CONFERENCE PROGRAM

The 12th Asian International Conference on FLUID MACHINERY

12TH AICFM

Date

25th- 27th September 2013

Venue

Hotel SANTIKA PREMIERE, YOGYAKARTA INDONESIA

Organized by

Asian Fluid Machinery Committee (AFMC)

&

Division of Energy Conversion
Faculty of Mechanical and Aeronautical Engineering
Fluid Mechanics Laboratory
Engineering Center for Industry
Institut Teknologi Bandung

Rundown AICFM12

Day 1 Wednesday, September 25, 2013

	Time		Place	Program
1	8:00	10:00		Registration SANTIKA Premiere Hotel
2	10:00	10:15		Coffe break
3	10:15	10:30	Ball room	Report from OC AICFM12
4	10:30	10:45	Ball room	Speech AFMC Chairman
5	10:45	11:00	Ball room	Opening Remark Rector ITB
6	11:00	11:30	Ball room	Keynote Speech 1
7	11:30	12:00	Ball room	Keynote Speech 2
8	12:00	13:00		Lunch

			Room		
			Pumps	**Hydro Turbine**	**Miscellaneous**
	Time		**1**	**2**	**3**
9	13:00	13:20	Paper 009	Paper 006	Paper 039
10	13:20	13:40	Paper 017	Paper 011	Paper 042
11	13:40	14:00	Paper 040	Paper 026	Paper 045
12	14:00	14:20	Paper 048	Paper 053	Paper 065
13	14:20	14:40	Paper 049	Paper 056	Paper 055
14	14:40	15:00	Paper 064	Paper 019	Paper 069

	Time		Place	Program
15	18:00	21:00	Ball room	Welcome dinner
				Opening ceremony
				Traditional performance
				Mahabrata or Dewi Sinta Dancing
				Javanese Dancing
				Welcome remark from delegates

Day 2 Thursday, September 26, 2013

	Time		Room		
			Pumps	**Hydro Turb & Cavit**	**Miscellaneous**
	Time		**1**	**2**	**3**
1	8:00	8:20	Paper 067	Paper 032	Paper 003
2	8:20	8:40	Paper 076	Paper 037	Paper 013
3	8:40	9:00	Paper 020	Paper 046	Paper 068
4	9:00	9:20	Paper 054	Paper 057	Paper 072
5	9:20	9:40	Paper 058	Paper 066	Paper 101
6	9:40	10:00	Paper 050	Paper 085	Paper 044
7	10:00	10:20	Coffee Break		
			Pumps	**Cavit & HELN Fan**	**Miscellaneous**
8	10:20	10:40	Paper 051	Paper 086	Paper 074
9	10:40	11:00	Paper 081	Paper 090	Paper 099
10	11:00	11:20	Paper 008	Paper 091	Paper 007
11	11:20	11:40	Paper 034	Paper 092	Paper 010
12	11:40	12:00	Paper 038	Paper 093	Paper 018
13	12:00	13:00	Lunch		
			Pumps & Jet Technology	**HELN Fan & Heat Trans.**	**Miscellaneous & Fluid Dynamics**
14	13:00	13:20	Paper 041	Paper 094	Paper 027
15	13:20	13:40	Paper 098	Paper 002	Paper 043
16	13:40	14:00	Paper 102	Paper 004	Paper 030
17	14:00	14:20	Paper 001	Paper 005	Paper 014
18	14:20	14:40	Paper 077	Paper 033	Paper 025
19	14:40	15:00	Paper 078	Paper 097	Paper 047
20	15:00	15:20	Coffee Break		
			CFD	**Fan Blower**	
21	15:20	15:40	Paper 012	Paper 015	
22	15:40	16:00	Paper 071	Paper 031	
23	16:00	16:20	Paper 082	Paper 052	
24	16:20	16:40	Paper 023	Paper 100	

	Time		**Place**	**Program**
25	16:40	17:15	Ballroom	Closing
26	17:15	18:00		Discussion of AFMC Member, Announcement AICFM13 place

Day 3 Friday, September 27, 2013

Visit Borobudur temple and Merapi Volcano

	Time		Place	Program
1	8:00	8:30		Registration
2	8:30	10:30		Go to Borobudur Temple
3	10:30	12:30		Borobudur Temple
4	12:30	13:30		Lunch
5	13:30	14:30		Go to Merapi vulcano
6	14:30	16:00		Merapi vulcano
7	16:00	17:00		Go back to Santika Premier Hotel

Paint Removal of Airplane & Water Jet Application

Xue Sheng-xiong[1], Chen Zheng-wen[1], Ren Qi-le[1], Su Ji-xin[1],

Han Cai-hong[1], Pang lei[1]

[1]Hefei General Machinery Research Institute, No.888 changjiang weat road,
Hefei city, Anhui province, China,xueshengxiong@sohu.com

Abstract

The paint removal and recoating are the very important process in airplane maintenance. The traditional technology is to use the chemical way corroding the paint with paint remover. For changing the defects, corrosion & pollution & manual working, of the traditional technology, the physical process which removes the paint of airplane with 250MPa/ 250kW ultra-high pressure rotary water jetting though the surface cleaner installed on the six axes robot is studied.

The paint layer of airplane is very thin and close. The contradiction of water jetting paint removal is to remove the paint layer wholly and not damage the surface of airplane. In order to solve the contradiction, the best working condition must be reached through tests. The paint removal efficiency with ultra-high pressure and move speed of not damaged to the surface. The move speed of this test is about 2m/min, and the paint removal efficiency is about 30~40m2/h, and the paint removal active area is 85-90%. No-repeat and no-omit are the base requests of the robot program.

The physical paint removal technology will be applied in airplane maintenance, and will face the safety detection of application permission.

Keywords: Airplane surface; paint removal; water jetting technology; test research.

1. Current Status of Airplane Paint Removal

Paint removal for airplane is an important process for airplane maintenance. The reasons for paint removal of airplane are as follows: 1) No matter the military aircraft or common airplane, the oxidize bottom paint/polyurethane surface paint will be cracked and dropped by all kinds of corrosion, radiation and complicated environment, so the average life of these paint is four to five years[1] and they must be removed partly or wholly. 2) When the airplane need be returned to factory to repair, old airplane removal and new paint spraying is a necessary process. But the popular paint removal method is chemical process using chemical paint remover.

Chemical method of airplane paint removal is achieved through smear, dissolution, permeability, swelling, peeling, and the reaction etc[2-3]. series physical and chemical processes with solvent-based paint remover, which mainly contains chlorinated hydrocarbon solvent. The principle is that the main solvent molecules could permeate to the gap between the macromolecular chains of the polymer. It could swell and stretch the chains, and thereby to produce stress to release the adhesion of the film to the substrate. The efficiency is high but it need 2.4 ton paint remover for removal of a medium-sized transport aircraft. The toxic solvent will pollute the air, environment and human health. For solving these problems, people tried many other technologies such as paint remover without chlorinated solvent, shot blasting with plastic pellet, and laser light. But these methods could not be used for many reasons.

Due to serious harm with chemical paint removal process, the author used ultra-high pressure and big flow rotary water jetting technology to remove paint of airplane which has big size and curve surface. This process is cooperated by six axes robot and vacuum suction system. This paper introduced the threshold pressure test and water jetting test, with surface cleaner device and vacuum suction system. Furthermore, teach and program robot to move the surface cleaner device toward airplane and change two rotary water jetting type, we got expected result of paint removal quality and efficiency from old airplane. It is absolutely no polluted for atmosphere, environment and human health, and it is a physical process paint removal with water.

2. Similarities and Differences for Paint Removal and Rust Removal

The research for airplane paint removal using ultra-high pressure rotary water jetting technology is mainly based on ship rust removal application, which is researched by author for long time. What are the similarities and differences for paint removal and rust removal? The common points of paint and rust are as follows: they are the subsidiary matter of metal surface and they can be removed by force; they are adhered to the big size and curve surface, the robot is the perfect operating device; after removal they

A parametric study on anti-vortex film-cooling holes

Jun-Hee Kim[1], Sun-Min Kim[1], and Kwang-Yong Kim[2]

[1]Department of Mechanical Engineering, Graduate School, Inha University
253 Yonghyun-Dong, Incheon, 402-751, Republic of Korea, jhkim0125@inha.edu, sunmin@inha.edu
[2]Department of Mechanical Engineering, Inha University
253 Yonghyun-Dong, Incheon, 402-751, Republic of Korea, kykim@inha.ac.kr

Abstract

A parametric study on anti-vortex film-cooling holes has been conducted with two geometric variables, viz., the diameter of anti-vortex holes and the lateral distance between the primary and the anti-vortex holes. Three-dimensional Reynolds-averaged Navier-Stokes equations with shear stress transport turbulence model were used to analyze the flow and convective heat transfer. Numerical results were validated compared to experimental data. In order to assess the film-cooling performance, the spatially-averaged film-cooling effectiveness averaged over an area of 3 primary hole diameters in width and 21 primary hole diameters in streamwise length, was evaluated. It was observed that the two geometric variables have large effects on the film-cooling effectiveness.

Keywords: Anti-vortex holes, Navier-Stokes equations, Film-cooling effectiveness, Kidney-vortex, Anti-kidney vortex.

1. Introduction

In recently years, advanced gas turbines require high inlet temperature of the turbine stage to achieve high thermal efficiency. Therefore, cooling techniques for the turbine blades are necessarily needed to maintain the performance and acceptable life span of the blades. Especially, film-cooling techniques have been widely used to reduce surface temperature of the turbine blades by forming a protective layer of coolant air on the blade surface.

Initially, various researches on cylindrical film-cooling hole were performed to investigate effects of geometric variables on film-cooling performance. The geometric variables, viz., the length of holes, blowing ratio (M), ejection angle, and compound angle have been regarded as main variables which affect the film-cooling performance. Nasir et al [1] investigated the effects of the lateral ejection angle and inclination angle on film-cooling performance for the cylindrical hole shape. In this study, the film holes with the lateral ejection angle showed higher film-cooling performance than others. Walters and Leylek [2] carried out both numerical and experimental studies on the flow field with the cylindrical holes and visualized kidney-vortices in the main duct. It was reported that the kidney-vortices causes reduction in attachment length of coolant air to the surface.

In case of cylindrical film-cooling hole, cost savings on manufacturing is ideal due to the simply geometry, but there is an adverse effect: The improvement of the film-cooling performance is limited. For the sake of the better film-cooling performance, shaped film-cooling holes were introduced. Many researches focused on ways to enhance the lateral spreading of the coolant. It can be achieved by reducing momentum of the coolant upwards. Experimental investigations with the cylindrical and the fan-shaped holes were performed to study the effects of the inclination angle and the lateral ejection angle on the film-cooling performance by Yuen et al. [3]. As the results of the experiment, the fan-shaped holes showed higher film-cooling performance than the cylindrical holes due to the reduced momentum at the hole exit plane. Also, Gritsch et al. [4] provided the experimental data for the film-cooling effectiveness of fan-shaped hole. The study was performed with five variables: the length of holes, area ratio of entrance to exit of holes, period of holes, lateral ejection angle, and lateral expansion angle. Among these variables, the period of holes had the largest effects on the film-cooling effectiveness. Baheri et al. [5] performed numerical study of how trenched cylindrical and shaped holes affect the film-cooling effectiveness. They reported that the trenched hole showed the best cooling performance. Brauckmann and Wolfersdorf [6] experimentally investigated the effects of the compound angle and blowing ratio on heat transfer coefficient and the film-cooling effectiveness, and reported that the variation of the compound angles hardly influenced the film-cooling effectiveness.

Conjugate Heat Transfer Study at Coolant-Mainstream Interaction Surface of NGV Leading Edge with Combined Shower Head and Impingement Cooling

P. M. Dileep Chandran[1] and B.V.S.S.S. Prasad[1]

[1]Thermal Turbomachines Laboratory, Department of Mechanical Engineering,
Indian Institute of Technology Madras, Chennai, 600036, India.
dileeppm@rediffmail.com, prasad@iitm.ac.in

Abstract

As leading edge experiences the highest thermal load in a nozzle guide vane, a detailed flow and heat transfer study in that region is imperative. A computational study is carried out on a solid curved plate, representing a typical gas turbine NGV leading edge, cooled by a combination of four rows of impingement jets and five rows of showerhead holes. The SST κ-ω turbulence model is used for computations. The effects of blowing ratio and Reynolds number are studied on the overall effectiveness of two plates with different thermal conductivity using Conjugate heat transfer technique. Blowing ratio effect is found to be different for different plates and the overall effectiveness is found to decrease with increase in Reynolds number.

Keywords: Gas turbine blade cooling, conjugate heat transfer, leading edge, film cooling, impingement cooling, CFD

1. Introduction

Heat transfer studies related to the leading edge cooling of a nozzle guide vane (NGV) are of great relevance and interest for the design of modern gas turbines, as it is the part most exposed to the hot gases. Improper understanding and design of cooling configuration in this region may cause very high thermal stress and may ultimately lead to the failure of the blade. Hence the effects of hole configuration, hole shape and size, blowing ratio, coolant to mainstream density ratio, Reynolds number, mainstream turbulence intensity etc on the leading edge heat transfer has been of immense interest to researchers.

Most of the earlier investigations were reported by comparing the adiabatic effectiveness (η) on the leading edge surface, which is defined as follows:

$$\eta = \frac{T_m - T_{aw}}{T_m - T_c}$$

Mehandale and Han [1] used a cylindrical model with two rows of film holes for studying the effect of Reynolds number and film hole spacing on the adiabatic effectiveness. They found an increase in effectiveness value with increase in Reynolds number. A similar study was conducted by Ou and Rivir [2,3] by adding an extra row of film hole at the stagnation region and concluded that Reynolds number effect is significant at low blowing ratios and high turbulence. The effect of mainstream turbulence was also investigated by

Experimental and Numerical Investigations on Performances of Darriues-type Hydroturbine with Inlet Nozzle

Daisuke MATSUSHITA[1], Kei TANAKA[2], Satoshi WATANABE[1], Kusuo OKUMA[1] and Akinori FURUKAWA[3]

[1]Department of Mechanical Engineering, Kyushu University
744 Motooka, Fukuoka, 819-0395, Japan, matu@mech.kyushu-u.ac.jp, fmnabe@mech.kyushu-u.ac.jp
[2]Graduate School of Engineering, Kyushu University
[3]Oita National College of Technology, 1666 Maki, Oita, 870-0152, Japan, a-furukawa@oita-ct.ac.jp

Abstract

Low head hydropower is one of realistic renewable energies. The Darrieus-type hydro turbine with an inlet nozzle is available for such low head conditions because of its simple structure with easy maintenance. Experimental and numerical studies are carried out in order to examine effects of gap distances between the runner pitch circle and two edges of the inlet nozzle on turbine performances. By selecting narrower gaps of left and right edges, the performance could be improved. From the results of two dimensional numerical simulations, the relation between the performance and flow behaviors around the Darrieus blade are discussed to obtain the guideline of appropriate inlet nozzle design.

Keywords: low head hydropower, Darrieus turbine, inlet nozzle-edge gap, performance test,

1. Introduction

Needless to say, the global warming and energy problems are getting into serious ones. The most preferable solution is to utilize renewable energies. Hydropower is one of them, but extra-low head hydropower less than 2m is almost undeveloped yet. Figure 1 shows a selection chart for hydro turbines, where *H* and *Q* are effective head and flow rate, respectively. As there has been no turbine suitable for utilization of low head power (*H*<2m), we are developing a Darrieus-type hydro-turbine system for such extra-low head hydropower, and have demonstrated its effectiveness through laboratory experiments [1-3].

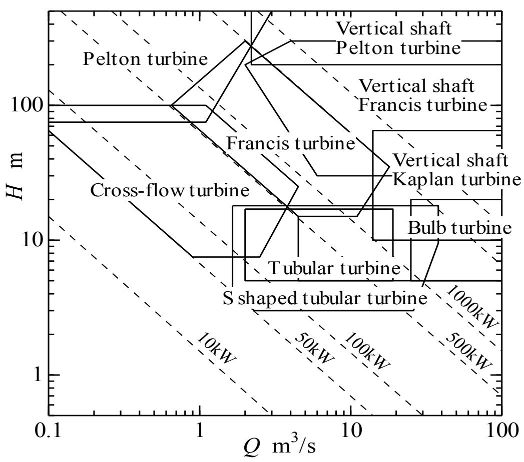

Fig.1 Selection chart of hydro turbines