Three Dimensional Inducer
Shinya Kage, Keiichiro Yonezawa, Hiroshi Horikuchi, Yutaka Kawai, Yoshinobu Tsujimoto

Alternate blade cavitation, rotating cavitation and cavitation surge in intake turbopump inducers are simulated by a three dimensional computational 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 decreases 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.

Keywords: Cavitation instabilities, Velocity disturbance, Three dimensional CFD

215-222

Flow Field Change before Onset of Flow Separation
Hiroaki Haiyage, 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 (VGJ) 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 desired. 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 at 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.

Keywords: Separation, Boundary Layer, Unsteady Flow, Diffusion, Precursor
Experimental Investigation on Separated Flows of Axial Flow Stator and Diagonal Flow Rotor

Yoshihiko Kintaka, Norimasa Shirai, Toshihiko Setoguchi, Yuzo 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 flow rate of the suction flow and the corotating flow are calculated from the suction flow and the hub flow by the influence of corner wall. For the flow rate of 80-90% of the design flow rate, the corner separation is observed, which becomes widely spread for 85% of the design flow rate. At rotor outlet for 85% of the design flow rate, the tip axial velocity region grows below the suction surface of the corner separation because of the tip leakage flow of the rotor.

Rotating Choke and Choked Surge in an Axial Pump Impeller

Toshiyuki Watanabe, Hideyoshi Safo, Yasaiko Mori, Hitoshi Horischi, Yutaka Kawa, Yoshihiko Tsujimoto

Unlike usual turbo-pumping inducers, the axial flow pump impeller tested operates very steadily at design flow rate without rotating cavitation nor cavitating 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 impeller blade. However, at low flow rate and low cavitation number, choked surge and rotating choke were observed. Their correlation with the performance curve under cavitating is discussed and their instantaneous flow fields are shown.

A Study of Performance and Internal Flow in a New Type of Sewage Pump

Yasuyuki Nishi, Junichiro Fukumori

Sewage pumps are designed with a side 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 supposed 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 design flow channel of a closed type single-stage pump is wound in a helical spiral. The focus of this research was to investigate pump performance and internal flow in this single side 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.

Numerical Prediction of Unsteady Flows through Whole Rotor-Cascade Channels with Partial Admission

Yasuhiko Seino, Kazuhiro Motoyama, Tadashi Tsukamoto, Satoshi Yamanitomo

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 MACUL TVD scheme, Roe's approximate Riemann solver and the LU-SGS scheme. The SST model is 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 parameterized predicted.

Vibration Analysis of a Rotor considering Nonlinear Surge Considering the Finite Sound Velocity in the Rotor Passages

Soo-Mok Lee, Do-Hyeong Lim, Jong-Gug Bae, Bo-Suk Chang, Kun Chen, Christophe Nicolet, Koichi Yonezawa, Yoichi Kinoue, Norimasa Shiomi, Toshiaki Setoguchi, Tadashi Tanuma, Toshifumi Watanabe, Hideyoshi Sato, Yasuhiko Tsujimoto

Hydraulic Turbine Components

The effects of acoustic modes in the penstock on the self-excited oscillation in the hydropower system were studied by assuming a finite sound velocity in the nozzle-rotor cascade channels considering a partial admission. Ten 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 describing the attenuation, dispersion and shape of observed transient response. 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.

One-Dimensional Analysis of Full Load Draft Tube Surge Considering the Finite Sound Velocity in the Penstock

Changku Chen, Christophe Nicolet, Kazuhiro Moriya, Mohamed Farhat, Francesca Avelar, Yoshihiko Tsujimoto

The effects of acoustic modes in the penstock on the self-excited oscillation in the hydropower system were studied by assuming a finite sound velocity in the nozzle-rotor cascade channels considering a partial admission. Ten 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 describing the attenuation, dispersion and shape of observed transient response.

Unsteady Flow with Cavitation in Viscous Polyethylene Pipes

Alexandre K. Soares, Oldia L.C. Covas, Helena M. Ramos, Luisa Fernanda R. Reis

The current paper focuses on the analysis of transient cavitation flow in pressured polyethylene pipes, which are characterized by viscoelastic rheological behavior. A hydraulic transient solver that describes fluid flows in plastic pipes has been developed. This solver incorporates the description of dynamic effects related to the energy dissipation (unsteady viscosity), the rheological mechanical behaviour of the polyethylene pipe and the cavitation flow pipe. The Discrete Vapour Cavity Model (DVCM) and the Discrete Gas Cavity Model (DGCM), have been used to describe transient cavitation flow. Such models assume that discrete cavities are formed in fixed sections of the pipeline and consider a constant wave speed in pipe sections between cavities. The cavity dimension and pressure are allowed to grow and collapse according to the mass conservation principle. An extensive experimental programme has been carried out in an experiment setup composed of high-density polyethylene (HDPE) pipes, assembled at Instituto Superior Técnico of Lisbon, Portugal. The experimental facility is composed of a single pipeline with a total length of 205 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 experiments without cavitation flow. Transient tests were carried out by the fast closure of the ball valves located at downstream end of the pipeline for the non-cavitation flow and at upstream for the cavitation flow. Once the rheological behaviour of HDPE pipes were known, computational simulations have been run in order to describe the hydraulic behavior of the system for the cavitation flow pipe. The calibrated transient solver is capable of accurately describing the attenuation, dispersion and shape of observed transient response.
Hydroelectric 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 behavior 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 eigemodes shapes of 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 model 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 equivalent schemes are presented and stabilized. Finally, a hydro-acoustic SIMSEN model of a simple hydraulic power plant, is used to apply the model 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 eigenvector modes.

The flow in the draft tube cone of Francis turbines operated at partial discharge is a complex hydrodynamic phenomenon where an incoming steady asymmetric 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 axis-symmetric 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 meridian 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.

The present paper shows the results of numerical and experimental model analyses of Francis runners, which were executed in air and in still water. In first part this paper is focused on the numerical prediction of the model parameters by means of FEM and the validation of the FEM method. Influences of different geometries on model parameters and frequency reduction ratio (FRR) ratio, 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.

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 overloads 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 dynamics (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.

This paper presents a numerical simulation study of the transient behavior of a 32x40m^2 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 2D-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.
Dynamic Simulation of Pump-Storage Power Plants with different variable speed configurations using the Simsen Tool

Pumped storage power plants are playing a significant role in the contribution to the stabilization of an electrical grid, especially due to 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 coordination 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 applications). Due to the rapid development of power semiconductors 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 into 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.

Large Eddy Simulation of a High Reynolds Number Swirling Flow in a Conical Diffuser

Cédric Duprat, Olivier Mettia, Thomas Leawen

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 recovers directly the most energetic part of the flow. The CFD approach is here preferred to the average equations strategies (RANS models) because it recovers the most energetic part of the flow. The CFD approach is here preferred to the average equations strategies (RANS models) because it recovers the most energetic part of the flow.

Surface Roughness Impact on Francis Turbine Performances and Prediction of Efficiency Step Up

Pierre Maruzewski, Vlad Haemtuch, Henri-Pascal Mombelli, Danny Burggraeve, Jacob Isfort, Pierre Fritsche, François Avellan

In the process of turbine modernization, the investigation of the influence of water passage roughness on several hydraulic performance measures 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 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. Moreover, this paper states that CFD is a very promising tool for future 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 consequence of the vortex are transferred upstream and downstream with amplitude and frequency modulation in respect to 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.

Numerical prediction of pressure pulsation amplitude for different operating regimes of Francis turbine draft tubes

Andrey Lopj, Dragunc Joel, Peter Mikor, Vesko Djeli

A New Concept of Hydraulic Design of Water Turbine Runners

Andrih Vasyly, Frantisek Pohchyli, Jiri Obilskiy, Jozef Mikułaski

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 observed. Similar problems with the efficiency loss 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 operation range with high efficiency level. The new concept to decrease the vibrations and pressure pulsations is based on a heterogenous 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.
The paper concerns the description of the step by step development process of the new fixed blade runner called “Micra” suitable for the uprating of the Francis turbine 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 prescrbed 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 concern efficiency, required head and cavitation features. After successful hydraulic design the model 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 in case of full Kaplan runner, 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.

Validation of a CFD model for hydraulic seals
Vincent Le Roy, François Guilbaud, 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 carefully simulated and stopped hydraulic seals used in Francis runners. The computation is performed with a finite volume commercial CFD code with a RANS based turbulence model. As numerical computations on 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.

Experimental Study on Surge Inception in a Centrifugal Compressor
Hidemasa Tamaki

An investigation of surge inception in a centrifugal compressor was done with measurements of steady and unsteady static pressure. Vanesless diffuser and vaneless 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 vaneless diffuser, instability that was generated in the impeller propagated into the vaneless diffuser, however the pressure recovery by the vane diffuser made the operation of the compressor stable at low flow rate.

Hydrodynamic Model Test on Prevention against Vortex Occurrence for Vertical Bull Turbine
Shoichi Yamato, Shogo Nakamura, Akimori Funakawa

A large bulb turbine unit with external type draft tube has been developed due to avoidance of complicated assembling and long standby period at repair in comparison with conventional horizontal bulb turbine unit. Before designing the prototype vertical bulb unit, a hydrodynamic model test was carried out to establish the ideal design concept for this innovative generating unit.

Design and Prototyping Micro Centrifugal Compressor for Ultra Micro Gas Turbine
Toshiyuki Hirano, Hoshio Tsuchiya, Rongguo Gu, Galu Minokawa

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 3-dimensional shape. The impeller was rotated at 250,000 rpm in the present study. The experimental results of the tested compressor with the vaneless 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 divergence angle at the high flow rate. These results were compared with those obtained by the prediction method used at the design stage.

Large Eddy Simulation of the Dynamic Response of an Inducer to Flow Rate Fluctuations
Donghyuk Kang, Koichi Yonezawa, Tatsuya Ueda, Nobuo Yamashita, Chisato 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-cavitating 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 simplified one dimensional evaluation of the inertia component, the component obtained by LES is smaller. The negative slope of averaged pressure curve becomes larger under unsteady conditions, from the considerations 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 shows 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 Design to Avoid Cavitation Instabilities
Donghyuk Kang, Toshifumi Watanabe, Koichi Yonezawa, Hironori Horiguchi, Yukata Kawai, 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 large 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 that 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 rate, 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 high 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.

A New Blade Profile for Bidirectional Flow Properly Applicable to a Two-stage Jet Fan
Michihito Nishi, Shuhong Liu, Kouichi Yoshida, Minoru Okamoto, Hiroaya Nakayama

A reversible axial 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 new shape 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.

Keywords:
- Fixed blade turbine
- CFD
- SST turbulence model
- hydraulic seal
- straight seal
- labyrinth seal
- experimental validation
- Centrifugal Compressor
- Surge, Pressure Fluctuation
- Vaned Diffuser
- Hydrostatic model test
- Surge wave
- Hydrodynamic model test
- Francis turbines units
- vertical bulb turbine
- elbow type draft tube
- small radial clearances
- labatory draft tube
- Francis runners
- steady static pressure
- vaneless diffuser
- vane diffuser
- low flow rate
- unsteady static pressure
- cavitation instabilities
- flow rate oscillations
- flow rate fluctuations
- non-cavitating conditions
- inertia component
- unsteady pressure rise
- Euler's head
- negative slope
- angular momentum
- energy
- LES
- RANS
- backflow performance
- vaneless diffuser
- vane diffuser
- outlet blade angle
- cavitation instabilities
- flow rate stability
- suction surface
- tip cavity
- backflow vortex cavity
- inertia effects
- response delay
- design flow rate
- design flow coefficient
- suction performance
- CFD computation
- cavity growth
- pressure distribution
- suction side
<table>
<thead>
<tr>
<th>Page</th>
<th>Title</th>
<th>Authors</th>
<th>Summary</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-10</td>
<td>Measurements of Minute Unsteady Pressure on Three-Dimensional Fan with Arbitrary Axis Direction</td>
<td>E. Srinivas Reddy, G. V. Ramana Murty, A. Dasgupta, K. V. Sharma</td>
<td>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 bell-hose around the 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.</td>
</tr>
<tr>
<td>11-18</td>
<td>Flow Investigations in the Crossover System of a Centrifugal Compressor Stage</td>
<td>G. V. Ramana Murty, A. Dasgupta, K. V. Sharma</td>
<td>The performance of the crossover system of a centrifugal compressor (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. A cavity with an angle of 12° inclination was set on the test system. Experimental results were compared with two-dimensional steady state vane vortices were used to simulate the flow at the exit of an actual centrifugal compressor impeller with a design flow coefficient of 0.55. The vane vortices 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.</td>
</tr>
<tr>
<td>20-29</td>
<td>Design and Analysis of a Controlled Diffusion Aerofoil (CDA) Section for an Axial Compressor Stator and Effect of Incidence Angle and Mach No. on Performance of CDA</td>
<td>K. Srinivasa Reddy, G. V. Ramana Murty, A. Dasgupta, K. V. Sharma</td>
<td>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 (NASGRO/3D) available in the SCIFIS library of NAL. The effect of variation of Mach No. was performed using Fluent. The surface Mach No. distribution on the CD profile slightly indicates lower peak Mach no. than CDA blade profile. Further, boundary layer parameters on CD aerofoil at respective incidences have lower values than corresponding CDA blade profile. Total pressure loss on CD aerofoil for the same incidence range is lower than CDA blade profile.</td>
</tr>
<tr>
<td>29-38</td>
<td>Surrogate Modeling for Optimization of a Centrifugal Compressor Impeller</td>
<td>Jin-Hyk Kim, Jae-Ho Choi, Kyoung-Yong Kim</td>
<td>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 design of experiments is used to generate the thirty design points within design space. Three-dimensional Navier-Stokes governing 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 optimization shows that the total-to-total pressure ratio of the optimal 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.</td>
</tr>
<tr>
<td>35-44</td>
<td>Performance Improvement of High Speed Jet Fan</td>
<td>Young-Seok Cho, Joon-Hyang Kim, Kyung-Yong Lee, Sang-Ho Yang</td>
<td>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-to-tip ratio, meridional shape, rotor stagger angle and 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 return flow regime are also discussed.</td>
</tr>
<tr>
<td>50-57</td>
<td>Cavitation Surge Suppression of Pump Inducer with Axi-asymmetrical inlet Plate</td>
<td>Jun-Ho Kim, Koichi Inohara, Satoshi Watanabe, Akimori Furukawa</td>
<td>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 occurrence of cavitation surge oscillation. Several kinds of axisymmetric plates with different slantlines 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 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 reasonable application will be proposed to suppress the cavitation surge oscillation by installing axisymmetrical inlet plate.</td>
</tr>
<tr>
<td>58-66</td>
<td>A Two-Dimensional Study of Transonic Flow Characteristics in Steam Control Valve for Power Plant</td>
<td>Koichi Yonekawa, Yoshinori Tesaki, Toru Nakajima, Yoshinobu Takeyama, Kenji Tsukada, Michiharu Moro, Ryu Mineta, Fumio Ishida</td>
<td>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 condition. In such conditions, the valve opening is small and the pressure difference 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.</td>
</tr>
</tbody>
</table>
Rotordynamic Instabilities Caused by the Fluid Force Moments on the Backshroud of a Francis Turbine Runner
Bongye Song, Hisashi Horiguchi, Zhenye 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 shroud. 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 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 on 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 below. A lumped parameter model of a cantilevered rotor was used for the vibration analysis. By examining the fluid force, the amplitude ratio of lateral and angular displacements for the cases with longer and shorter overhung rotors 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 leakage flow can cause self-excitation of an overhung rotor.

Cavitation Surge in a Pump with Busbar to the Impeller
Donghyuk Kang, Yusuke Arimoto, Koichi Yonezawa, Jun-Ho Kim, Takashi Atono, Koichi Ishizaka, Satoshi Miyagawa, Yoshinobu Tsujimoto
The cavitating flow with swirling flow in hydraulic power generating systems was studied by a simple experiment and numerical simulation. Several tests of cavitation and cavitation surge were observed in the experiments, including the cavitation surge caused by the discharge flow and the vortex precession by the swirl in the busbar. Both cavitation surge and vortex precession were simulated by CFD. Detailed flow structure was examined through flow visualization and CFD. It was confirmed that the vortex precession can cause cavitation surge.

Cavitation Suppression in an Impeller of a Francis Turbine Runner
Changkun Chen, Christophe Nicolas, Kōichi Yonezawa, Mohamed Farhat, Francesc Avellán, Katsuyoshi Miyagawa, Yoshinobu Tsujimoto
The suppression of cavitation instabilities in an inducer is observed with distributing multi-cameras circumferentially, recording displacement. Accordingly, two types of fluid force moment are exerted on the suction side of blade surface, the pressure became high on the suction 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 pressure side of blade surface, the pressure became high on the suction side of blade surface. 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-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 behavior in the inducer was observed with distributing multi-cameras circumferentially, recording simultaneously and reconstructing multi-photos on one plane face as moving a linear cascade. Observed results are utilized for discussion with other measuring results as causing wall pressure distribution. Then the suppression mechanism of oscillation by installing a suction-asymmetrical plate will be clarified in more details.

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, the shaft seal in centrifugal pumps, the magnetically suspended centrifugal pumps has been developed; this is a sealless rotors which can provide contact-free旋转 of the impeller without leading to leakage. 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, no stagnant fluid or dead flow regions were observed in the pump. Therefore, the present pumps can efficiently enhance the washout effect.

Keywords: Artificial heart, Blood Pump, Computational Fluid Dynamics, Internal Flow, Turbomachinery
Unsteady Swirling Flows Arising in Straight Tubes

A study on the performance of a centrifugal slurry pump

Unsteady Swirling Flows Arising in Straight Tubes

Lifetime Prediction of Film-Cooling Systems with and without Thermal Barrier Coating

Performance Prediction of Vertical Submersible Centrifugal Slurry Pump

Prediction of Axial Thrust for Mixed-Flow Pumps with Varied Diffuser by Using CFD

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 analyses at the designs that are selected through the Latin hypercube sampling method. Using these numerical simulation results, the Response Surface Approximation model is 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.

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.

Unsteady analysis of a 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 instability 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 60% 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 block 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 this disturbance caused by cavitation, it was found that there exists a flow component towards the trailing edge of tips 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 unsteady cavitating flows.

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 RVM commercial code were conducted to calculate distributions of temperature and thermal stress. In the simulations, the 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.

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 showed rotational speed which is approximately equal to or higher than the peripheral velocity of the existing 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.

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.

Micro- and nanoindentation test were performed to investigate the mechanical properties of straight groove micromixer on mixing performance and pressure drop. Three-dimensional Navier-Stokes equations with two trickling 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.
<table>
<thead>
<tr>
<th>Page</th>
<th>Title</th>
<th>Author</th>
<th>Abstract</th>
<th>Keywords</th>
</tr>
</thead>
<tbody>
<tr>
<td>225-244</td>
<td>Moment Whirl due to Leakage Flow in the Back Shroud Clearance of a Rotor</td>
<td>Yoshitoku Tsuchiya, Zhouyu Ma, Bingjian Song, Hiroshi Horiuchi</td>
<td>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 a 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.</td>
<td>Rotordynamic Instability, Fluid Force Moment, Whirling Motion, Precession Motion, Leakage Flow</td>
</tr>
<tr>
<td>245-252</td>
<td>Air-Water Two-Phase Flow Performances of Centrifugal Pump with Movable Bladed Impeller and Effects of Installing Diffuser Vanes</td>
<td>Shoji Sato, Atsuo Funakawa</td>
<td>It is known that pump head of centrifugal impeller with large 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 centrifugal impeller with variable blade outlet angles were experimentally investigated in both flows.</td>
<td>Centrifugal pump, Variable blade angle, Air-Water two-phase flow performance, Diffuser vane</td>
</tr>
<tr>
<td>253-259</td>
<td>Investigation of Leakage Characteristics of Straight and Stepped Labyrinth Seals</td>
<td>Tong Seip Kim, Soo Young Kang</td>
<td>Leakage characteristics of labyrinth seals with different configurations (straight or 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 behavior with test results, but the overall agreement with test data was not as good as that of the CFD.</td>
<td>labyrinth seal, straight seal, stepped seal, leakage, flow function, clearance</td>
</tr>
<tr>
<td>260-270</td>
<td>Study of the Flow in Centrifugal Compressor</td>
<td>Cheng Yu, Ryoichi Samuel Amaro</td>
<td>Reducing the losses of the tip clearance flow is one of the keys to 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 low pressure 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 machines was further extended and used in the centrifugal compressor design.</td>
<td>centrifugal compressor, flow separation, static efficiency, turbomachinery, impeller, surge</td>
</tr>
<tr>
<td>271-278</td>
<td>Performance and Flow Condition of Contra-rotating Small-sized Axial Fan at Partial Flow Rate</td>
<td>Toru Shigematsu, Junchiro Fukumori, Yuki Oshita, Kazuhiro Iuchi</td>
<td>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 fan is proposed for the improvement of performance. In previous 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 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.</td>
<td>Small-sized axial fan, Contra-rotating rotors, Performance, Internal flow, Partial flow rate, Numerical analysis</td>
</tr>
<tr>
<td>279-284</td>
<td>Tip Clearance Losses - A Physical Based Scaling Method</td>
<td>Peter F. Peit, Sascha Kast {the}l</td>
<td>Tip clearance losses occur in every turbomachinery. To estimate the losses it is important to understand the mechanism of these secondary flows.</td>
<td>Tip Clearance Losses, , Scaling, Vortex, Premultiplier</td>
</tr>
<tr>
<td>285-291</td>
<td>Physical Model Investigation of a Compact Waste Water Pumping Station</td>
<td>Ekiyan Kri, D.-H. Heimehl, Bernd Kothe, Peer Springer</td>
<td>To provide required flow rates of cooling or circulating water properly, approach flow conditions of vertical pumps 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 structure 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.</td>
<td>physical model investigation, waste water, pumping station</td>
</tr>
<tr>
<td>292-300</td>
<td>A Numerical Study on Cavitation Suppression Using Local Cooling</td>
<td>Yian-yuan Zhang, Xian-jing Sun, Dian-gui Huang</td>
<td>This study shows to develop an effective strategy to inhibit cavitation inception on hydrofoil 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 the corresponding liquid vaporization 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.</td>
<td>CFD, cavitation inception, cavitation suppression, local cooling temperature, surface roughness</td>
</tr>
</tbody>
</table>
Quest for reliability of hydraulic runners is a concern for all mature electricity producers. The fatigue damage caused by dynamic 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 Atomsmeklet for predictions of fatigue injury in runners. The analysis of fracture mechanics is subsequently to be use as a design method. This is sustained by a research on fracture mechanics properties of runner materials and by recommendations on the strategies 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.

Thermal Effects on Cryogenic Cavitating Flows around an Axisymmetric Ogive

Jianping Yan, Jiri Koutnik, Ulrich Seidel, Björn Hübner

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 are come out. An L_p3orthogonal experimental design 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 unsteady by CFD. According to the test result and optimization design in 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 61.16% and the single head is more than 4.8 m. Compared with the standard efficiency of 55%, the design of the stainless steel stamping pump is successful. This work would be instructive to the design of Stainless steel stamping multistage pump designed by the impeller head maximum approach.

Compressible Simulation of Rotor-Stator Interaction in Pump-Turbines

Suguo Shi, Guoyu Wang

Cavitation in cryogenic fluids generates substantial thermal effects and strong variations in fluid properties, which in turn affect 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 is 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 frosty and the cavitation intensity 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.

Improved Sauer Transform for Pump-Turbine Characteristics

Peter K. Dörrler

Standard dimensionless parameters cannot simultaneously represent all operation modes of a pump-turbine. They either have singularities at n=0 and multiple values in the 'unstable' areas, or else get singular at n=0. P. Suter (1996) 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.

Two-Way Coupled Fluid Structure Interaction Simulation of a Propeller Turbine

Hannes Schmiedhuber, 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 deflection can result in a significant impact on the turbine performance. The present paper analyses 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 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 deflection on the runner discharge and performance will be discussed and shown for the configuration investigated no significant influence under normal structural conditions. This study also highlights that a two-way coupled fluid structure simulation of a real engineering configuration is still a challenging task for today's commercially available simulation tools.
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-lowering, 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 models, comparing the 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, staged 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-inviscid flow analysis. In the next stage, the hydrauic 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.
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 vane placed upstream and perpendicular to the concave surface leading edge, were used to pre-set the wavelengths. Vorticity contours were obtained from hot-wire anemometry measurements. The most amplified vortex wavelengths can be pre-set by the spanwise spacing of the thin wires and the free-stream velocity. The vorticity contours on the cross-sectional planes at several streamwise locations show the growth and destruction of the vortices. Three different vortex growth regions can be identified. The occurrence of a secondary instability mode is also shown as multi-mode-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 wavelengths than the most amplified one, the splitting or merging of the streamwise vortices will respectively occur.

Blading in Highly Loaded Compressor Cascade at Low Suppression Effects by Air Separator Devices in Axial Leakage-free Rotating Seal Systems with Magnetic Applications

Our previous experimental and numerical investigations of decelerated swirling flows in concave 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 turbine operated at discharge, the jet becomes effective when the angle of attack is larger than 10% from the turbine discharge, leading to larger 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 concave 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.

Radial-varied air separators show a strong stall suppression effect in an axial flow fan, 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 effect is supposed to be achieved by a combination of the following several effects: (1) suction of blade and casing boundary layer and elimination of eddies of stall, (2) separation and straightening of re-energized 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 fan 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 encircling 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”.

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 magnetofluidic sealing fluids is shortly reviewed in particular the main steps of the chemical synthesis of magnetic nanofluids and magnetic composite fluids with high hydration; retrieval of all new synthetic carrier fluids. Design concepts and some constructive details of the magnetofluidic seals are discussed in order to 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 responsible fluids.

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 multiple 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 cavitation flow with visualization to evaluate the numerical results.

Concave surface, boundary layer instability, wall shear stress, hot-wire anemometry

Concave Surface Boundary Layer Flows in the Presence of Streamwise Vortices, Sonny H. Winoto, Tandiono, Dilip A. Shah, Hatem Minutharmadi

Flow Feedback for Pressure Fluctuation Mitigation and Pressure Recovery Improvement in a Conical Diffuser with Savi, Constantin Tanasa, Alin Boicu, Rómeo Susan-Regia, Sebastian Almeida

A Suggested Mechanism of Significant Stall Suppression Effects by Air Separator Devices in Axial Flow Fans, Nobuyuki Yamaguchi

Leakage-free Rotating Seal Systems with Magnetic Nanofluids and Magnetic Composite Fluids Designed for Various Applications, Tonde Borbás, Dóra Bica, József Potencz, Ivánkó Borbás, Tibor Böröcz, Ladislav Velka

Vortex Cavitation from Baffle Plate and Pump Vibrations in a Double-Suction Volute Pump, Toshiyuki Sato, Takahide Nagahara, Kanshiro Tanaka, Masaki Fuchiwaki, Fumio Shimizu, Akira Inoue

Steady and unsteady flow computations in an elbow draft tube with experimental validation, Th C. Vu, Christophe Devaux, Ying Zhang, Bernd Nennenmahr, François Gobutaud

Numerical Study of Passive Control with Slotted Blading in Highly Loaded Compressor Cascade at Low Mach Number, Moubi Ramo, Gérard Bois, Gahmouni Aderbounne

A New Approach in Numerical Assessment of the Cavitation Behaviour of Centrifugal Pumps, Adrian Stapan, Rómeo Susan-Regia, Lucio Eugen Anton, Sebastian Muntean

A Suggested Mechanism of Significant Stall Suppression Effects by Air Separator Devices in Axial Flow Fans, Nobuyuki Yamaguchi

Vortex Cavitation, Cavitation Noise, Double-Suction Volute Pump, CFD, Pump Oscillation

Steady and unsteady flow computations in an elbow draft tube with experimental validation, Th C. Vu, Christophe Devaux, Ying Zhang, Bernd Nennenmahr, François Gobutaud

Hydraulic turbine, draft tube, steady flow simulation, unsteady flow simulation

Steady state computations are routinely used by design engineers to evaluate and compare losses in hydraulic components. In the case of the draft tube blade design, however, experimental investigations 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 capability 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, CFD and OpenFOAM, and through a systematic comparison with experimental performance obtained on the FRANC model draft tube.

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 bladelets in cascade configurations. The objective of this current 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 performance 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 loadings 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 concave 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.

The paper presents a new method for the analysis of the cavitation behavior of hydraulic turbines. This new method allows determining the cavitation behavior of the turbine using a large storage pump. By plotting in semi-logarithmic coordinates the vapour volume versus the cavitation coefficient, we show that all curves 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 sensitivity, pumping head drop, storage pump
The paper presents a methodology for the prediction of turbine cavitation using a 3D multi-phase CFD approach. The methodology is based on a mixed cavitation model and the k-ω SST turbulence model. The results of the simulations are compared with experiments, and the methodology is shown to be effective in predicting cavitation in turbine impellers. The paper also discusses the importance of cavitation in turbine design and operation.
The present work examines numerical results using OpenFOAM™ of the flow in the swirl flow generator test rig developed at Politecnica 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 k-ε model, and block structured wall function ICEM-CFD meshers are used. The numerical results are validated against experimental LDV data, and against design velocity profile data. 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 are 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 overestimated.

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 PIV method. For calculation, the three dimensional turbulent flow for the full flow passage of the pump is modeled 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 case is large because of the unfavorable flow upstream of the pump impeller.

The present work examines numerical results using OpenFOAM™ of the flow in the swirl flow generator test rig developed at Politecnica 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 k-ε model, and block structured wall function ICEM-CFD meshers are used. The numerical results are validated against experimental LDV data, and against design velocity profile data. 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 are 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 overestimated.

The performance characteristics of centrifugal pumps were measured experimentally when running with tap water and drag-reducing surfactant (Didodecyl 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 (3150rpm, 3250rpm, 3150rpm and 2900rpm). Compared with tap water case, the experimental results show that the total pump heads for surfactant cases are higher. And the pump efficiency with surfactant solutions also increases, but the shaft power for surfactant solutions cases decreases compared to tap water case. There exists an optimal temperature for surfactant solutions, which maximizes the pump efficiency.

The performance characteristics of centrifugal pump were measured experimentally when running with tap water and drag-reducing surfactant (Didodecyl 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 (3150rpm, 3250rpm, 3150rpm and 2900rpm). Compared with tap water case, the experimental results show that the total pump heads for surfactant cases are higher. And the pump efficiency with surfactant solutions also increases, but the shaft power for surfactant solutions cases decreases compared to tap water case. There exists an optimal temperature for surfactant solutions, which maximizes the pump efficiency.

The performance characteristics of centrifugal pump were measured experimentally when running with tap water and drag-reducing surfactant (Didodecyl 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 (3150rpm, 3250rpm, 3150rpm and 2900rpm). Compared with tap water case, the experimental results show that the total pump heads for surfactant cases are higher. And the pump efficiency with surfactant solutions also increases, but the shaft power for surfactant solutions cases decreases compared to tap water case. There exists an optimal temperature for surfactant solutions, which maximizes the pump efficiency.

The performance characteristics of centrifugal pump were measured experimentally when running with tap water and drag-reducing surfactant (Didodecyl 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 (3150rpm, 3250rpm, 3150rpm and 2900rpm). Compared with tap water case, the experimental results show that the total pump heads for surfactant cases are higher. And the pump efficiency with surfactant solutions also increases, but the shaft power for surfactant solutions cases decreases compared to tap water case. There exists an optimal temperature for surfactant solutions, which maximizes the pump efficiency.

The performance characteristics of centrifugal pump were measured experimentally when running with tap water and drag-reducing surfactant (Didodecyl 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 (3150rpm, 3250rpm, 3150rpm and 2900rpm). Compared with tap water case, the experimental results show that the total pump heads for surfactant cases are higher. And the pump efficiency with surfactant solutions also increases, but the shaft power for surfactant solutions cases decreases compared to tap water case. There exists an optimal temperature for surfactant solutions, which maximizes the pump efficiency.

The performance characteristics of centrifugal pump were measured experimentally when running with tap water and drag-reducing surfactant (Didodecyl 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 (3150rpm, 3250rpm, 3150rpm and 2900rpm). Compared with tap water case, the experimental results show that the total pump heads for surfactant cases are higher. And the pump efficiency with surfactant solutions also increases, but the shaft power for surfactant solutions cases decreases compared to tap water case. There exists an optimal temperature for surfactant solutions, which maximizes the pump efficiency.

The performance characteristics of centrifugal pump were measured experimentally when running with tap water and drag-reducing surfactant (Didodecyl 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 (3150rpm, 3250rpm, 3150rpm and 2900rpm). Compared with tap water case, the experimental results show that the total pump heads for surfactant cases are higher. And the pump efficiency with surfactant solutions also increases, but the shaft power for surfactant solutions cases decreases compared to tap water case. There exists an optimal temperature for surfactant solutions, which maximizes the pump efficiency.
<table>
<thead>
<tr>
<th>Page</th>
<th>Title</th>
<th>Author</th>
<th>Abstract</th>
<th>Keywords</th>
</tr>
</thead>
<tbody>
<tr>
<td>287-288</td>
<td>Review of Mathematical Models in Performance Calculation of Screw Compressors</td>
<td>Nikola Stosic, Ian E. Smith, Ahmed Kowasan, Edbid Mojic</td>
<td>The mathematical modeling of screw compressor processes and its implementation in their design began about 35 years ago with the publication of several pioneering papers on this topic, mainly at the Purdue Compressor Conference. This led to the gradual introduction of computer aided design, which, in turn, resulted in huge improvements in these machines, especially in oil-flooded at 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 optimization 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.</td>
<td>Screw Compressor, Mathematical Model, Performance Calculation</td>
</tr>
<tr>
<td>324-331</td>
<td>Phase Resonance in a Centrifugal Compressor</td>
<td>Yoichi Kinos, Norimasa Shiomi, Toshiaki Setoguchi</td>
<td>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 near the hub and the suction surface of the stator blade was presented. Separation vortices were observed in the limiting streamline patterns both on the blade suction side and on the hub surface 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.</td>
<td>Diagonal flow fan, Axial flow stator, Internal flow, Corner separation, Five-hole probe survey, CFD</td>
</tr>
<tr>
<td>207-212</td>
<td>Vortices Features in a Half-ducted Axial Fan with Large Bellmouth (Effect of Tip Clearance)</td>
<td>Norimasa Shiomi, Yoichi Kinos, Toshiaki Setoguchi, Kenji Kaneko</td>
<td>As an example, 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 were tested: those sizes were 1mm (0.55% of blade chord length at blade tip), 2mm (2.61% of blade chord length at blade tip) and 4mm (2.23% of blade chord length at blade tip), and those sizes were shown as TC=1mm, TC=2mm and TC=4mm, respectively. Fan characteristic tests and the velocity field measurements were done for each TC. A flow -rate characteristics and two-dimensional 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 these tip clearances sizes. In case of TC=4mm, the tip leakage vortices was outflow to downstream, or 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=3mm, 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.</td>
<td>Vortex leakage vortices, Tip clearance, Ventilation fan, LDV measurement</td>
</tr>
<tr>
<td>317-322</td>
<td>Influence of Blade Outlet Angle and Blade Thickness on Performance and Internal Flow Conditions of Mini Centrifugal Pump</td>
<td>Toru Shigemitsu, Junichiro Fukutomi, Kensuke Kaji</td>
<td>The experimental research was conducted on mini centrifugal pumps having a diameter smaller than 100mm in many fields: automobile, automobile radiator pump, interior unit, pump, domestic pump, and so on. Further, the needs for mini centrifugal pumps would become larger with the increase of the application of it for many fields; automobile radiator pump, interior unit, pump for electric devices and so on. The mini centrifugal pumps would become larger with the increase of the application of it for electronic devices. It is desirable that the mini centrifugal pump design be as simple as possible as precise manufacturing is required. But the design method for mini centrifugal pumps 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 this, 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 numerical analysis was 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 flow condition is clarified with the results of the experiment and the numerical flow analysis.</td>
<td>Mini centrifugal pump, Performance, Internal flow condition, Blade outlet angle, Blade thickness</td>
</tr>
<tr>
<td>234-237</td>
<td>Phase Resonance in a Centrifugal Compressor</td>
<td>Fumoto Nishiyama, Takakuki Suzaki, Koichi Yonezawa, Hiroshi Tanaka, Peter Doerfler, Yoshinobu Tsujimoto</td>
<td>In the development of very high head pump stored projects, one of the critical problems is the strength of pump turbine runners. Data obtained by stress measurements at high head pump-turbine runners indicated that dynamic stress due to the vibration of runner might be detrimental; possibility to cause fatigue failure, if the runner were designed without proper consideration on dynamic behavior. Numerous field stress measurements of runners and model tests conducted with hydrodynamic similarities 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 frequency of runner have been conducted to explore this forced vibration problem.</td>
<td>Vibration behavior, Dynamic stress, Hydraulic excitation force, Vibration mode, Prototype head model test</td>
</tr>
</tbody>
</table>
Effects of the Geometry of Components Attached to the Drain Valve on the Performance of Water Hammer Pumps

Masami Suzuki, Toshihiko Setoguchi, Kenji Kanieko

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 the 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 drive pipe.

Keywords:
Water Hammer Pump, Fluid Transients, Pump Performance, Pressure Fluctuation, Flow Visualization

Centrifugal pump, impeller, vortex flow, pressure fluctuation

Centrifugal flow, helical swimming motion, bio-renewable number, computational fluid dynamics

Circular-arc blades, Cascade, Guide Vane, Potential Flow, CFD

Circular-arc blades, Cascade, Guide Vane, Potential Flow, CFD
Numerical Analysis of the Influence of Acceleration on Cavitation Instabilities that arise in Cascade

**Title:** Numerical Analysis of the Influence of Acceleration on Cavitation Instabilities that arise in Cascade

**Authors:** Yulja Ipa, Taeoul Kwon

**Abstract:** 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 cavitation and cavitation surge 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-cavitation-number 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.

**Keywords:** Cavitation Instability, Cascade, Acceleration, Homogeneous Model, CFD

---

**Title:** Experimental Study on Adjustment of Inlet Nozzle Section to Flow Rate Variation for Darrieus-type Hydro-Turbine

**Authors:** Satoshi Watanabe, Katsuhiko Kurata, Akira Funakawa, Kusao Okuma, Dankei Matsumata

**Abstract:** A two-dimensional Darrieus-type turbine has been proposed for the hydrospenser 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 Darrieus turbine system, the Darrieus turbine should be operated with high efficiency in the wider range of flow rate possibility by using an additional device with simple 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.

**Keywords:** Hydro-turbine, Darrieus-type runner, Inlet nozzle, Partial inlet flow, Self-starting characteristics

---

**Title:** Oxygen Transfer Characteristics of an Ejector Aeration System

**Authors:** Hee-Cheon Yang, Sang-Kyoo Park

**Abstract:** 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 aeration water temperature on the oxygen transfer efficiency was investigated. The dissolved oxygen concentration increased with increasing entrained air flow rate, but decreased with increasing aeration water temperature for two aeration systems. The volumetric mass transfer coefficients increased with increasing entrained air flow rate and with increasing aeration water temperature for both aeration systems. The average mass transfer coefficients for the ejector aeration system was about 20% and 4% higher than that of the blower aeration system within the experimental range of entrained air flow rates and aeration water temperatures.

**Keywords:** Ejector aeration system, Dissolved oxygen, Oxygen mass transfer, Aeration water, Entrained air
Improving Flow Distribution in a Suction Channel for 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 Fluent code and experimental data from the literature.

Vol. 5, No. 1, April-June, 2012

Axial Wall Slit 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 DVR when the rotating Reynolds number is small. The wall slits were axially located along the inner cylinder and the slits’ number of each model was 9 and 18. Another plan-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 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.

Computational Investigations of Impingement Heat Transfer on an Elliptic Condenser 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 jet impinging on a condenser surface with four rows of effusion holes. The effects of exit configurations of jet air and the arrangement of jet orifices and effusion holes for a jet Reynolds number of 5300 is investigated. In all eight cases are studied and a good qualitative correlation is found among the five flow patterns, pressure variations and heat transfer distributions.

Effects of Management of High Speed Flexible Coupling on the Fighter Aircraft Transmission
Nagelit Samikoon, Aliu Mohammed Junaid Jachta

The Fighter aircraft transmission system consists of a light weight, high speed Flexible Coupling (HSC) known as PowerTake-Off shaft (PTO) for connectivity. Flexible Couplings, Management, rotor dynamics, aerodynamic system dynamics and simulation, Experimental Design parameters, Suction channel, Compressor, Stage efficiency, Circumferential flow distortion

Improving Flow Distribution in a Suction Channel for a Highly Efficient Centrifugal Compressor
Matsuo Yagi, Takanori Shibata, Hiroki Kobayashi, Masanori Tanaka, Hiroshi Ishida

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 dynamic (CFD) calculation, the passage sectional area ratio Ac/Ae, Ae/As and Ac/As were shown to be the dominant parameters for the pressure loss and circumferential flow distortion at upstream, where Ac and As are passage cross sectional areas at the stator and impeller outlet 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 Optimised suction channel achieved 3.8% higher stage efficiency than the Base suction channel while maintaining the same operating range.

Effects of the Lift Valve Opening Area on Water Hammer Pump Performance and Flow Behavior in the Valve Chamber
Surin Saith, Keita Dejima, Masaaki Takahashi, Galio Hiyatke, 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 pump sectional area ratio Ac/Ae, Ae/As and Ac/As were shown to be the dominant parameters for the pressure loss and circumferential flow distortion at upstream, where Ac and As are passage cross sectional areas at the stator and impeller outlet 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 Optimised suction channel achieved 3.8% higher stage efficiency than the Base suction channel while maintaining the same operating range.

Vol. 5, No. 2, July-September, 2012

Asper-Ratio Effects and Uneven Pressure Measurements inside a Cross-Flow Impeller
Katsuya Hira, Yusuke Ono, Shigeyo Nagaoka, Ryosuke Matsuzato, Juri 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/D2. In order to eliminate the influence of the strong centrifugal flow, the impeller rotates in open space without any casings. As a result, by using hot wire anemometer measurements and by conventional flow visualizations with a particle image velocimetry technique, the authors show that both the outlet flow and the maximum vortex area and the maximum for L/D2 = 0.6. In order to investigate the aspect-ratio effect, they further reveal minute fluctuating pressure on an impeller and found for a singular L/D2 = 0.6. Especially in these pressure measurements, the acentric vortex is prevented to resolve by the insertion of a tongue, in order to consider the spatial structure of flow more precisely.

Cross-Rotation Flow, Blower, Fan, Asper-Ratio, Pressure Measurement

Recirculation of Dynamic Transfer Matrix and Unsteady Cavitation Characteristics in an Inducer
Koichi Yonezawa, Jun Amino, Donghyuk Kang, Hitoshi Honjou, Yutaka Kawata, Yoshinobu Tajimasto

The transfer matrix and unsteady cavitation characteristics, cavitation compliance and mass flow gain factor, of cavitation 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.

Induced, dynamic transfer matrix, cavitation compliance, mass flow gain factor, CFD
<table>
<thead>
<tr>
<th>Page</th>
<th>Title</th>
<th>Author</th>
<th>Abstract</th>
<th>Keywords</th>
</tr>
</thead>
<tbody>
<tr>
<td>145-147</td>
<td>Experimental and computational analysis of behavior of three-way catalytic converter under axial and radial flow conditions</td>
<td>Anil Zalik Abadi, Vivas Malakar</td>
<td>The competition to deliver ultra-low emitting vehicles at a reasonable cost is driving the automotive industry to invest significant manpower and 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 development 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 catalytic chemical kinetics 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-ε model is used for turbulence. The flow pattern is changed from axial to radial flow 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 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.</td>
<td>CFD modeling, chemical reaction, performance, efficiency, simulation</td>
</tr>
<tr>
<td>152-155</td>
<td>Cavitating Surge in a Small Model Test Facility Simulating a Hydraulic Power Plant</td>
<td>Koichi Hozawa, Daisuke Komori, Katsuyoshi Kayagawa, Franjoz A. Ktny, Peter Dressler, Yoshikazu Tugimoto</td>
<td>This paper presents a numerical investigation on the aerodynamic performance of a Centrifugal Fan with Splitter Blades. A numerical study to investigate the effect of intake vortex occurrence on the performance and internal flow condition of different blade row distances are shown and the blade row performance and the internal flow condition are discussed in this paper.</td>
<td>Numerical analysis, Performance, Internal flow, Blade row distance</td>
</tr>
<tr>
<td>156-170</td>
<td>Numerical Investigation on Aerodynamic Performance of a Centrifugal Fan with Splitter Blades</td>
<td>Tomoaki Kim, Kyung-Chan Hong, Choon-Man Jang</td>
<td>This paper presents a numerical investigation on the aerodynamics performances according to the application of splitter blades in an impeller of a centrifugal fan.</td>
<td>Centrifugal fan, impeller, splitter, performance, efficiency, Reynolds-averaged Navier-Stokes equations</td>
</tr>
<tr>
<td>174-179</td>
<td>Effect of Intake Vortex Occurrence on the Performance of an Axial Hydraulic Turbine in Shihwa Lake Tidal Power Plant, Korea</td>
<td>In-Hyuk Kim, Min-Moong Hae, Kyung-Hun Cha, Keong-Yong Kim, Choong-Man Jang, Hong, Marie Collins</td>
<td>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 Shihwa-lake tidal power plant, Korea is performed. Numerical analysis of flow through an axial hydraulic turbine is carried out by solving three-dimensional Reynolds-averaged Navier-Stokes equations with standard wall function 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 field 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.</td>
<td>Small-sized axial fan, Contra-rotating rotors, Performance, Internal flow, Blade row distance</td>
</tr>
<tr>
<td>1-10</td>
<td>Flow Analyses Inside Jet Pumps Used for Oil Wells</td>
<td>Abdus Samed, Mohamed Nizamuddin</td>
<td>An pump is one type of artificial lift 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 relations of area and length to diameter of the mixing tube were considered as design parameters. The pump efficiency was considered to maximize for the given 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 larger throat length increased efficiency up to 40% if light oil was used as primary fluid and secondary fluid viscosity was 350 cSt.</td>
<td>Artificial lift, well pumping, jet pump, hydraulic lift, primary fluid, secondary fluid</td>
</tr>
<tr>
<td>11-17</td>
<td>Performance and Internal Flow Condition of Mini Centrifugal Pump with Splitter Blades</td>
<td>Taru Ishikawa, Junichi Kubota, Kenji Yamauchi, Takashi Wada</td>
<td>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 in this research. Splitter blades were adopted in this research to improve the performance and the internal flow condition of mini centrifugal pump. The experiments were conducted with the splitter blades 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.</td>
<td>Mini centrifugal pump, Performance, Internal flow, Splitter blade</td>
</tr>
</tbody>
</table>
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 400-700 (m·min⁻¹, m³/min, m⁻¹) radial-outflow type diagonal flow fan which specific speed was 670 (m·min⁻¹, m³/min, m⁻¹) was designed by a two-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

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 bearing. The PTFE composite guide bearing was tested by experimental operation 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 plants are reduced.

Water lubricated guide bearing, PTFE, Tilting segment, Vertical shaft hydro turbine, Hydrodynamic lubrication

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 end. The pressure wave measurements showed that there are certain effects of aerodynamic and geometrical factors on the change of the characteristic region, one is tip leakage vortex near rotor tip and another is pressure surface separation on the rotor blade.

Compressor, surge, stall, stall initiation, unsteady flow, flow oscillation

Data centers have been built with spread of cloud computing. Further, electric energy consumption, which occurred along the stator hub-suction surface, changes the exit flow 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 end. The pressure wave measurements showed that there are certain effects of aerodynamic and geometrical factors on the change of the characteristic region, one is tip leakage vortex near rotor tip and another is pressure surface separation on the rotor blade.

Phase resonance, acoustic resonance, rotor-stator interaction, centrifugal fan

In order to study the effect of fluid-structure interaction (FSI) on the design results, the external characteristics and internal flow features of a five-hole probe.

Fluid-Structure Interaction, Diffuser Pump, Two-Way Coupling Method, head prediction

Yoichi Kinoue, Norimasa Shiomi, Toshiaki Setoguchi

Effects of Acoustic Resonance and Volute Geometry on Phase Resonance in a Centrifugal Fan

Nobuyuki Yamaguchi

An Outlook on the Draft Tube-Surge Study

Mitsühiro 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 IAW 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, vortex rope, cavitation, review paper

Design and Experimental Studies of Radial-Outflow Type Diagonal Flow Fan

Toshiyuki Iwashita, Jiroshi Nishimori, Takuya Aoyagi

Internal Flow Condition of High Power Contra-Rotating Small-Sized Axial Fan

Toshiyuki Iwashita, Jiroshi Nishimori, Takuya Aoyagi

Data centers have been built with spread of cloud computing. Further, electric energy consumption, which occurs along the stator hub-suction surface, changes the exit flow 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 end. The pressure wave measurements showed that there are certain effects of aerodynamic and geometrical factors on the change of the characteristic region, one is tip leakage vortex near rotor tip and another is pressure surface separation on the rotor blade.

Compressor, surge, stall, stall initiation, unsteady flow, flow oscillation

Page | Title | Author |
--- | --- | --- |
40-55 | Water Lubricated Guide Bearing with Self-aligning Segments | Tadashi OKUYA, Naohiro NAKAGAWA, Masato UKAMI, Long THANTRONG, Yasuyuki IZUMA, Fumio TAGOMOTO |
56-74 | Analytical Study on Stall Stagnation Boundaries in Axial-Flow Compressor and Duct Systems | Nobuyuki Yamaguchi |
75-85 | Effects of Acoustic Resonance and Volute Geometry on Phase Resonance in a Centrifugal Fan | Yoshishika Tegami, Hiroshi Tanaka, Peter Cheung, Koushi Yonezawa, Takayuki Suzuki, Keisuke Makimura |
87-93 | Fluid Structure Interaction Study on Diffuser Pump With a Two-Way Coupling Method | Xu Huan, Liu Houdong, Tan Minggao, Cui Jianbao |
18-24 | Design and Experimental Studies of Radial-Outflow Type Diagonal Flow Fan | Yoichi Kinoue, Norimasa Shiomi, Toshiaki Setoguchi |
25-32 | An Outlook on the Draft Tube-Surge Study | Mitsühiro Nishi, Shuhong Liu |
33-48 | An Outlook on the Draft Tube-Surge Study | Mitsühiro Nishi, Shuhong Liu |
49-55 | An Outlook on the Draft Tube-Surge Study | Mitsühiro Nishi, Shuhong Liu |
56-74 | An Outlook on the Draft Tube-Surge Study | Mitsühiro Nishi, Shuhong Liu |
75-85 | Effects of Acoustic Resonance and Volute Geometry on Phase Resonance in a Centrifugal Fan | Yoshishika Tegami, Hiroshi Tanaka, Peter Cheung, Koushi Yonezawa, Takayuki Suzuki, Keisuke Makimura |
87-93 | Fluid Structure Interaction Study on Diffuser Pump With a Two-Way Coupling Method | Xu Huan, Liu Houdong, Tan Minggao, Cui Jianbao |
18-24 | Design and Experimental Studies of Radial-Outflow Type Diagonal Flow Fan | Yoichi Kinoue, Norimasa Shiomi, Toshiaki Setoguchi |
25-32 | An Outlook on the Draft Tube-Surge Study | Mitsühiro Nishi, Shuhong Liu |
33-48 | An Outlook on the Draft Tube-Surge Study | Mitsühiro Nishi, Shuhong Liu |
49-55 | An Outlook on the Draft Tube-Surge Study | Mitsühiro Nishi, Shuhong Liu |
56-74 | An Outlook on the Draft Tube-Surge Study | Mitsühiro Nishi, Shuhong Liu |
75-85 | Effects of Acoustic Resonance and Volute Geometry on Phase Resonance in a Centrifugal Fan | Yoshishika Tegami, Hiroshi Tanaka, Peter Cheung, Koushi Yonezawa, Takayuki Suzuki, Keisuke Makimura |
87-93 | Fluid Structure Interaction Study on Diffuser Pump With a Two-Way Coupling Method | Xu Huan, Liu Houdong, Tan Minggao, Cui Jianbao |
18-24 | Design and Experimental Studies of Radial-Outflow Type Diagonal Flow Fan | Yoichi Kinoue, Norimasa Shiomi, Toshiaki Setoguchi |
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 present 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 ones 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 cavitation 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.

Keywords: Counter Rotating Turbine, Axial Spacing, Turbine Performance, Counter-rotating rotors, blade design, cavitation, CFD.
A second order exact scaling method for turbomachinery performance prediction

Peter Franz Palz, Stefan Sebastian Stojpek

A scaling method valid for most turbomachines based on first principles is derived. It accounts for axial and centrifugal turbomachinery with respect to relative gap width, tip clearance, relative roughness, Reynolds number and Mach number for design and off design operation as well. The method has been successfully validated by a vo-y of experimental data obtained at TU Graz. The physically based, hence reliable and universal method is compared with previous, empirical scaling methods.

Development of a simulation method of surge transient flow phenomena in a multistage axial flow

Nobuyuki Yamaguchi

A practical method of surge simulation in a system of a high pressure ratio, multimodule axial flow compressor and duct, named DRUMTAN, is described.

The numerical simulation of unsteady flow in a mixed flow pump guide vane

Li Yun, Li Ren-nian, Wang Xiao-yong

In order to investigate the characteristics of unsteady flow in a mixed flow pump vane under the small flow conditions, several indicator points in a mixed flow pump guide vane vane set, the three-dimensional unsteady turbulence numerical value of the mixed flow pump which 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 blade tip 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 evenly symmetric 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.

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 univustly 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 least bandwidth of amplitude in low frequency range will lead to the wider zero of outlet velocity in it's time domain, and the larger force of the strike will be gained. By studying the St of self-excited nozzle, not only the frequency of certain nozzle can be predicted, but also a nozzle structure with a certain frequency can be designed.

Experimental study on the performance of a propulsive nozzle with a booster piping system

Masahiko Sakamoto

The characteristics of the thrust for the propulsive equipment directly driven by air compressor by pressure fluctuation in a booster piping system are investigated. The exhaust valve is positioned upon the suction hole in the pipe discharge pipe in order to induce the large-scale pressure fluctuation, and the effects of the valve on the pressure in the pipe and the thrust for the propulsive nozzle are examined. The pressure in the pipe decreases immediately after the valve is opened, and it increases just before the valve is closed. The thrust for the propulsive nozzle monotonously 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.

1-4 Experimental Investigation on Fluid Transportation Performance of Propellant Acquisition Vane in Microgravity Environment

Baotang Zhuang, Yong Li, Xiumei Luo, Hail 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 efficienctly. 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 PAVs. PAVs with bottom length ratio of 0.5, 0.75 and 1 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 type PMDs, but also are helpful for fluid control applications in space environment.

7-15 Numerical investigation on the characteristics of flow-induced noise in a centrifugal blower

Chanyoung Lee, Taehoon Jeong, Kyung-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 a turbulence model using CFLO 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 pressure fluctuation at impeller outlet represents relative overall sound level of the blower. Sound spectra show that there are some specific peak frequencies at each mass flow rate and it can be explained by flow pattern.

Predicting Double-Blade Vertical Axis Wind Turbine Performance by a Quadruple-Multiple Streamtube Model

Jaka Hara, Takakuni Kawamura, Hisanori Akimoto, Kenji Tanaka, Takuya Nakamuro, Kenjiro Kikutsumoto

Double-blade vertical axis wind turbines (DB-VWTs) can improve the self-oscillating 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-VWT performance. Model validity is investigated by comparison to computational fluid dynamics (CFD) predicitions for two kinds of two-dimensional DB-VWT rotors for two rotor scales with three inner-outlet radius ratios: 0.2, 0.3, and 0.75. The BEM-QMS model does not consider the effects of an inner rotor on the flow speed in the upstream 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-VWT is thus closer to the CFD prediction.

28-17 Numerical simulation analysis of tip clearance flow in a centrifugal compressor

Guosheng Zhou, Jun Wang, Chuanhui 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 seven compressor stages, which were 0.5 times, 1 times, 1.5 times and 2.5 times of the design tip clearance was carried on. Efficiency and pressure ratio curves were obtained under different mass flow rates. 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.

Microgravity, Propellant Acquisition Vane, Fluid Transportation, Experiment

Wind turbine, Double-blade rotor, VAWT, BEM, CFD, QMS

Leakage flow, Tip clearance, Pressure ratio, Pressure difference

177-187

Volume

Page

1-6

177-187

189-199

200-205

206-212

213-218

219-224

225-231

232-237

238-243

244-249

250-255

350-356

412-417

418-423

424-429

430-435

436-441

442-447

448-453

454-459

460-465

466-471

472-477

478-483

484-489

490-495

496-501
<table>
<thead>
<tr>
<th>Page</th>
<th>Title</th>
<th>Author</th>
<th>Keywords</th>
</tr>
</thead>
<tbody>
<tr>
<td>45-53</td>
<td>Phase Resonance in Centrifugal Fluid Machinery: A Comparison between Pump Mode and Turbine</td>
<td>Yoichi Yonekawa, Shingo Toyohara, Shingo Motsuki, Yoshihiko Tanaka, Peter Dheel, Yoshikazu Tsujimoto</td>
<td>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 waveform of the load, and which has a great heat transfer surface 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 nucleates 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.</td>
</tr>
<tr>
<td>54-59</td>
<td>Research of liquid-solid two phase flow in centrifugal pump with crystallization phenomenon</td>
<td>Liu Dong, Wang Ya-yun, Wang Ying-ze, Wang Chun-li, Yang min-guan</td>
<td>In order to clarify the effect of inlet bellmouth size of semi-opened type axial fan on its performance and test flow fields around rotor, fan test and flow field measurements using hotwire anemometry 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 nearly became zero, and the best efficiency point became small with the bellmouth size decreasing. Therefore, it is concluded that the bellmouth size should be large. 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.</td>
</tr>
<tr>
<td>60-67</td>
<td>Effect of Inlet Geometry on Fan Performance and Inlet Flow Fields in a Semi-opened Axial Fan</td>
<td>Pin-Liu, Norimasa Shomoi, Yoichi Kinno, Toshiki Setoguchi, Ying-zh Lin</td>
<td>In this study, we visualized the internal flow of a regenerative turbomachinery. Fluid is injected at the inlet of the machinery and the streak were recorded using a high-speed camera with high-power light source. While circulating inside the axial turbomachinery, 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 returned into the groove and circulated again. Then the droplets either flowed to the outlet or to the stopper. As a result, this experiment has confirmed the internal circulating flow of a regenerative turbomachinery.</td>
</tr>
<tr>
<td>68-79</td>
<td>Design and Simulation of Very Low Head Axial Hydraulic Turbine with Variation of Swirl Velocity Criterion</td>
<td>Abdul Muak, Priyono Sutikno</td>
<td>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 color (magenta color). Droplets were injected at the inlet of the machinery and the streak were recorded using a high-speed camera with high-power light source. While circulating inside the axial turbomachinery, 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 returned into the groove and circulated again. Then the droplets either flowed to the outlet or to the stopper. As a result, this experiment has confirmed the internal circulating flow of a regenerative turbomachinery.</td>
</tr>
<tr>
<td>80-83</td>
<td>Visualization of Flow inside a Regenerative Turbomachinery</td>
<td>Yang Hyunmook, Lee Kyung-song, Chi Youngyong Jeong Kyung-seok</td>
<td>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, CFD, is introduced in the present work. SST turbulence model is employed for the energy dissipation. Throughout the numerical analysis, it is found that the thickness of impeller tip is effective to increase the efficiency in the present blower. Pressure is successfully increased up to 2.8 bar compared to the reference blower at the design flow condition. And efficiency is also enhanced up to 2.8 % compared to the reference one. It is noted that low velocity region existed to make strong circulation flow inside the blade passages, thus increases local pressure loss. Detailed flow field inside the regenerative blower is also analyzed and compared.</td>
</tr>
<tr>
<td>86-89</td>
<td>Performance Enhancement of 20KW Regenerative Blower Using Design Parameters</td>
<td>Choon-Allen Jung, Hyun-Jin Jeon</td>
<td>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 caused by 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.</td>
</tr>
<tr>
<td>94-106</td>
<td>Study of Cavitation Instabilities in Double-Suction Centrifugal Pump</td>
<td>Shinya Hatono, Donghyuk Kang, Shushuai Kajewa, Motokiho Nozomi, Kaasuhito Yokota</td>
<td>In this study, we visualized the internal flow of a regenerative turbomachinery. Fluid is injected at the inlet of the machinery and the streak were recorded using a high-speed camera with high-power light source. While circulating inside the axial turbomachinery, 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 returned into the groove and circulated again. Then the droplets either flowed to the outlet or to the stopper. As a result, this experiment has confirmed the internal circulating flow of a regenerative turbomachinery.</td>
</tr>
<tr>
<td>107-110</td>
<td>Blood Pump, Ventricular Assist device, LVAD, BIAD, Computational fluid dynamics, artificial heart</td>
<td>Jeerasit Bumrungpetch, Andy Chit Tan, Shu-Hong Liu, Dong Liu, Wang Ya-yun, Wang Ying-ze, Wang Chun-li, Jeong Kyungseok, Zhang Zhe, Setoguchi, Ying-zi Jin</td>
<td>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 (BIAD) which combines the functions of LVAD and BIAD in a compact unit. The BIAD construction consists of two separate chambers with similar impellers, volutes, inlet and outlet sections. To achieve the required flow characteristics of an average flow rate of 5 l/min and different pressure heads (left – 100mmHg and right – 200mmHg), the impellers were set at different rotating speeds. From the CFD results, a six-blade impeller design was adopted for the development of the BIAD. 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 thrombosis and hemolysis. Based on the compatible Reynolds number the flow through the model was calculated for the left and 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.</td>
</tr>
</tbody>
</table>
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 by considering the 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.

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 by considering the 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.

Surge phenomena are strongly related with the flow in the coupled impeller-pump system. This phenomenon is known as a self-excited oscillation and is commonly referred to as the surge phenomenon. The surge phenomenon is characterized by a sudden drop in the flow rate, which results in a drop in the pressure. This drop in pressure causes the compressor to operate in a suboptimal mode, resulting in a decrease in efficiency and a potential risk of damage. The surge phenomenon can be studied using both experimental and computational methods. In this study, the surge phenomenon was investigated experimentally for a three-stage axial flow compressor system in the reduced-speed zone. The experimental results were compared with those obtained from numerical simulations. The results showed good agreement between the experimental and numerical results, indicating the validity of the numerical simulation method. The study also highlighted the importance of the parameters such as the rotational speed of the compressor and the back pressure on the surge characteristics. These findings can be used to design and optimize the performance of axial flow compressors in real-world applications.
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 runner changes with variation in the rotational speed such that the flow field itself is significantly altered. Thus, clear understanding of the flow field 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.

Keywords: Centrifugal Pump, Artificial Heart; Blood, Hemolysis; Thrombosis

Centrifugal Pump, Air conditioning, Heat pump, Electric bus, Efficiency, Variable frequency

Thrust-ring-pump, Centrifugal pump, Hydraulic design, Turbine generator, Low specific speed centrifugal pump

The Flow Field of Undershoot Cross-Flow Water Turbines Based on PIV Measurements and Numerical Analysis

Yasuyuki Nishi, Terumi Iragai, Yanning Li, Ryota Omiya, Kentaro Hatano

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 200 mmHg or higher, due to the high hydraulic resistance of the membrane oxygenator 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.

Performance Evaluation of a Variable Frequency Heat Pump Air-Conditioning System for Electric Bus

Qinghong Peng, Qingyu Du

This paper presents the 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 the 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 range of EVs because of the highest efficiency and the lowest energy consumption of AC system.

Thrust-ring-pump, Centrifugal pump, Hydraulic design, Thrust bearing, Lubricating and cooling system, Hydro dynamics

23-35
Rotordynamic Performance Measurements of An Oil-free Turbocharger Supported on Gas Foil Bearings and Their Comparisons to Floating Ring Bearings

Yang-Bok Lee, Dong-In 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 companion to floating ring bearings (FRBs). Oil-free TC was designed and developed via the rotodynamic analysis using dynamic force coefficients from GFB analyses. The rotordynamics and performance of the oil-free TC was measured up to 85 kpm 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 450 W. Incidentally, an external shock test on the oil-free TC casing was conducted at the rotor speed of 60 kpm, and showed a good capability of vibration damping due to the well-known dry friction mechanism of the GFBs.

Thrust-ring-pump, Centrifugal impeller, Off design condition, Rotational speed, Viscosity, Inlet recirculation

36-45
Analysis of Centrifugal Impeller for Different Viscous Fluids

Sayed Ahmed Irmen Bellary, Abdul 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. Reynold averaged Navier Stokes (RANS) equations coupled with viscous flow solver and the performance analysis was made. Standard two equation k-ε 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.

Thrust-ring-pump, Centrifugal pump, Hydraulic design, Turbine generator, Low specific speed centrifugal pump

46-54
Hydrodynamic Design of Thrust Ring Pump for Large Hydro Turbine Generator Units

Kate Li, Xian Zhang, Xiaomin Chen, Shila 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 circulation 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 eliminates to impeller, and oil both equivalent to equivalent has great influence on hydropneumatic 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 circulation and cooling system. As far as the empirical calculation method is employed during the actual engineering design, in order to guarantee the operating performance of the oil circulation and cooling system with thrust-ring-pump at different conditions, a collaborative hydrodynamic design and optimization is pursued in this paper. Firstly, the head and flow rate at different conditions are decided by 3D flow numerical simulation of the oil circulation and cooling system. Secondly, the flow passage 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 circulation and cooling system are collaborative hydrodynamic optimized with predicted head-flow rate curve and the efficiency-flow rate curve of thrust-ring pump. The present methodology has been adopted in the design of thrust-ring pump for hydro turbine generator units with high specific speed.

Method of characteristics, Hydraulic transient, Discharge propagation, Saint-Venant equations, Numerical Approach, Hydraulic Systems

55-62
Case studies for solving the Saint-Venant equations in two cases studies: 1) a peristaltic model 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 DC concerning the development of hydrodynamics 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.

Case studies for solving the Saint-Venant equations, Numerical Approach, Hydraulic Systems

Vol. 8, No. 1, January-March, 2014
### 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 unidirectional 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.

<table>
<thead>
<tr>
<th>Author</th>
<th>Title</th>
<th>Keyword(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximilian Manderla, Karl N. Kiniger, Jiri Koutnik</td>
<td>Improved prediction of Pump Turbine Dynamic Behavior using a Thoma number dependent Hill Chart and Site Measurements</td>
<td>Water hammer phenomena, Dynamic transients, Pump-turbine, Site measurements</td>
</tr>
</tbody>
</table>

### Bubble size characteristics in the wake of ventilated hydrofoils with two aeration configurations

Ashish Kari, Christopher R Ellis, Christopher Milliner, Parasing Hong, David Scott, Roger I A Arnott, John S. Gulliver

Aerating hydrofoils 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 hydroaerator requires a precise understanding of the dependence of the generated bubble size distribution upon the operating conditions (i.e. liquid velocity, air supply rate, hydrofoil configuration, etc.) and the consequent rise in dissolved oxygen in the wake. As part of the current research we wish to investigate the effect of location of air injection on the resulting bubble size distribution, thus leading to a quantitative analysis of aerator statistics and capabilities for two turbine blade hydrofoil designs. The two blade designs differed in their location of an injection. Extensive sets of experiments were conducted by varying the liquid velocity, air supply 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 on 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.

<table>
<thead>
<tr>
<th>Author</th>
<th>Title</th>
<th>Keyword(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ashish Kari, Christopher R Ellis, Christopher Milliner, Parasing Hong, David Scott, Roger I A Arnott, John S. Gulliver</td>
<td>Bubble size characteristics in the wake of ventilated hydrofoils with two aeration configurations</td>
<td>Hydrofoil design, Aeration, Air supply rate, Hydrofoil configuration, Bubble size distribution</td>
</tr>
</tbody>
</table>

### Numerical studies on cavitation behavior in impeller of centrifugal pump with different blade profiles

Pengfei Song, Yonggui Zhang, Cong Xu, Xin Zhou, Jinya Zhang

In the Francis 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 Francis runners have recently experienced cracks in the runner blades due to fatigue, obviously due to the runner 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 3D NASTRAN 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 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 Timess, and for the simulations presented here the variations in Timess are found to be around 5 % 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.

<table>
<thead>
<tr>
<th>Author</th>
<th>Title</th>
<th>Keyword(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pengfei Song, Yonggui Zhang, Cong Xu, Xin Zhou, Jinya Zhang</td>
<td>Numerical studies on cavitation behavior in impeller of centrifugal pump with different blade profiles</td>
<td>Centrifugal pump, Blade profile, Dynamics, Grid frequency variations</td>
</tr>
</tbody>
</table>

### Simulations of the Dynamic Load in a Francis Runner based on measurements of Grid Frequency Variations

Kabir Ellingsen, Pål-Tore Storli

In the Francis 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 Francis runners have recently experienced cracks in the runner blades due to fatigue, obviously due to the runner 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 3D NASTRAN 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 Timess, and for the simulations presented here the variations in Timess are found to be around 5 % 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.

<table>
<thead>
<tr>
<th>Author</th>
<th>Title</th>
<th>Keyword(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kabir Ellingsen, Pål-Tore Storli</td>
<td>Simulations of the Dynamic Load in a Francis Runner based on measurements of Grid Frequency Variations</td>
<td>Grid Frequency Variations, Torque Oscillations, 3D simulations, Francis turbine, Dynamic loading</td>
</tr>
</tbody>
</table>

### CFD-based Design and Analysis of the Ventilation of an Electric Generator Model, Validated with Experiments

Hamed Jamshidi, Håkan Nilsson, Valery Chernoyarov

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 air flow to the stator channels. The design is validated using geometric parameters of IS 150-125-125 centrifugal pump. The numerical results show that the blade profile lines has a 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 result 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.

<table>
<thead>
<tr>
<th>Author</th>
<th>Title</th>
<th>Keyword(s)</th>
</tr>
</thead>
</table>
The good performance of a vortex shedder is defined by strong and stable vortex generated under the condition of most favorably 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, Strouhal 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 reveals that the design with 0.20~0.267 blockage ratio and 0.10 split ratio as the best shedder for vortex flowmeter and these 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 database, 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

Comparative study of sediment erosion on alternative designs of Francis runner blade
Balath Rajandran, Dr. Hari P. Naseem, Braj S. Thapa
The aim of this study is 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.

Leakage Flow Influence on SHP pump model performances
Patrick Dupont, Annie-Claude Bayles-Laine, Antonio Dias, Gérard Bois, Olivier Rousette, Quiana Si
This paper deals with the influence of leakage flow existing in SHP pump model on the analysis of internal flow behaviour inside the vaned 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+ 10.06 code (RANS frozen and unsteady calculations). Some results were already presented at the 19th 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-hole 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.

124-125 Numerical Study of Important Factors for a Vortex Shedder using Automated Design Cycle
Su Myat Nyet, He Xu
The paper presents a CFD-based methodology for the prediction of guide vane flow in hydraulic turbine design. The tests were performed using three guide vanes 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 reveals that the design with 0.20~0.267 blockage ratio and 0.10 split ratio as the best shedder for vortex flowmeter and these 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 database, 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

122-124 CFD Analysis for Aligned and Misaligned Guide Vane Torque Prediction and Validation with Experimental Data
Christophe Devaux, Thi C. Vu, François Gubault
This paper presents a CFD-based methodology for the prediction of guide vane flow in hydraulic turbine design. The tests were performed using three guide vanes 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 reveals that the design with 0.20~0.267 blockage ratio and 0.10 split ratio as the best shedder for vortex flowmeter and these 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 database, 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

142-155 3D Casing-Distributor Analysis for Hydraulic Design Application
Christophe Devaux, Ying Zhang, Julien Dompierre, Thi C. Vu, Luca Menegatti, François Gubault
Nowadays, computational fluid dynamics is commonly used by design engineers to evaluate and compare losses in hydraulic components. It is less expensive and less time consuming than model tests. For that purpose, an automatic tool for casing and distributor design will be presented in this paper. An in-house mesh generator and a Reynolds Averaged Navier-Stokes equation solver using the standard k-ω shear stress transport (SST) turbulence model will be used to perform all computations. The +• OpenFOAM library will be used and compared to a commercial solver. The performances of the new fully coupled block solver developed by the University of Lucerne and Andritz will be compared to the standard 5.6t segregated simpalsolver and to a commercial solver. In this study, three 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.

155-168 Comparison of steady and unsteady simulation methodologies for predicting no-load speed in Francis Turbines
Hussein Hameiri, Christophe Roux, Bernd W. F. Li, J. J. Feng, H. Wu, J. L. Lu, W. L. Liao, X. Q. Luo, Dazin, Gérard Bois, Olivier Roussette, Qiaorui Si
Steady state analysis tool that can be integrated into their design process. drag coefficient, linearity of Strouhal number, K-factor, integration of GAMBIT, FLUENT, iSIGHT

169-182 Simulation model for Francis and Reversible Pump Turbines
Torbjørn K. Nielsen
This paper presents simulations of Francis turbine performance with good enough accuracy for the dynamic simulations. The only input to 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.

183-196 Hydrodynamic, Turbines, Characteristics, Simulation
Francis turbine, optimized design, rotating disc apparatus, runner blade, sediment erosion, wear pattern
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 furthermore help provide solutions to improve reliability. With such models, the effect of materials properties on runner reliability can be highlighted.

The 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 transient measurement different other signal characteristics are extracted from the individual signals and it's 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 scenarios: a sphere in a water flow in circular tube the HSV in Lucerne, a NACA profile in a cavitation tunnel and two Francis model turbines all at LMH in Lausanne. From the raw data several statistical parameters in the time and frequency domain as well as 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 of these 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.
Leakage Flow Influence on SHF pump model performances

Patrick Dupont, Antoine-Claude Bayed-Luacad, Antoine Doin, Gérard Bois, Olivier Roussette, Quinau Sir

model on the analysis of internal flow behaviour inside the vortex 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 impulse positions. The performance 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 + 15.6 code (RANS frozen and unsteady calculations). Some results were already presented at the French IAHR Symposium for three turbines for BANS模仿和URANS calculations. In the present paper, comparisons between URANS calculations with and without leakages and experimental results are presented and discussed for three flow rates. The performances of the diffuser obtained by numerical calculations are compared to those obtained by the three-hole 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.

Quantitative and qualitative analysis of the flow field development through T99 draft tube caused by optimized inlet velocity profiles

Sergio Galvan, 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 parametrized 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 parametrized inlet velocity profiles. Thus, this work develops a qualitative and quantitative analysis of these new draft tube flow field structures provided 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 recirculating flow area which used to promote an adverse pressure gradient in the core, deteriorating the pressure recovery effect. Thanks to the evolution of the 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 index 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.

Selection of Optimal Number of Francis Runner Blades for a Sediment Laden Micro Hydropower Plant in Nepal

Binaya Balad, Saleek Chitrakar, Rani Kharki, Hari Prasad Neupane

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 11 blades shows better performance at design as well as in variable discharge conditions. 96.2% efficiency is obtained from the runner with 11 blades at the design point, and the runners with 17 and 11 blades have 68.2% and 76.63% efficiencies respectively. Further, the runner with 12 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 05% more erosion tendency than in the runner with 17 blades.

Numerical simulation of slit wall effect on the Taylor vortices flow with radial temperature gradient

Dong Liu, Cheng-qing Chai, Fang-ning Zhu, Xi-qiang Han, Cheng Tang

Numerical simulation was applied to investigate the Taylor vortices 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 and velocity temperature in plane and 12-k radial location for different exit temperatures were compared, and the heat flux distributions along the inner cylinder wall at different working conditions were obtained. In the plane model, the average surface heat flux of inner cylinder increased with the inner cylinder rotation speed. In static 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.

Correction and Experimental Verification of Velocity Circulation in a Double-Blade Pump Impeller Outlet

Wang Ke, Liu Qiang

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 Star CFX formula. Weisser formula and Stechkin formula. Simultaneously, the internal flow of impeller for the double-blade pump was measured with PIV technology and average velocity circulations at 0.3, 1.0 and 2.0 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 Weisser 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 165%, 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.

Modeling and testing for hydraulic shock regarding a valve-less electro-hydraulic servo steering device for ships

Lian Jiet, He Lin, Xu Rongxue

A valve-less electro-hydraulic servo steering device (short: VSSD) for ships was shown as a study object. and its mathematical model of hydraulic shock was established on the basis of flow properties and force balance of each component. The influence of system 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 mathematical 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.
Genesis of Researches on Surges in Pumping Systems in Japan

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.

Impact performance for high frequency hydraulic rock drill drifter with sleeve valve

Guo Yang, Yu Jun Peng, Zhang Long Yan

A high frequency hydraulic rock drill drifter with sleeve valve is developed to use on an arm of excavator. In order to ensure optimal working parameters of system, the performance test is built using the arm and the hydraulic system. The evaluation indexes are gained by 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 75.6Hz. Optimal pressure of 1544 bar and drill drill is 12.8 MPa 11.68MPa, in which the energy efficiency reaches above 58.6%, and feature moment of energy distribution is more than 0.650.

Numerical Analysis of Damping Effect of Liquid Film on Material in High Speed Liquid Droplet Impingement

Hirotoshi Sasaki, Natsuy Nishii, Takuji Otsuka

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 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 wettable material surface is simulated. As a result, in the case of LDI on wet surface, maximum equivalent stress is less than those of dry surface due to damping effect of liquid film. Empirical formula of the damping effect is formulated with the fluid 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 75.6Hz. Optimal pressure of 1544 bar and drill drill is 12.8 MPa 11.68MPa, in which the energy efficiency reaches above 58.6%, and feature moment of energy distribution is more than 0.650.

Hydraulic Turbine, Runner, Collection Device, Airfoil, Performance, Flow Field

Hydraulic Turbine, Runner, Collection Device, Airfoil, Performance, Flow Field

Impact performance for high frequency hydraulic rock drill drifter with sleeve valve

A high frequency hydraulic rock drill drifter with sleeve valve is developed to use on an arm of excavator. In order to ensure optimal working parameters of system, the performance test is built using the arm and the hydraulic system. The evaluation indexes are gained by 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 75.6Hz. Optimal pressure of 1544 bar and drill drill is 12.8 MPa 11.68MPa, in which the energy efficiency reaches above 58.6%, and feature moment of energy distribution is more than 0.650.

Hydraulic Turbine, Runner, Collection Device, Airfoil, Performance, Flow Field

Numerical Analysis of Damping Effect of Liquid Film on Material in High Speed Liquid Droplet Impingement

Hirotoshi Sasaki, Natsuy Nishii, Takuji Otsuka

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 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 wettable material surface is simulated. As a result, in the case of LDI on wet surface, maximum equivalent stress is less than those of dry surface due to damping effect of liquid film. Empirical formula of the damping effect is formulated with the fluid 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 75.6Hz. Optimal pressure of 1544 bar and drill drill is 12.8 MPa 11.68MPa, in which the energy efficiency reaches above 58.6%, and feature moment of energy distribution is more than 0.650.
Numerical Simulation of Unsteady Cavitation in a High-speed Water Jet
Guo-yi Peng, Kunchiro Okada, Congmin Yang, Yasuyuki Oguma, Seiji Ishimaru

Concerning the numerical simulation of high-speed water jet with intense 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 jet issuing from a sheeted sharp edge orifice nozzle is treated when the cavitation number, σ = 0.1, and the computation results are 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 shear exit. The pattern of cavitation cloud shedding evaluated by simulation agrees with experimental one, and the possibility to capture the unsteady shedding of cavitation clouds is demonstrated. The decay of core velocity in cavitation jet is delayed greatly compared to that in no-cavitation jet, and the effect of the nozzle sheath is demonstrated.

Submerged water jet, cavitation, bubble dynamics, two-phase flow, computational fluid dynamics

Historical Perspective on Fluid Machinery Flow Optimization in an Industry
Akiro 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) technology” 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 optimisation, Multi-objective, Adjoint

Unsteady Wet Steam Flow Measurements in a Low Pressure Test Steam Turbine
Chongfei Duan, Koji Ishihashi, Daisuke Sendo, Ilia Bodas, Michel Lefebvre, Anestis T. Kalivas, 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 PDA-HT 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 currents, downstream from the individual rotational blades in the wet steam environment.

Steam turbine, Wet steam, Unsteady flow measurement, Low-pressure stage

Optimization of Blade Profile of a Plenum Fan
Shu Wu, Hua-Shou Dou, Yikan Wei, Yongqin Chen, Wenbin Cao, Cunlie Ying

A method of optimization design for the blade profile of a centrifugal impeller by controlling distribution velocity 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
107-118	Numerical And Experimental Study Of Single stage And Multistage Centrifugal Mixed Flow Submersible Borewell Pumps	C. Murugan, 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 simulation has been done for different turbulence models and grid sizes. In Omega SST model has been selected for the performance simulation and was validated with 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 inlet except the 1st stage impeller inlet.

velocity distribution, blade profile, static pressure, static pressure efficiency

119-128	Numerical Analysis of Unsteady Cavitation Flow around Balancing Drum of Multistage Pump	Milan Sedlak, Tomati Kratik, Patrick Zima

This work presents the numerical investigation of an unsteady cavitation flow around a balancing drum of a multistage pump. The main attention is focused on the cavitation phenomena, which occur in the real 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 cavitation 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 Steam Power Unit	Kuoeng 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 method and the performance 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 Group P possessed the highest power coefficient of 0.41 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, Hisashi TSUJI, 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 turbine at three operating conditions by applying the corresponding nozzle vane exit angles, which were set up in the experimental study. The inlet boundary condition. The numerical results revealed the characteristic flow behaviors at each operating condition.

Radial Turbine, Numerical Analysis, Turbocharger, Secondary Flow
A Study on the Multi-Objective Optimization of Impeller for High-Power Centrifugal Compressor

Hyun-Su Kang, Young-Hee Kim

In this study, a method for the multi-objective optimization of an impeller for a centrifugal impeller 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 three levels. A total of 23 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 design point was calculated by numerical simulation, and the optimal solution was chosen by analyzing the results of the DOE technique.

Centrifugal compressor, Shape optimization, Response surface method, Isentropic efficiency, Fluid-structure interaction

Parametric Optimization of Vortex Shedder based on Combination of Gambit, Fluent and iSIGHT

Su Myung, Hyung, Ye Xu, Honggang Yu

In this paper, a new framework that works the automatic execution with less design cycle time and human intervention bottleneck is introduced to optimize the vortex shedder design by numerical integration method. This framework is based on iSIGHT combined with the pre-processor GAMBIT, and flow analysis software FLUENT. Two vortex shedders, circular with slit and triangular with semi-circular, are employed as the design 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 20mm for circular cylinder with slit and 35 to 40 mm for semi-circular cylinder. For slit ratios, 0.1 and 0.15 are the optimized values for circular with slit and semi-cylinder, respectively. It is found that the optimal results obtained by DOE automated design cycle are in well agreement with the experiment.

Optimization, Vortex shedder, how to integrate the GAMBIT and FLUENT to iSIGHT

Drifted Flow of Droplet Impingement with Liquid Film on Material in High-Speed Liquid Droplet Impingement

Hiroshi Sasaki, Naoya Onishi, Yuka Iga

An error has been found in the footnote in Vol. 9 (2016) No. 1 (January-March) p. 57

Dynamic Response of Blade Surface Cavitation

Masakazu Tsuchi, Kimiya Sakaguchi, Kota Tsubouchi, Hitonori Horishita, Kansyuu 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 modeled 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

Performance and Flow Condition of Cross-Flow Wind Turbine with a Symmetrical Casing Having Side Boards

Toru Shirigentani, Junichiro Fukutomi, Masaki 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 $C_{p_{max}} = 0.17$ in the case with the casing and $C_{p_{max}} = 0.086$ 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

PIV Measurement of Inlet and Outlet Flow of a Contra-Rotating Small-Sized Cooling Fan

Toru Shirigentani, 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 400mm-square impeller were 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

Kaplan Turbine, Runner pressure measurement, Load variations, Rotating vortex rope formation, Rotating vortex rope mitigation

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 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. In this study, the effect of transient operations on the pressure fluctuations exerted on the runner and mechanism of the RVR formation/mitigation is studied. Draft tube and runner blades of the Pelton UR model, a Kaplan turbine, were equipped with pressure sensors for this purpose. The model was run in off-curve 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

Dynamic Characteristics of Floating Ring Seals in Rodlet Turbopumps

Yoshio Tokunaga, Hideyuki Inoue, Ken Hirano, Tomoaki Itoya, Yushiro Kuniti, Masaharu Uchiumi

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 rodlet turbopump applications, fundamental behavior and design philosophy have been revealed. However, further work is needed to explore the rotodynamic characteristics associated with rotor vibrations. In this study, rotodynamic forces for floating ring seal under rotor’s whirling motions are calculated to elucidate rotodynamic 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 rotodynamic 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.

Rotodynamic Characteristics, Floating ring seal, Dynamic characteristics, Rotodynamic coefficient
205-212 Efficiency Increase and Input Power Decrease of Converted Prototype Pump—Performance
Masao Odima

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.

213-221 CFD and surrogate-based inducer optimization
Tomáš Krátký, Lukáš Zawadí, Yih Ouababa

Due to the nature of curvature 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.

222-232 Hydraulic Model Test of a Floating Wave Energy Converter with a Cross-flow Turbine
Sangsoom Kim, Byunghi 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 designing a wave energy device that uses a cross-flow turbine (CT). In the preliminary design 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 in 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 meters.

239-249 Energetic analysis for optimization of a rotating triangular-tangential cooling channel with staggered square ribs
Mi-Ae Moon, Kang-Yong Kim

Energetic analysis was introduced in optimizing of a rotating equilateral triangular-tangential cooling channel with staggered square ribs to maximize the net energy gain. The objective function was defined as the net energy gain considering the energy gains by heat transfer and energy losses by friction and heat transfer process. The flow field and heat transfer in the channel were analyzed using three-dimensional Reynolds-averaged Navier-Stokes equations under the uniform temperature condition. Heat 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 22.5% 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.

253-244 Analysis on Characteristics of Pressure Fluctuation in Hydraulic Turbine with Guide Vane
Fangyi Shi, Jinrui Yang, Xiaohui Wang

An unsteady three-dimensional simulation based on Reynolds time-averaged Navier-Stokes equations and RNG k-ω turbulence model was presented for pump-turbine, the pressure fluctuation characteristic of hydraulic turbine with guide vane was obtained. The results show that the time domains of pressure fluctuations in the channel 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.

245-252 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 (AFL) had demonstrated the unique features in the past. One of the most well-known inventors was the hydraulic machine gun simulator, which had become the standard equipment of airplane during World War I. The studies on the AFL between 1960 and 1980 had triggered many researchers' interests and reached the summit. The disadvantages of the AFL like low efficiency and cooling difficulty had prevented the further development. Few people are engaged in studying the AFL at present. However, the unique characteristics of the AFL inspire the researchers to continuously explore the new special suitable applications. The overview of the AFL and the new potential application in the geotechnical testing field have been discussed in this paper. First, the research results of the AFL in the past have been summarized. Then, the classifications of the AFL 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 AFL 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 Tariel instrument have been given, which will be more innovative 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 Tariel instrument is feasible. The paper work will also give some inspirations in the corresponding motion control system.

256-264 Design and performance research of a mixed-flow submersible deep well pump
Qinh Zhang, Yuehui Xu, Li Cao, Wading Shi, Wengang 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 environmentally-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.
Energy Efficient Design of a Jet Pump by Ensemble of Surrogates and Evolutionary Approach

Akhil Hauran, Anshul Sonawat, Sarath Mohan, Abdul 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 optimisation via numerical modeling for fluid dynamics and implementation of an evolutionary algorithm for the optimisation 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 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.

Page 103-109

A Twin Impulse Turbine for Wave Energy Conversion -The Performance under Unsteady Airflow-

M M Ahsanul Alam, Rabbi Sait, Manabu Takas, Shinya Ochiara, Toshiaki Setoguchi

Turbocompounding is a key technology to satisfy the future requirements of diesel engines' fuel economy and emission reduction. A turbocompound diesel engine was developed based on a conventional 11-Liter heavy-duty diesel engine's fuel economy and emission reduction. A turbocompound diesel engine is considered to be a key technology to satisfy the future requirements of heavy-duty diesel engines. The performance of a turbocompound engine under unsteady flow conditions was 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. 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 effect of a turbocompound 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. The present study reported the use of special shaped blade to reduce the difference in pressure across the Wells turbine for wave energy conversion. The blade profile was constructed by using the CAD tool and the blade was fabricated and the trailing edge was shaped 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. Moreover, it is verified that airflow goes backward in the reverse turbine in low flow rates.

Page 107-112

Wells Turbine for Wave Energy Conversion -Effect of Trailing Edge Shape-

Katuya Takeda, Tomohiro Sunahara, Manabu Takas, M M Ahsanul Alam, Toshiaki Setoguchi

The present study reported the use of special shaped blade to reduce the difference in pressure across the Wells turbine for wave energy conversion. The blade profile was constructed by using the CAD tool and the blade was fabricated and the trailing edge was shaped 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. The present study investigated 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. Moreover, it is verified that airflow goes backward in the reverse turbine in low flow rates.

Page 113-124

Centrifugal Impeller Blade Shape Optimization Through Numerical Modeling

Sayed Ahmad Jaffar Belalay, Abdul 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 angle 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 point. 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.

Page 125-131

Free Surface Vortex in a Rotating Barrel with Rods of Different Heights

Zhang Xueyue, Zhang Min, Chen Wanyu, Yang Fan, Guo Xueyan

A barthrot vortex above the outlet of a rotating barrel is simulated. By analyzing the Elman layer theory, it can be found that the main flow circulation is inversely proportional to the thickness of Elman layer. The thicker the Elman boundary layer, the weaker the rotational strength and the shorter the length of gas core is. According to this law, models of barrels with rods of different heights are established. The reduction of air-core length in this air circulation vortex and weakening the strength of rotation field were achieved.

Page 132-137

Experimental study on the performance of a turbocompound diesel engine with variable geometry turbocharger

Yong Yin, Zhengzhi Liu, Weilin Zhuge, Rongchao Zhao, Yongting 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 blade specific fuel consumption (BSFC) are 3% and 8% respectively.
A Comparison of Surge Behaviors in Multi-Stage and Single-Stage Axial Flow Compressors

Nobuyuki Yamaguchi

The surge behaviors in multi-stage and single-stage axial flow compressors are compared in terms of the subsonic and supersonic behaviors, with particular attention to the qualitative and quantitative differences. The subsonic surge is identified as the region where the flow is choked, while the supersonic surge is characterized by the occurrence of shock waves. The results show that the single-stage compressor has a higher surge margin compared to the multi-stage compressor, which is attributed to the larger blade spacing and lower pressure ratio in the single-stage design. The research also highlights the importance of incorporating robust surge suppression systems in multi-stage compressors to ensure reliable operation.

Performance Study of Throat Control Unit with the Various Geometric Shapes

Kyunghyun Kim, Jong Ho Park

This study aims to identify aerodynamic characteristics of the throat, a mechanical deflector, by conducting non-experimental study using compressors and supersonic flow test equipment. With the throat, the flow is divided into two sections: the throat section and the expansion section. The results show that the throat section is a critical part of the flow control system, and its design significantly affects the performance of the compressor. The study also suggests that the optimal design parameters for the throat section can improve the efficiency and stability of the compressor.

Optimal Design of Two-Dimensional Cascade with Shock-Free Inflow Criterion

Abdel Muaq, Priyono Sutikno, Aryadi Soewono, Firman Ravi Koirala, Sailesh Chitrakar, Hari Prasad Neopane, Young-Seok Choi, Kwang-Yong Kim, Jin-Hyuk Kim, Makoto Kawanishi, Takahiro Sumi, Shigeo Yoshida Joon-Hyung Kim, Bo-Min Cho, Youn-Sung Kim, Balendra Chhetri, Bhola Thapa Wenbin Cao, Cunlie Ying Lotfi Bouazizi, Saïd Turki

As a single-channel pump is used for wastewater treatment, the particular pump type can prevent performance reduction or damage caused by foreign substances. However, the design methods for single-channel pumps differ and are more difficult than those for general pumps. In this study, a design optimization method to improve the hydraulic 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 result, the improved impeller design showed improved flow characteristics, such as reduced flow-induced noise and vibration, and increased efficiency and stability. The optimized impeller can be applied to various wastewater treatment systems, improving their performance and sustainability.

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 functions 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, 0.5 and 1% respectively, for a fixed Reynolds number of Re = 500. 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.

Computational Design of Bifurcation: A Case Study of Daundari Khola-Hydropower Project

Rezai Koriia, Salehe Chitrakar, Hari Prasad Neopane Bakendra Chitrakar, Brata Thapa

Bifurcation refers to a division of a fluid stream into two distinct flow paths. This study focuses on the computational design of bifurcations for hydropower projects. The research adopts a computational fluid dynamics (CFD) approach to analyze the fluid flow characteristics and optimize the design of bifurcation. The results show that the optimal bifurcation design can enhance the energy recovery efficiency and improve the structural integrity of the hydropower project. The study contributes to the development of more sustainable and efficient hydropower systems.

Employing rotating vanes to enhance the performance of a centrifugal fan

Hua-Shu Chen, Lin We, Yikun Wei, Yongxing Chen, Wenzhen Cao, Cuiying Yang

Numerical simulation is carried out for flow characteristics in a centrifugal fan and the influence of the diameter ratio of the rotating vanes on the performance of the centrifugal fan is analyzed. The results show that the optimal diameter ratio of the rotating vanes is 1.2 at rated and low flow coefficient.

Experimental and Numerical Studies on the Possibility of Duct Flow Low-power Generation Using a Butterfly Wind Turbine

Yukita Naka, Shokou Kogo, Keisuke Tabagaki, Makoto Kawashima, Takahiro Sunie, Shigeyoshi Yada

An objective of this study is to demonstrate the viability 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 vertical axis type (D = 0.4 m) was used. The output performance is measured at various locations relative to the exit of a small wind tunnel. The performance measured 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 1.2% is obtained.

Vertical Axis Wind Turbine, Duct Flow Power Generation, Wind Energy, Butterfly Wind Turbine, Computational Fluid Dynamics

Plenum fan, rotating vanes, diffusion, static pressure, efficiency

9-18

19-29
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

In the current study, identifying regimes and behaviors of the various viscous fluids in a typical horizontal single-stage 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 by enhancing 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 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 have been obtained in viscosity 1 cSt which was by 12.2% and 44.6% 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 viscosity 1 cSt values.

Thrust Characteristics and Nozzle Role of Water Jet Propulsion

Vongvan N. Warner Liu, Zhuanhe Shen, Xiwen Fan

Butterfly valve, Life test, Real time monitoring system, Rated pressure, Leakage, Valve seat, Disc

Keywords

A Design Method for Cascades Consisting of Circular Arc Blades with Constant Thickness

Tao Ban, Qiangping Han, Martin Böhlé

In the current study, identifying regimes and behaviors of the various viscous fluids in a typical horizontal single-stage 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 by enhancing 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 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 have been obtained in viscosity 1 cSt which was by 12.2% and 44.6% 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 viscosity 1 cSt values.

Performance Analysis of a Combined Blade Savonius Wind Turbines

Arifin Sanui, Sudjipto Soepaman, Slamet Widyusah, Lilis Yuliari

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. After Saint-Venant Release 14.3, the then 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 overfly 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.

Effect of Deflected inflow on flows in a strongly curved 90 degree elbow

Yashu Kawamoto, Ryo Kusakabe, Mitsuoki Sogo, Toshiyuki Yasuda, Hidehiko Yamano, Masaoi Tanaka

Numerical Simulation on the Performance of Axial Vane Type Gas-Liquid Separator with Different Guide Vane Structure

Yang Fan, Liu Allan, 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 has Rankine vortex characteristics, pressure distribution exhibits a high similarity which value become 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.
Sediment monitoring for hydro-abrasive erosion: A field study from Himalayas, India

A multi-frequency acoustic instrument was installed at a declining 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 microscopy, 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 = 0.96) 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 11,500 ton. This study shall be useful for researchers and hydropower managers in measuring/monitoring sediment for hydro-abrasive erosion in hydropower plants.

The Effect of Different Inflows on the Unsteady Hydrodynamic Characteristics of a Mixed Flow Pump

Long Yan, Wang Dusheng, Yin Jianjun, Cai Youlin, Feng Chao

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 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 declining 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 microscopy, 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 = 0.96) 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 11,500 ton. This study shall be useful for researchers and hydropower managers in measuring/monitoring sediment for hydro-abrasive erosion in hydropower plants.
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 design 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. from 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 vane 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 mode rotating and plunging of the rotating vortex rope. The time of appearance and disappearance of rotating and plunging modes of vortex rope were 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.

Numerical Simulation and Experimental Research of the Flow Coefficient of the Nozzle-Flapper Valve Considering Cavitation

Lei Li, Changyou Li, Hengquan Zhang

The nozzle-flapper valves are widely applied as a pilot stage in aerospace and military systems. 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 used to predict flow coefficient variation and cavitation under different conditions with varying gap width and inlet pressure. Moreover, a new test device has been designed and manufactured to measure the flow coefficient and for visualized cavitation. The numerical simulation and test results both indicate the cavitation intensity gets into a critical and strong level, firstly, as the gap width varies from small to large, from the curve, the flow coefficient increases, the pressure drop increases, especially as the gap increases; then, as the gap increases, the flow coefficient decreases, the pressure drop increases, especially as the gap increases. The experimental results are useful to determine the optimal null clearance for a specific cavitation level. The numerical results are also validated with the experiments conducted with the proposed valve design. The numerical simulation results are also compared with the experiments for different gap widths. The numerical simulation results are in good agreement with the experimental results for different gap widths and different gap directions. The numerical simulation results are also compared with the experiments for different gap widths and different gap directions. The numerical simulation results are also compared with the experiments for different gap widths and different gap directions. The numerical simulation results are also compared with the experiments for different gap widths and different gap directions.
240-251
Effects of Latin hypercube sampling on surrogate modeling and optimization
Avrind Aftab, Eunyoung Kim, Jan-woon 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 four types of problems: (1) thermal-fluid design problems, (2) optimization of convergent-divergent microcooler coupled with pulsed flow and boost-shaped riser, and (2) analytical test functions (sin-pump, 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-plot. The surrogate models were assessed 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.

254-260
Numerical Cavitation Intensity on a Hydrofoil for 3D Homogeneous Unsteady Viscous Flows
Christophe Lecce, Antoine Archer, Rajeena Forte, Patella, Fabien Carru

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 in both terms of modeling the cavitation flow and predicting the erosion due to cavitation. For this purpose, a numerical methodology was proposed to estimate cavitation intensity from 3D steady/varying flow simulations. CFD calculations were carried out using Code_Saturne, which enables u-RANS equations resolution for a homogeneous fluid mixture using the Morkel’s model. A coupled a-turbulence model with the Reynolds correction. A post-process cavitation intensity prediction model was developed based on pressure and cavitating derivatives. This model is applied on a flow around a hydrofoil using different structural (pump velocities) and numerical (meshes, time steps) parameters. This 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 jetting test.

264-273
Annual Energy Production Maximization for Tidal Power Plants with Evolutionary Algorithms
Evgenia Kontoleontos, 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 sluice mode) and provide the optimal plant operation that maximizes the AEP to the customer. 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.

287-295
Pressure Pulsation Characteristics of a Model Pump-turbine Operating in the S-shaped Region: CFD Simulations
Linsheng Xia, Yongguang Cheng, Yang 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 mid/low side 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 swanzę section. Under the condition of RFVS at the mid-span, the smaller flow rate make the smaller difference of pressure pulsation amplitudes in the swanzę direction. Moreover, the rotating stall, rotating at 35%-62.5% of the runner rotational frequency, make the low frequency component of pressure pulsations distribute unevenly along the circumference in the variable space. However, it have little influence on the distributions of high components.

How to Avoid Severe Incidents at Hydropower Plants
Masahiro Yasuda, Satoshi Watanabe

Hydropower plant, severe incident, flood, fire, turbine failure, generator failure

297-305
Measurement of Dynamic Characteristics of an Inducer in Castable Conditions
Takuya Ashida, Keita Yamamoto, Koschi Yonezawa, Hironori Horie, Yutaka Kawaia, Yoshinobu Togimoto

In liquid-propellant rockets, POGO instability can occur, in which a fluctuation correlates the upstream and downstream pressure and flow rate fluctuations. In liquid-propellant rockets, a thrust fluctuation, and a structural failure, generator failure can caused by the inertia of the flow rate fluctuation. 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 mid/low side 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 swanzę section. Under the condition of RFVS at the mid-span, the smaller flow rate make the smaller difference of pressure pulsation amplitudes in the swanzę direction. Moreover, the rotating stall, rotating at 35%-62.5% of the runner rotational frequency, make the low frequency component of pressure pulsations distribute unevenly along the circumference in the variable space. However, it have little influence on the distributions of high components.

297-306
How to Avoid Severe Incidents at Hydropower Plants
Masahiro Yasuda, Satoshi Watanabe

In liquid-propellant rockets, POGO instability can occur, in which a fluctuation correlates the upstream and downstream pressure and flow rate fluctuations. It is caused by the inertia of the flow rate fluctuation. 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 mid/low side 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 swanzę direction. Under the condition of RFVS at the mid-span, the smaller flow rate make the smaller difference of pressure pulsation amplitudes in the swanzę direction. Moreover, the rotating stall, rotating at 35%-62.5% of the runner rotational frequency, make the low frequency component of pressure pulsations distribute unevenly along the circumference in the variable space. However, it have little influence on the distributions of high components.

317-327
Study on Influence of Blade Number on Aerodynamic Noise of Half-ducted Propeller Fans for Packaged Air-conditioners
Taku Iwase, Tetsushi Kishitani, Masato Furukawa

Half-ducted propeller fans, Tip vortex, Aerodynamic noise, Large eddy simulation

Keywords
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.

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 ozone depletion. Since the existing chlorofluorocarbon (CFC) and hydrochlorofluorocarbon (HCFC) 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(s)? The present work makes an effort to answer this question.

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 different locations around the volute under different flow rates conditions ranging from 30% 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 intensit broadside 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 vortex in the flow passages of impeller.

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.

The miniaturization and weight equipments and household appliances. Electric motors are used as the main power sources in many industrial systems work efficiently when the existing refrigerant is replaced by a viable environment friendly substitute(s). One such question is: Will the existing systems work efficiently when the existing refrigerant is replaced by a viable environment friendly substitute(s)? The present work makes an effort to answer this question. Fluid power transmission and control, servo valve, temperature drift characteristics, electrohydraulic control equipment.

The miniaturization and weight equipments and household appliances. Electric motors are used as the main power sources in many industrial systems work efficiently when the existing refrigerant is replaced by a viable environment friendly substitute(s). One such question is: Will the existing systems work efficiently when the existing refrigerant is replaced by a viable environment friendly substitute(s)? The present work makes an effort to answer this question. Fluid power transmission and control, servo valve, temperature drift characteristics, electrohydraulic control equipment.

Small hydro-turbine, Contra-rotating rotors, Internal flow instability, Centrifugal pump, Crystallization phenomenon, CFD-DEM, Particle feature and behavior.


Centrifugal pump, crystallization phenomenon, CFD-DEM, particle feature and behavior.

Small hydro-turbine, Contra-rotating rotors, Internal flow, Performance, Flow visualization.

Small hydro-turbine, Contra-rotating rotors, Internal flow, Performance, Flow visualization.
Effect of Solid Particles on Cavitation and Lubrication Characteristics of Upstream Pumping Mechanical Seal Liquid Membrane

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 turbulent 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 suction, opening force and friction torque of the film under different working conditions. The results showed that the particles have an inhibition 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 particle volume fraction, and the volume fraction increased in 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. The friction torque did not change drastically 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 the solid particles were added. All of these operators have been applied to turbomachinery configurations and the advantages and disadvantages are discussed.

Unstructured Grid Smoothing for Turbomachinery Applications

In the present study, two mesh smoothing techniques, Laplace and Winslow, smoothing techniques, for unstructured grids on turbomachinery application were investigated. These operators are based on the solution of elliptic or hyperbolic equations. In the first case, Laplace’s equations are solved using a barotropic 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.

Unsteady Flow Condition of Centrifugal Pump for Low Viscous Fluid Food

For fluid machinery in the food industry, fluid food has been used in wide variety of field such as transportation, filling, and improvement of quality of fluid food. The flow condition of these fluid machinery is quite complicated because the fluid food is different from water. Therefore, a detailed design method on the internal fluid is necessary to conduct this research. In this study, a pump with small number of blades was used to decrease the size and keep wide fluid process. 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 fluid condition, the unsteady numerical analysis using low viscous fluid was conducted. In this paper, the relationship between the blade geometry and the performance was investigated. In that case, the internal fluid condition of the centrifugal pump using low viscous fluid was clarified by the numerical analysis results.

Numerical simulations of vortex-induced vibration of a three-dimensional flexible cylinder under uniform turbulent flow are calculated when Reynolds number is 3.5×10^{5}. 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 Newton method. The fluid-Cylinder interaction is realized by utilizing the diffusion-based smooth dynamic mesh method. Considering MRF system, the system tends of lift coefficient, drag coefficient, total force, and vortex shedding frequency of the cylinder. Delay angle of the cylinder are analyzed under different frequency ratios. The nonlinear phenomenon of locked-in, phase-switch is 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 sudden 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.

Surface Roughness, Microchannel, Heat Transfer, Vortex-induced, Dimensional, Finite Element, Lift Coefficient

Keywords

Dynamic evolutions between the draft tube pressure pulsations and vortex ropes of a Francis turbine during runaway

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 conveyor system and a three-dimensional model of the Francis turbine. The results show that the annulus-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 annulus-like pressure pulsations.

Unstructured Grid Smoothing for Turbomachinery Applications

In the present study, two mesh smoothing techniques, Laplace and Winslow, smoothing techniques, for unstructured grids on turbomachinery application were investigated. These operators are based on the solution of elliptic or hyperbolic equations. In the first case, Laplace’s equations are solved using a barotropic 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.

Unsteady Flow Condition of Centrifugal Pump for Low Viscous Fluid Food

For fluid machinery in the food industry, fluid food has been used in wide variety of field such as transportation, filling, and improvement of quality of fluid food. The flow condition of these fluid machinery is quite complicated because the fluid food is different from water. Therefore, a detailed design method on the internal fluid is necessary to conduct this research. In this study, a pump with small number of blades was used to decrease the size and keep wide fluid process. 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 fluid condition, the unsteady numerical analysis using low viscous fluid was conducted. In this paper, the relationship between the blade geometry and the performance was investigated. In that case, the internal fluid condition of the centrifugal pump using low viscous fluid was clarified by the numerical analysis results.

Numerical simulations of vortex-induced vibration of a three-dimensional flexible cylinder under uniform turbulent flow are calculated when Reynolds number is 3.5×10^{5}. 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 Newton method. The fluid-Cylinder interaction is realized by utilizing the diffusion-based smooth dynamic mesh method. Considering MRF system, the system tends of lift coefficient, drag coefficient, total force, and vortex shedding frequency of the cylinder. Delay angle of the cylinder are analyzed under different frequency ratios. The nonlinear phenomenon of locked-in, phase-switch is 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 sudden 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.

Unsteady Flow Condition of Centrifugal Pump for Low Viscous Fluid Food

For fluid machinery in the food industry, fluid food has been used in wide variety of field such as transportation, filling, and improvement of quality of fluid food. The flow condition of these fluid machinery is quite complicated because the fluid food is different from water. Therefore, a detailed design method on the internal fluid is necessary to conduct this research. In this study, a pump with small number of blades was used to decrease the size and keep wide fluid process. 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 fluid condition, the unsteady numerical analysis using low viscous fluid was conducted. In this paper, the relationship between the blade geometry and the performance was investigated. In that case, the internal fluid condition of the centrifugal pump using low viscous fluid was clarified by the numerical analysis results.
Non-equilibrium wet-steam model, Rotating machinery, Steam condensing flow

Effective bulk modulus, Pseudo-excitation, Condition monitoring, Frequency domains, Bubbles volume fraction

On-line Measuring Method of Effective Bulk Modulus in Hydraulic System Based on Frequency Analysis

Yinghu Chen, Nan Wang, Liuchen Gu

Flow of steam is different from other gas flows and involves droplet generation in the flow field. This phase transition affects not only flow aspects, but also machine performance in a negative way. Thus, CFD is widely used in machine design and optimization processes, so phase transition phenomena in steam flows should be considered in CFD to predict the internal flows precisely. In the past, non-equilibrium wet-steam model was implemented in house code T-Flow and it was applied to steady calculations of a steam turbine model with changing state/rotor interface. The results showed that mixing plane method is not appropriate to simulate steam condensing flow and frozen rotor method can be affected by relative position between a stator and a rotor. Therefore, steady wet-steam flows with non-equilibrium phase-transition were simulated for a steam turbine model in this study with 4 different stator-rotor positions and their effects on the wet-steam flow fields were investigated in detail.

On-line Measuring Method of Effective Bulk Modulus in Hydraulic System Based on Frequency Analysis

Based on the gas-liquid phase flow theory, an indirect method is proposed to on-line measure the effective bulk modulus by constructing the mathematical model which reveals the relationships of the effective bulk modulus, bubble volume fraction, pressure and natural frequency. The natural frequency of hydraulic fluid is a key parameter for measuring effective bulk modulus utilizing the model, so it is online measured by the proposed pseudo-excitation method. The frequency response function is obtained by solving the amplitude spectrum functions of pseudo-excitation signal and the response signal, and the natural frequency is obtained. The numerical simulation and experiment are carried out, and we can deduce from the analysis of simulation and experimental results that the effective bulk modulus in hydraulic system can be easily online measured by the proposed indirect method.

Numerical Analysis of Contra-Rotating Small Hydro Turbine with Cylinder Spoke

Ding Nan, Toru Shigemitsu, Shengyan Zhao, Yasutomo Takehisa

Small hydro-turbine is one of potential energy resource existing in our living environment and yet not been widely used. Small hydro-turbine is suitable in these energy resources while it still has some shortages, like performing low efficiency and easy to be out of control when there are foreign materials in the water. Thus we designed this contra-rotating small hydro-turbine to increase its efficiency and enhance its ability of keeping stable operation. The experimental apparatus was designed and assembled, and some experiments were conducted. A new kind of cylinder spoke which supports the front and rear rotor was adopted and the numerical analysis was carried out in this paper. As the numerical analysis results shown, the efficiency of test turbine with cylinder spoke increased in a wide range of flow rate. The maximum efficiency increased about 2.2%, reaching to 66.4%, and it was obtained at 1.2Qn. The internal flow condition, especially at the areas behind the spoke, was improved by the cylinder spoke.

Simulation Analysis of Cylinder Impact Performance Influenced by Shaft Sleeve Structure

Cong Liu, Pei Xiao, Yong Li, Gangnan Li

The numerical simulations are performed on the centrifugal pump impeller without splitter blades. To improve the design of the split blades in the centrifugal pump impeller, flow domain simulations are performed on the centrifugal pump impeller with two types of blades. Firstly, inner flow simulations are performed based on the theory of the boundary vorticity dynamics, and distributions of the boundary vorticity flux (BVF), friction force as well as vorticity on the inner walls of the impeller are carefully analyzed to find the location of bad flows and their dynamic sources. Later, according to the inner flow diagnosis results, splitter blades are designed and refined for the original impeller. The inner flow field in the impeller equipped with the preliminary splitter blade and reference blade in the centrifugal pump impeller are numerically simulated and compared. Finally, comparisons are made among the three impellers, and it is found that, compared to the original impeller, the BVF, friction force and vorticity distribution in the impeller equipped with the preliminary splitter blade are lower than that in the reference blade. The distribution of mean swirl also decreases. The friction force is smaller, the hydraulic force acting on the impeller increases, with the pump head and efficiency increasing simultaneously. It is proven that the diagnosis based on the theory of the boundary vorticity dynamics is effective.

Design Improvement of the Splitter Blade in the Centrifugal Pump Impeller Based on Theory of Boundary Vorticity Dynamics

Brijar ZHAO, Zhengyu ZHANG, Youhao ZHAO, Yuning FU, Qi LIU, Huilong CHEN

It improves the design of the split blades in the centrifugal pump impeller. Inner flow simulations are performed for the centrifugal pump impeller without split blades. Firstly, inner flow diagrams are performed based on the theory of the boundary vorticity dynamics, and distributions of the boundary vorticity flux (BVF), friction force as well as vorticity on the inner walls of the impeller are carefully analyzed to find the location of bad flows and their dynamic sources. Later, according to the inner flow diagnosis results, splitter blades are designed and refined for the original impeller. The inner flow field in the impeller equipped with the preliminary splitter blade and reference blade in the centrifugal pump impeller are numerically simulated and compared. Finally, comparisons are made among the three impellers, and it is found that, compared to the original impeller, the BVF, friction force and vorticity distribution in the impeller equipped with the preliminary splitter blade are lower than that in the reference blade. The distribution of mean swirl also decreases. The friction force is smaller, the hydraulic force acting on the impeller increases, with the pump head and efficiency increasing simultaneously. It is proven that the diagnosis based on the theory of the boundary vorticity dynamics is effective.

Vol. 11, No. 1, January-March 2018

Effect of Vortex Design on a Half-Ducted Axial Flow Fan [In the case of forced vortex design]

Ashok Kaj, Yoshio Kinose, Norimasa Shimbo, Yoshishige Setoguchi

An improvement of the half-ducted axial flow fan design by applying the forced vortex design to the rotor blade was conducted by CFD. The quasi three-dimensional flow theory was applied for the half-ducted fan design, and the meridional flow was calculated by the method of streamline curvature. When the radial balance equations from hub to tip was solved, the vortex design was changed from the constant swirl velocity design to the forced vortex design in reference to our previous experimental results of the half-ducted fan's internal flow. The analysis of the three-dimensional internal flow fields of the half-ducted fan designed by adopting the forced vortex design was conducted. As a result of CFD, the efficiency of the fan was about 1-3% larger at the flow rate coefficient Q/0.3Qn=1.1 than the constant swirl velocity design.

Wave Energy Harvesting Turbine: Effect of Hub-to-Tip Profile Modification

Madhan Kumar, Panchar Haldar, Abdul Samed, Shih Hyung Hee

The present paper investigates the leading edge (LE) undulations of a Wells turbine blade through numerical analysis. The aspiration for this modification came from Humphreys' work, which have unproven prototuses at the LE of their turbulent models. The tip blades help to move away from stagnation area. The CFD results are used for implementation in house code T-Flow and it was applied to steady calculations of a Wells turbine model with changing state/rotor interface. The results showed mixing plane method is not appropriate to simulate steam condensing flow and frozen rotor method can be affected by relative position between a stator and a rotor. Therefore, steady wet-steam flows with non-equilibrium phase-transition were simulated for a steam turbine model in this study with 4 different stator-rotor positions and their effects on the wet-steam flow fields were investigated in detail.

Review of Fluid Structure Interaction Methods Application to Floating Wave Energy Converter

Mohammed Asad Zulfiqar, Young-Ho Lee

Computational fluid dynamics (CFD) is a highly efficient paradigm that is used extensively in marine renewable energy research studies and commercial applications. The CFD paradigm is ideal for simulating the complex dynamics of Fluid-Structure Interactions (FSI) and can capture all kinds of nonlinear fluid motions. While nonlinear simulations are considered more expensive and resource intensive compared to the frequency domain approaches, they are much more accurate and ideal for commercial applications. This review study presents a comprehensive overview of the computation fluid dynamics paradigm in context of wave energy converter (WEC) and highlights different CFD tools that are available today for commercial and research applications. State-of-the-art CFD codes such as ANSYS CFX that are highly ideal for WEC simulation problems are highlighted and aspects such as time and frequency domains are also thoroughly discussed along with efficiency of the nonlinear simulations compared to the linear models. The paper presents a comparative evaluation of different WEC modeling codes available today and illustrates the code framework of different CFD simulation software suites.
Experimental Analysis of Diffuser-Rotating Stall in a Three-Stage Centrifugal Pump
Takaki Takamine, Osachi Furukawa, Satoshi Watanabe, Hisanori Watanabe, Kazuyoshi Miyagawa

Numerical Investigation of the Turbulent Cavitating Flow over Submerged Bodies
Hong Feng, Qiao Zhenyun, Liu Li, Yuan Jiangping

Comparison of Loss Models for Performance Prediction of Radial Inflow Turbine
Seung Kuli Cho, Jeong Il Lee, Jeong Il Lee

Effect of Impulse Turbine Geometry on the Performance of a Dual-Turbine System for Wave Energy Conversion
Manabu Takas, Haruka Katsubu, M. M. A. Alam, Senadie Furama, Shinya Okubara, Yoshih Kiyonoe

Effect of Different Guide Vane Numbers on Transients Radial Force of Pump as Turbine
Ngoc Thi, Juntal Yang, Xiaohui Wang

Design of Mixed-flow pump for NAMS Based on Optimum Design Database
Lung Kim, Kyoung Yang Lee, Ah-Hyuk Kim, Seon-Yong Yoon, Young-Seok Choi

Numerical Simulation and Experimental Research about Shock Wave in 1+1/2 Counter-Rotating Turbine
Chao Li, Huaing Liu

Keywords
- Submerged bodies, multiphase approach, cavitating flow, turbulence, numerical simulation
- Radial inflow turbine, Preliminary design, Performance estimation, Loss models, 1D mean stream line analysis
- Fluid Machinery, Impulse Turbine, Wave Energy Conversion, Wells Turbine
- Hydrodynamic design system, Mixed-flow Pump, Impeller, Diffuser, Design optimisation

Page 77-84
Page 85-98
Page 97-109
Page 109-120
Page 115-122
Page 125-128
Page 129-138

Abstract
Rotating stall phenomenon limits the operation range of turbomachines, therefore it is important to understand the crucial parameters of this phenomenon. In the present study, the diffuser rotating stall in a three-stage centrifugal pump was experimentally studied. Examinined main parameter was an axial offset of rotor against the stationary part, which might be unavoidable due to accumulation of geometrical tolerances and assembling errors. The effect of leakage flow rate at the balance drum section employed as a thrust balancing device, which increases the thruflow rate at the first and second stage diffusers, was also studied. The effect of rotor axial offset was clearly observed and, with the rotor axial offset to the suction side, the rotating stall appeared only at the third stage diffuser. By setting the balance flow rate set to zero, the onset range of rotating stall became wider in the first and second stage diffusers, which was well explained by the decrease of the thruflow rate.

A numerical method for the calculation of turbulent cavitating flow over submerged objects is proposed in present work. Cavitation is modeled via a single-fluid cavitatoin model which is derived based on a truncated form of the Rayleigh-Plesset equation and the mixture multiphase theory. The approach has been implemented by user-deine function which is widely used in ANSYS FLUENT. Detailed results are presented for sheet cavitational over a submerged hemispherical object in a wide range of cavitation numbers and the cloud cavitation around a Clark-Y hydrofoil. In particular, for the submerged body, we compared the surface pressure distribution with experimental data which was available in literature. Later the cloud cavitation structure and its effect on the forces of the hydrofoil were studied. The comparisons between the simulating and experimental results show that present numerical approach has good capability to predict the surface pressure coefficient and the pulsation frequency at cavitation number 0.15, 0.45 and 0.85 of the hemispherical body under cavitation conditions. Meanwhile, for the hydrofoil, the proposed approach is sufficiently robust to predict the characteristics of the time-averaged lift and drag coefficients and the evolution of the cloud cavity with time.

This paper is aimed for suggesting the best combination of loss models for a radial inflow turbine among various empirical models from the open literature. The 10 mean streamline analysis procedure is utilized to confirm the suitability of loss models and the recommended combinations. The performance of loss models are evaluated by comparing to the well-documented NASA turbine test data. To suggest a more acceptable combination of loss models, each loss model is analyzed and assessed in accordance with the present loss mechanisms. This study especially focused on the rotor passage loss and the tip clearance loss, and the tip clearance loss model modification was suggested. The agreement of the newly suggested loss model combination with test data is significantly improved in comparison to the existing combinations. The design procedure presented in this paper can be used as a 1D tool of a radial inflow turbine for the preliminary design phase.

This paper is aimed for suggesting the best combination of loss models for a radial inflow turbine among various empirical models from the open literature. The 10 mean streamline analysis procedure is utilized to confirm the suitability of loss models and the recommended combinations. The performance of loss models are evaluated by comparing to the well-documented NASA turbine test data. To suggest a more acceptable combination of loss models, each loss model is analyzed and assessed in accordance with the present loss mechanisms. This study especially focused on the rotor passage loss and the tip clearance loss, and the tip clearance loss model modification was suggested. The agreement of the newly suggested loss model combination with test data is significantly improved in comparison to the existing combinations. The design procedure presented in this paper can be used as a 1D tool of a radial inflow turbine for the preliminary design phase.

In this study, a mixed-flow pump with a specified specific speed was optimally designed by utilizing computational fluid dynamics (CFD). Additional, the shape of the mixed-flow pump and flow characteristics depending on the specific speed were studied, in order to development of hydrodynamic design system for mixed-flow pump. A mixed-flow pump consists of an impeller and a diffuser and the shapes of mixed-flow pumps show a certain tendency depending on specific speed. With an analysis of the shapes of a mixed-flow pump, the optimal shape to satisfy design specifications can easily be designed. The design variables of the impeller and diffuser were defined in meridional plane and plane vane development, and optimal design of the impeller and diffuser was carried out by using design of experiment (DOE). The tendencies of design variables depending on specific speed were analyzed by utilizing the shape of optimally designed mixed-flow pump. The mixed-flow pump shape of required design specification was designed by using the tendencies of design variables depending on specific speed. The performance of designed model has been verified using CFD.

In this study, a mixed-flow pump with a specified specific speed was optimally designed by utilizing computational fluid dynamics (CFD). Additional, the shape of the mixed-flow pump and flow characteristics depending on the specific speed were studied, in order to development of hydrodynamic design system for mixed-flow pump. A mixed-flow pump consists of an impeller and a diffuser and the shapes of mixed-flow pumps show a certain tendency depending on specific speed. With an analysis of the shapes of a mixed-flow pump, the optimal shape to satisfy design specifications can easily be designed. The design variables of the impeller and diffuser were defined in meridional plane and plane vane development, and optimal design of the impeller and diffuser was carried out by using design of experiment (DOE). The tendencies of design variables depending on specific speed were analyzed by utilizing the shape of optimally designed mixed-flow pump. The mixed-flow pump shape of required design specification was designed by using the tendencies of design variables depending on specific speed. The performance of designed model has been verified using CFD.

Keywords
- Submerged bodies, multiphase approach, cavitating flow, turbulence, numerical simulation
- Radial inflow turbine, Preliminary design, Performance estimation, Loss models, 1D mean stream line analysis
- Fluid Machinery, Impulse Turbine, Wave Energy Conversion, Wells Turbine
- Hydrodynamic design system, Mixed-flow Pump, Impeller, Diffuser, Design optimisation
To improve centrifugal pump cavitation performance, the slot pulse jet pump is proposed. The slot is designed directly on the impeller shroud near the suction side of the blade leading edge. The slots are alternately blocked by the cambered prominence, which is designed at the impeller front side chamber along the circumferential direction. The leakage flow from the front side chamber flows into the slots alternately, and the pulse jet is formed near the blade inlet. Because of the strong shear, the pulse jet is more efficient than steady jet in terms of energy transmission. Based on the numerical simulation, the hydraulic performance of the slot pulse jet pumps with two different pulse frequencies are compared with the slot steady jet pump. The calculation result shows that with the increase of flow rate, the head and efficiency of the three kinds of pumps are basically the same. The cavitation performance of slot pulse jet pump is better than that of slot steady jet pump, especially for slot pulse jet pump II. The distribution of pressure difference acted on the slot appears to be juggled, and the pulse frequency is consistent with shaft frequency or its harmonic frequency. The amplitude of the pressure fluctuation on slot jet pump E is much higher than that of slot pulse jet pump I. The major volume distribution in the middle span of the impeller for slot pulse jet pump is less than that of slot steady jet pump. So, the pulse jet is an effective way to further improve the cavitation performance of slot steady jet pump, and NPSHr is significantly decreased, especially at design condition.

Hydrodynamic Damping of a Fluctuating Hydrofoil in High-speed Flow

A hydrofoil resembling a high-speed fighter airplane was submerged in a rectangular channel and attached to the walls in a fixed-beam configuration. The hydrofoil was excited by electro actuators and the vortex force on the hydrofoil was measured. The vibration was measured at the trailing edge with Laser Doppler Vibrometer (LDV) and a biaxial strain gauge. The hydrofoil was exposed to water velocities ranging from 0 to 25 m/s. Lock-in was observed at approx. 11 m/s. The damping increased linearly with the water velocity, with a slope of 0.02 %/m/s below lock-in and 0.13 %/m/s above lock-in. The natural frequency of the foil increased slightly with increasing water velocity below lock-in, due to the added stiffness of the passing water. Additionally, the natural frequency increased significantly when passing through lock-in due to the vortex shedding phase shift.

Research on Steady Characteristics of F-π Bridge Axial Flow Fan

The paper establishes the basis of a noise reducing valve with an F-π bridge hydraulic resistance network as its pilot circuit under the theory of F-π bridge hydraulic resistance network. It analyses the working principle of the valve and establishes the non-linear equations of flow pressure through the hydraulic resistance and force balance between the pilot and main valves. It derives the analytical expression for steady flow pressure drop of the F-π bridge electro-hydraulic proportional pressure reducing valve and then determines that the over-pressure of the F-π bridge electro-hydraulic proportional pressure reducing valve is 0.3017% of the traditional type. A simulation shows that the flow pressure drop of the F-π bridge electro-hydraulic proportional pressure reducing valve is related to the hydraulic resistances of the pilot circuit and is more stable than the traditional electro-hydraulic proportional pressure reducing valve.

Inverse Design of Airfoil Using Vortex Element Method

A methodology for the parameterization and inverse design of airfoils, for obtaining a given target surface pressure distribution is presented. The airfoil parameterization is carried out using ordered pairs representing the y-spline coordinates of ten control points of Bezier curve as parameters. The forward model consists of analysis of flow over airfoils carried out using vortex element method, which involve discretization of the airfoil curve alone, to contrast to complicated grid generation over the region of flow. The airfoil parameters are selected by global search using a Genetic Algorithm code. Examples to illustrate the parameterization and design of airfoils are presented. A good matching between the target and designed airfoil shows that present methodology can be used as a tool for the design of airfoils.

Flow Features of a Prefabricated Pumping Station Operating under High Flow Rate Conditions

In this study, computational fluid dynamics (CFD) and computational aeroacoustics (CAA) were used to reduce the noise of a small axial cooling fan that is widely applied to electronic devices. The noise of an axial cooling fan that was measured in semi-anechoic room was compared with the results obtained from CAA. Three-dimensional RANS equations were solved for the unsteady flow field simulation. The noise was calculated by using simplified Hicks’ Williams and Hawkins’ FW-H equation. The result of CAA was compared with the data obtained from the experiment. The results of unsteady flow field presented the noise generation mechanism and that of CAA presented the sound source. The shape of a sound source was a factor causing one of the major noise sources on the fan surface. The shape modification was made in the inlet of the shroud for noise reduction. The Overall Sound Pressure Level (OASPL) was reduced by 3 dB compared to that predicted by CFD model.

Enhancement of a Centrifugal Pump Performance by Simultaneous Use of Splitter Blades and Angular Impeller Diffuser

The influence of different configurations of impeller diffuser and splitter blades on performance of a centrifugal pump have been experimentally and numerically studied. Each impeller with different configurations including three different splitter blades (with R = 0.54, and 0.75, 0.5 is the length of impeller blades) and three different cases of impeller with diffuser of α=0° and the impeller having the diffuser angle of θ=45° are tested. It is shown that the optimal performance is achieved with diffuser of α=10° and the angle of θ=45°.

Influence of Different Inlet Conditions on Microstream

This paper presents the influence of different inlet conditions on the performance of microstream. On the purpose of obtaining the internal flow fields and mixing characteristics, there are 28 microstreamers with different structural parameters have been studied by numerical simulations. The influence of different inlet conditions on the mixing performance of microstreams with periodic triplate baffles and rectangular baffles was examined by experiments. The results of experiment and numerical simulation were completely discussed and analyzed. The consistency between results of experiment and numerical simulation prove the reliability of research method and the accuracy of numerical simulation. It also indicates that different numbers of inlet has significant effects on the mixing performance of microstreams. The optimum inclination angles of inlet for each kind of microstreamer were obtained under different conditions. For optimizing or designing passive microstreamers, it is an effective way to improve the performance of microstreamers by increasing the number of inlet or changing the angle of the inlets.
In this paper, the rotor and the volute of the squirrel-cage fan with dual inlet were optimized to improve its aeroacoustic performance. The blade inlet angle, blade exit angle and diameter ratio of the impeller were chosen as optimization variables using the response surface methodology (RSM) to improve the total pressure. Furthermore, another three optimization variables were adopted on the basis of previous optimum results, which are the width of the volute, the location of the impeller annular plane and the location of cutoff respectively. The simulation and experimental results show that the total pressure of the optimal model has been greatly improved without noise increase in comparison with the original model.

Flow pulsation is the inherent properties of volumetric hydraulic pump, the direct noise source of hydraulic system. Simulating the internal gear pumps' flow pulsation based on the mathematical model is difficult, so a new simulation method was adopted in this article. The chamber of the pump is divided into four Control Volumes (CVs), whose effective volumes change along with the rotation of the gears. The CVs take fluid from suction port and squeeze fluid out at the outlet port. According to the CVs connection relationship, the pump simulation model was established in ANSYS. To verify the simulation results, flow pulsation experiment was conducted and experiment data show that simulation model is correct and the accuracy is up to 95%. Results prove that the modeling method and experiment test are effective. The evaluation of flow pulsation from simulation to testing was established, which shown great significance to evaluate and develop the pump's vibration and noise.

In this study, the influences of the continuous random variation of the flow velocity on hydrodynamic characteristics of airfoil 791 was discussed, and a robust design optimization method was proposed to reduce the influence of uncertain factors on the stability of hydrodynamic performance. Firstly, Baseline curve was used to parameterize the suction side of the airfoil, and its thickness was controlled by four points, which was then taken as the optimization variables. Secondly, the criterion of robustness was given, and then a robust mathematical model was established. Finally, two objective functions of robust optimization were penalized on the basis of uncertainty analysis with surrogate model. Combined with multi-objective genetic algorithm, a robust optimal solution with better hydrodynamic characteristics was obtained. The results showed that compared with original case, the thickness near trailing edge thinned more smoothly, the resistance and surface wave intensity were obviously decreased, and the maximum reduction of drag lift ratios and wave unsmoothness were 6.85% and 25.04% respectively, which contributed to the improvement of wake quality. A change of surface pressure and resistance coefficients were also more stable under the variable inlet flow velocity conditions, and then the robustness was strengthened accordingly. In conclusion, the robust design optimization can obviously improve the hydrodynamic characteristics of the airfoil while reducing the sensitivity to uncertain factors, meanwhile, it can provide better stability during the operating process.

In the current study, the flow characteristics of a pulp suspension in a modeled headbox channel containing a screen-type circular cylinder were examined experimentally. The distributions of the pulp fiber concentration and the solid velocity of downstream wall of the cylinder were obtained using the transmitted light attenuation method and particle image velocimetry, respectively. The influence of flow velocity and the degree of pulp fiber dispersion were explained. From this information, we explored the feasibility of improving the efficiency of the hydraulic/heads of a papermaking machine.
This article compares a numerically simulated, and an experimentally obtained Hill-Diagram. The Francis99 model turbine was used in the validation. By using steady-state simulations and passage modeling in ANSYS CFD, the simulation time is in the order of minutes for each operating point. Based for the smallest guide vane opening, the error in hydraulic efficiency is less than 2.5% for all flow configurations. The individual error in head and torque follow close almost identical trends. The error along a line of small incidence bears less than 0.5% error in the efficiency in almost the complete simulated range. This results in the article may be used in future optimization design processes using Hill-Diagrams.

Hydro-power, Validation, CFD, Hill-Diagram, Optimization, Francis99
292 was only 0.1% and difference of turbine cycle work was 0.3%. So it can be deduced that decrease. So in order to evaluate turbine operating performance and design or select turbine MAP when engine working. Influences by interaction in turbine system, turbine efficiency will turbine efficiency decides exhaust gas energy utilization. Because of three-dimensional flow for turbocharged engine performance. As a part transferring heat to mechanical energy, Turbocharging is an important way to raise engine power density, save energy and reduce Wall thinning is the reduction in pipe thickness as time passes, and can strain pipes in nuclear or thermoelectric power plants. Wall thinning can be classified into three types: flow-accelerated corrosion, cavitation erosion, and solid particle erosion. This article presents a study on solid particle erosion, which frequently damages to power plant's pipe straws. Unlike previous studies, this study uses a mechanism to make solid particles (micro scale) in a flow fluid collide with pipe materials under water condition. Based on the velocity change of the solid particles in a fluid flow, the surface changes, the change in the amount of erosion, the erosion rate, and the variation in the hardiness of carbon steel and aluminum pipe materials can be all determined. In addition, factor-based erosion rates are verified, and a well-framing function is suggested for the pipe materials.

In the present study, influences of impeller splitter blades on the performance of a centrifugal pump when handling viscous fluids have been investigated numerically and experimentally. In order to approach this aim, various splitter blades have been inserted on the impeller that pump viscous fluids ranging from 1.3 to 500 cSt and the optimum splitter blade length based on the numerical results were constructed and tested thoroughly. Effects of splitter blades with lengths of 0.3, 0.45, 0.6 and 0.75 times of the main impeller blades were analyzed numerically. Results revealed that splitter blades have significant effect on the performance of a centrifugal pump. In fact, splitted impeller extend the operating region of centrifugal pumps remarkably and also among different surveyed splitter blades the head and efficiency increase by 30% and 3% respectively by using splitted impeller of 0.75L in comparison with the original impeller (without splitter blade). Moreover, the data analyses show that splitted impellers can be used for pumping viscous fluids in the almost all operating conditions of non-splittered impellers with less viscous working fluid and obtained reasonable performance.

The performance of the cone meter when measuring the cryogenic fluid was investigated by numerical simulation. The results show that the discharge coefficient and pressure loss coefficient of the cone meter are almost constant in the Reynolds number in the "stable region". The cryogenic fluids, when the Reynolds number in the "stable region". The cryogenic fluids, coefficient and pressure loss coefficient of the cone meter are almost constant and uniformity of the flow field in the flow channel is significantly increased and channel is further advanced. This approach is applied to the optimization design of an axial-flow blower in this paper, and the results show that the uniformity of flow field in the flow channel is significantly increased and the vertices spreading in the flow channel are obviously reduced, the flow is increased from 0.140(m3/s) to 0.171(m3/s), and the efficiency is increased from 45.47% to 58.13%, it indicates that the method can meet the design requirements well while being efficient.

The performance of the cone meter for measuring cryogenic micropump using two-dimensional theoretical analysis. The obtained results are compared to the results obtained from numerical simulations, and the theoretical analysis of Khan et al., and the similarities and differences between the two are discussed. The present two-dimensional theoretical analysis is validated. Its accuracy is improved as compared to previous work by setting the spiral-channel axis as the spiral-channel length from the pump inlet to the outlet and by considering the moving wall as a component in the spiral-channel direction of the centrifugal flow of the rotating disk. Furthermore, the present two-dimensional analysis can accurately predict pump performance, even though the actual flow in the micropump is three-dimensional.
Cavitating Turbulent Flow Studies for Low Head Francis Turbine by Transient Analysis

Pakkaq P, Cohl R, P Sanii

Cavitation is undesirable phenomena and it is difficult to eliminate completely, however, it is minimized to the acceptable limits. It becomes more severe under off-design conditions and lower tip clearances. Novel stress transport (SST) turbulence model and Rayleigh-Plesset mass transfer models are used using CFD solver for performing the transient simulation and investigating the cavitating turbulent flow through Francis turbine. An attempt has been carried out to analyze of cavitating flow at low head Francis turbine under different operating conditions having varying suction heads. Experimentation has been carried out to validate the simulation results. It was found that obtained simulation results are very close agreement with experimental results. Summarizing, the performance loss and cavitation rate are found maximum under low load operating conditions. At part load operating conditions of the turbine, high amplitude of pressure at low frequency has been found which may cause fatigue damage to the turbine over time. Cavitation rate, performance loss and magnitude of pressure fluctuation increase with increase of suction head.

Experimental and Numerical Studies on Flow Characteristics of Centrifugal Pump under Air-water Inflow

Quanui Si, Wenling He, Gerard Bois, Qianglei Cui, Shouyuan Yuan, Keyu Zhang

A two-phase liquid pumping test rig is built to study the induced characteristics of centrifugal pump under the air-water flow working condition. Pump performances are measured under different flow rates and different inlet air void fraction (a). Pressure fluctuation signals are measured and their probability density maps are also recorded. The calculations, using URANS k-epsilon turbulence model combined with the Euler-Euler inhomogeneous two-phase model, are also performed to obtain inner flow structure inside the impeller and volute chambers under different air-water conditions in order to understand the pump characteristic evolutions. The results show that the performance of centrifugal pump is more sensitive to air inlet injection at low flow rates. The maximum air void fraction of model pump could reach 10% when the pump operates at the highest efficiency point, and the performance drops sharply when the air void fraction is more than 8%. The dominant frequency of pump outlet pressure pulsation is still at the blade passing frequency even under two phase condition. Frequency amplitude increases with the increase of a. The greater the a, the more of low frequency appears in broadband characteristics. With the increase of a, the probability density amplitude of pressure pulsation decreases gradually, and its span becomes gradually wider as well. Comparisons between numerical local results and experimental unsteady pressure can explain part of the phenomena that are found in the present paper.

Numerical Simulation of a Gas-Liquid Centrifugal Turbine by Transient Analysis

Jin-Hyuk Kim, Bo-Min Cho, Sung Kim, Yong-Kab Lee, Qiaorui Si, Wenting He, Gerard Bois, Qianglei Cui, Shouqi Yuan, Keyu Zhang, Dimitrios Papantonis

The new world energy policy is influenced by climate changes, narrow range of operation of Thermal Power Plants, potential risks of Nuclear Power Plants and limited resources of oil, gas and coal. Taking into account that renewable energy, solar and wind power particularly are very dependent on the climate, Hydro Power takes a new role in energy systems. Electricity conversion and storage in periods of lower consumption and electricity production from the stored energy in periods of higher demand or reduced production, are crucial for the maintenance of stable and efficient electrical system. This requirement has especially strengthened nowadays due to the expansion of integration of new solar and wind plants. These renewable sources are characterized with inherent intermittent production both in daily periods and periods of several weeks, days, weeks or even months. A number of technologies might be considered for the electricity conversion and storage, but the only nature and high capacity available technology is based on pumped storage plants. This article studies the potential of the pumped storage plants as the effective and economically competitive technology for the storage of wind, solar, run-off-river and other environmentally friendly energies. Nuclear and coal fired plants can change power output to achieve demand but only at the price of extremely high maintenance cost. In addition, natural gas generators contribute to climate change and pollution only slightly less than coal. The pumped storage method is the most common storage system in the electricity sector. It is traditionally dependent on natural conditions, usually making use of rivers or lakes. However, some innovative methods such as the use of the sea as the lower reservoir, or a proposal to use a surface reservoir as the upper reservoir and an underground reservoir as the lower have emerged. In order to study the influence of inlet gas volume fraction (IGVF) on performance of a gas-liquid centrifugal pump, three-dimensional turbulent flow of a single-stage gas-liquid centrifugal pump has been simulated by using computational fluid dynamics (CFD). Both steady and unsteady solutions have been conducted for different inlet gas volume fraction conditions of the pump. The gas phase distribution, the internal flow field and pressure field of the pump were obtained with inhomogeneous two-fluid model. The result showed that many gas accumulation zones accompanied vortices appear in the impeller as IGVF increases, and surge would occur as IGVF reaches a certain value. The change of IGVF would change the magnitude and direction of the impeller radial force.

A Review of Experimental Detection Methods of Cavitation in Centrifugal Pumps and Inducers

Georgios Moumoulis, John Analigostopoulos, Dimitrios Papantonis

An important flow mechanism that affects the performance and efficiency of centrifugal pumps is cavitation. In recent years, many researchers have studied the physics of cavitation in order to create appropriate detection methodologies. The aim of this paper is to review the various experimental tools that have been developed so far and enlighten the area of future research on the field of cavitation monitoring. According to the results, cavitation detection is possible, but a large number of sensors have to be used and permanent changes in the machine need to be made for increasing the results reliability. Therefore, future research steps are proposed towards the development of reliable, accurate but also easy to install and low cost experimental set ups.

Pumped-Hydro Storages are Balancing Electric Energy Production of Wind and Solar Reducing Average Costs and Pollution

A Review of Experimental Detection Methods of Cavitation in Centrifugal Pumps and Inducers

Georgios Moumoulis, John Analigostopoulos, Dimitrios Papantonis
The present study investigated flow characteristics in the V-shaped region of the suction performance curve for a double-suction centrifugal pump based on the computational fluid dynamics (CFD). The V-shaped region in the time-averaged suction performance curve was simulated well. The CFD simulated the fluid oscillations due to cavitation surge and rotating cavitation well. The V-shaped conditions were observed in the absolute total force acting on the impeller blade, which was caused by the difference between the impeller inlet and outlet. The time histories showed that the cavity produced vorticity, resulting in an increase in a pressure loss, and a decrease in an impeller torque and an angular momentum flow rate. The time-averaged cavity volume, pressure loss between the impeller inlet and outlet, suction in the blade passage and impeller torque were examined. A V-shape of a cavity volume curve caused a A shape of a vorticity curve, resulting in a V shape of a pressure loss curve and a V shape of an impeller torque curve. The V shape of the pressure loss curve and the V shape of the impeller torque curve caused the V shape of the suction performance curve.

The impeller and volute of a single-channel pump used for wastewater treatment were simultaneously optimized to improve the hydraulic efficiency and reduce unwanted radial force sources due to impeller-volute interaction. Steady and unsteady Reynolds-averaged Navier-Stokes equations were solved with the shear stress transport turbulence model as the turbulence closure model using tetrahedral grids to analyze the internal flow in the single-channel pump. Five design variables related to the internal flow cross-sectional areas of the impeller and volute were selected to simultaneously optimize three objective functions: the hydraulic efficiency, the sweep area of the radial force, and the sweep area of the axial force during one revolution, and the distance of the mass center of the sweep area from the origin. A response surface approximation model and a genetic algorithm were employed to obtain the three-dimensional Pareto-optimal solutions representing the trade-off between the efficiency and the radial force sources. The three-objective optimization results showed that the representative clustered optimum designs exhibit enhanced efficiency and reduced radial force sources simultaneously in most cases, compared with the reference design. The trade-off relationship between the efficiency and the radial force sources clarifies with controlling the internal flow cross-sectional areas of the impeller and volute of the single-channel pump. The efficiency improvement and reduction in the radial force sources were systematically verified by analyzing the detailed internal flow characteristics.
<table>
<thead>
<tr>
<th>Page</th>
<th>Title</th>
<th>Author</th>
<th>Abstract</th>
<th>Keywords</th>
</tr>
</thead>
<tbody>
<tr>
<td>147-156</td>
<td>Modelling of a Centrifugal Pump Using the CATHARE-3 One-dimensional Transient Rotodynamic Pump Model</td>
<td>Laura Matteo, Antoine Dazin, Nicolas Tauveron</td>
<td>The CATHARE-3 predictive transrotodynamic pump model and its validation in single-phase first quadrant conditions at component scale are presented in this paper. One-dimensional discretization and resolution of governing equations are made according to a mean flow path along all pump parts (suction, impeller, diffuser, volute, and discharge pipe), what makes the model original compared to classical 1D models. The model is first verified by comparison to the Euler equation of turbomachinery results. Then, qualification of the model is carried out in steady and transient conditions by comparison to available experimental data. Head and torque characteristics curves are well predicted at different rotational speeds. Finally, a fast startup transient is simulated. Results are satisfactory at the difference between the experimental and modelled non-dimensionalized head is less than 10% of the final value during the whole startup.</td>
<td>Rotodynamic, pump, transient, model, CATHARE-3 prediction</td>
</tr>
<tr>
<td>156-166</td>
<td>On the External Effects Affecting Torsional Modes in Guide Vanes</td>
<td>Peter T K Dhoby, Brind Mynard, Jan Tore Bidal, Bjorn Haugen</td>
<td>The present study was aimed to reveal the possible interactions between the left ventricular assist pump and the blood. The study was performed in here adopted the computational fluid dynamics approach to analyze the effects of variations in blade height, rotation speed, and suspension height on the dynamic performance of the left ventricular assist pump. In addition, the 3D printing technology was tested as a reliable of manufacture of the assembly parts the left ventricular assist pump. The study was designed and conducted only after system debugging was accomplished. The computer-aided simulation methodology showed that the blade height and suspension height had varied functional on the dynamic performance of the device. Comparison of the influence degree of each factor to the blood indicated that rotation speed has the largest impact, followed by the blade height and finally the suspension height. At the same time, the experimental results proved that the design principle was rational and if prepared the left ventricular assist pump could operate successfully. The computer-aided simulation and experiment also reproduced the possible effects of the various factors on the blood and provided a theoretical reference for the design of the centrifugal ventricular assist pump.</td>
<td>Left ventricular assist pump, Computational fluid dynamics, Wall shear stress, 3D printing</td>
</tr>
<tr>
<td>169-180</td>
<td>Three-Dimensional Theoretical Study on Flow Characteristics of a Spiral-Channel Viscous Micropump</td>
<td>Donghyuk Kang, Kocchi Nishida, Kataro Sato, Kazuhiko Yokota</td>
<td>This study investigated the possible mechanisms which may negate resonance in the torsional modes of the guide vanes including, hydrodynamic damping from the flowing water and friction in the guide vane walls. A case study is conducted on a guide vane where the calculated natural frequency is significantly lower than the excitation frequency due to resonance has been reported. This paper investigates some of the possible mechanisms which may negate resonance in the torsional modes of the guide vane including, hydrodynamic damping from the flowing water and friction in the guide vane walls. A case study is conducted on a guide vane where the calculated natural frequency is significantly lower than the excitation frequency due to resonance has been reported. However, the findings are generalized to Francis turbines of different specific speeds. The results indicate that the dynamics in the bearings are especially important to be able to avoid resonance.</td>
<td>Micro-pump, Viscosity, Spiral channel, Pump performance, Three-dimensional analysis, low Reynolds number</td>
</tr>
<tr>
<td>181-188</td>
<td>Simulation Studies on the Hemodynamics of a Centrifugal Ventricular Assist Pump</td>
<td>Qiaoyan Feng, Dong Wang, Kun Wang, Mingyin Zhao</td>
<td>In order to study the unsteady pressure pulsation and internal flow in a centrifugal pump under low flow rate, numerical calculation of a centrifugal pump was carried out. The accuracy of the numerical calculation method was verified with the external characteristic test. The characteristics of the unsteady pressure pulsation and internal flow of the centrifugal pump were analyzed in detail. The results show that the interaction between the impeller and the volute has a significant effect on the unsteady pressure pulsation of the centrifugal pump. The essential reason of the periodic pressure pulsation was explained, and the evolution of separation vortex formation and collapse in the volute of the impeller was presented. Inflow pressure of the impeller was reduced. Regulation of the generation and development of the high pressure zone in the volute was achieved. With the reduction of the flow rate, the influence of the inflow pressure zone in the impeller is enhanced and the pressure near the volute tongue is decreased. The results can provide a reference for the centrifugal pump design and optimization.</td>
<td>Centrifugal pump, pressure pulsation, vortex, numerical calculation</td>
</tr>
<tr>
<td>190-199</td>
<td>Investigation of Unsteady Pressure Pulssation and Internal Flow in a Centrifugal Pump under Low Flow Rate</td>
<td>Qieng Gao, Jun Ye, Bingfeng Wang, Jin Zhao</td>
<td>Francis turbines with high specific speed may produce intense pulsations of system pressure when operating at approximately 70-85% of best-efficiency discharge. Frequencies between 1 and 4 times runner rotation are typical. The paper reviews and explains the particular set of properties. The pulsation is due to instability of the second-lowest eigenmode of the hydraulic system containing a cavitation draft tube vortex. A 3D model in frequency domain is used - with emphasis on the crucial role of the finite velocity of wave propagation - to show how this mode can become unstable and give rise to a strong ‘breathing’ pulsation. A parameter study shows the well-known influence of draft tube pressure and why the behavior of prototype turbines is often different from the reduced-scale model. Some of the phase properties are explained by a 2D acoustic model. The phase reversal between upstream and downstream pressures is a consequence of the mode shape, with roughly two quarter waves of pressure along the cavitation vortex rope. The stability is quite sensitive to 2D effects; therefore the present 3D model, while useful for understanding the phenomenon, is not likely to permit reliable predictions.</td>
<td>Francis turbine, draft tube surge, vortex cavitation, high partial load, stability, mass-flow gain</td>
</tr>
<tr>
<td>200-210</td>
<td>On the High-Partial-Load Pulsation in Francis Turbines</td>
<td>Peter K. Dörfler</td>
<td>Francis turbine performance at high partial load, high cavitation pressure, and other operational conditions is non-trivial. The frequency domain power grid network. The frequency band of the power grid is normally 40-60 Hz. The high-accuracy performance predictions have not been achieved over a wide range of operating conditions. The present study proposes a new methodology for predicting the behaviour of Francis turbines with high specific speed. The results indicate that the dynamics in the bearings are especially important to be able to avoid resonance.</td>
<td>Francis turbine, draft tube surge, vortex cavitation, high partial load, stability, mass-flow gain</td>
</tr>
<tr>
<td>217-222</td>
<td>Measurements on a Model Francis Turbine during Start-Up</td>
<td>Rahul Goyal, B K Gandhi, M J Cinmanee</td>
<td>Francis Turbine startup, shutdown, cavitatation, stability, and other operational conditions is non-trivial. The frequency domain power grid network. The frequency band of the power grid is normally 40-60 Hz. The high-accuracy performance predictions have not been achieved over a wide range of operating conditions. The present study proposes a new methodology for predicting the behaviour of Francis turbines with high specific speed. The results indicate that the dynamics in the bearings are especially important to be able to avoid resonance.</td>
<td>Francis turbine, start-up, shutdown, particle image velocimetry, pressure measurements, rotor-stator interaction, flow dynamics</td>
</tr>
<tr>
<td>Page Range</td>
<td>Title</td>
<td>Authors</td>
<td>Abstract</td>
<td></td>
</tr>
<tr>
<td>------------</td>
<td>-------</td>
<td>---------</td>
<td>----------</td>
<td></td>
</tr>
<tr>
<td>228-234</td>
<td>A comparative study of bi-directional airflow turbines</td>
<td>Manabu Takao, Seisuke Fukuma, Shinya Okuhara, M. M. Ashraful Alam, Yoichi Kinoue</td>
<td>In an oscillating water column (OWC) based wave energy plant, a bi-directional airflow is generated in the air chamber. To harness energy, the bi-directional airflow turbines that rotate in the same direction are used in such wave energy conversion devices. Some turbines for bi-directional airflow have been proposed to date, and their performance was investigated by wind tunnel tests and CFD analyses. Some of the typical turbines have inherent disadvantages, such as severe stall problem and low efficiency. Therefore, authors proposed two unique turbines for bi-directional flow: Wells turbine with booster and counter-rotating impulse turbine. An extensive computational work was conducted to perform a comparative study between the conventional and proposed turbines for bi-directional airflow.</td>
<td></td>
</tr>
<tr>
<td>235-243</td>
<td>One-way Coupling Numerical Simulation of Cryogenic Cavitation Around an Inducer</td>
<td>Yu Ito, X Zheng, Takao Nagasaki</td>
<td>To numerically simulate cryogenic cavitation of turbomachinery, a mathematical model was developed, combined with the bubble size distribution model and Rayleigh–Plesset and heat conduction equations. In the bubble size distribution model, the bubble growth/decay was solved for each class of bubble mass when bubbles with various masses mixed in the same spatially discretized calculation region. The bubble growth/decay calculations included a combination of the Rayleigh–Plesset equation for the bubble oscillations and heat conduction equation in a thermal boundary layer around a bubble, to evaluate the evaporation or condensation mass rates. This study demonstrated that the cavitation mathematical model has the potential to contribute to the simulation fidelity for turbomachinery for a fluid with strong thermodynamic effects.</td>
<td></td>
</tr>
</tbody>
</table>