2005 Asian Fluid Machinery

Edited by Xue Shengxiong Ph.D., Shu Pingling

合肥工业大学出版社
Hefei University of Technology Press

2005 Asian Fluid Machinery

THE 8TH ASIAN INTERNATIONAL CONFERENCE ON FLUID MACHINERY

Oct 12~15, 2005

Yichang · China

Edited by

Xue Shengxiong Ph. D., Shu Pingling

合肥工业大学出版社

Hefei University of Technology Press

内容提要

本书集中了中国、日本、韩国、印度等亚洲国家相关学者近两年来在泵、阀门、压缩机、风机、流体密封喷射技术和水轮机等技术领域的最新科研与产品技术成果，从中反映出了亚洲国家流体机械的最新技术水平和专家队伍。

本书可供高等院校、科研院所、机械制造企业的流体机械技术人员和应用人员使用或参考。

图书在版编目(CIP)数据

2005 亚洲流体机械:第八届亚洲国际流体机械会议=2005 Asian Fluid Machinery:the 8th Asian International Conference on Fluid Machininery/薛胜雄,舒平玲主编.—合肥:合肥工业大学出版社,2005.9

ISBN 7-81093-264-0

Ⅰ.2… Ⅱ.①薛…②舒… Ⅲ.流体机械—国际学术会议—文集 Ⅳ.TH3-53

中国版本图书馆 CIP 数据核字(2005)第 103761 号

2005 亚洲流体机械:第八届亚洲国际流体机械会议

主编 薛胜雄 舒平玲　　　责任编辑 权 怡

出 版	合肥工业大学出版社	版 次	2005 年 9 月第 1 版
地 址	合肥市屯溪路 193 号	印 次	2005 年 9 月第 1 次印刷
邮 编	230009	开 本	889×1194 1/16
电 话	总编室:0551-2903038	印 张	64　字 数 1720 千字
	发行部:0551-2903198	发 行	全国新华书店
网 址	www.hfutpress.com.cn	印 刷	合肥星光印务有限责任公司
E-mail	press@hfutpress.com.cn	纸 张	山东光华纸业集团有限公司

ISBN 7-81093-264-0/TH·7　　　定价:300.00 元

AICFM8—YICHANG

THE 8TH ASIAN INTERNATIONAL CONFERENCE ON FLUID MACHINERY

Organizing Committee

International Advisory Committee

PREFACE

The Changjiang River flows east, the Three Gorges Dam towers up high and a high-level gorge and flat lake thus appears. The 8th Asian International Conference on Fluid Machinery (AICFM) will be held in Yichang, Hubei Province, China. Facing the great Changjiang River and seeing the bound volume of the conference papers thoughts flash across my mind and it makes me very excited. Since the 1st AICFM held in Beijing in 1984, the development of Asian fluid machinery technology looks like the scene on the Changjiang River that the powerful waves behind drive those before. The Asian countries gather here happily under the same roof, getting acquainted with new friends and keeping old friends in mind. Acquaintance, exchange, cooperation and development—AICFM has already become a forum and a bridge for the scholars of fluid machinery from Asian countries.

The 2005 AICFM conference proceedings have collected the latest scientific and technical achievements, assembled numerous advanced theories and aggregated strong professional teams. We seek differences while keeping union and seek common points while reserving differences. Here it shows us the broadth and depth of fluid machinery specialties -- pumps, valves and piping, compressors, fans and blowers, water turbines, jet technology and fluid sealing, and also presents us the commonness and characteristics of fluid machinery research: numerical simulation of flow fields, theory and experiments of vanes, inspection and analysis of fluid machinery, fluid machinery products and their applications; at the same time, this work makes the scholars of fluid machinery from Asian countries such as China, Japan, Korea, India, Singapore, etc. gather together more closely and jointly match to the future.

Chinese Fluid Engineering Society (CFES) has already successfully held 4 sessions of AICFM in China. For twenty-one years, CFES has successfully made a close cooperation with the institutional organizations of Asian counties. The conference this time also received the energetic support from the academic organizations and scholars of such countries as China, Japan, Korea, India, etc., and Mr. Cao Shuliang, Mr. Yoshinobu Tsujimoto and Mr. Kwang-Yong Kim provided a lot of help for the organization of this work, CFES hereby expresses hearty thanks to all of them!

Wish the 8th AICFM a satisfactory success!

Wish AICFM carry forward and become better and better!

Fan, Gaoding, Professor
President of Chinese Fluid Engineering Society
Aug. 2005

8th Asian International Conference on Fluid Machinery
October 12~15, 2005 • Yichang, China

Table of Contents

Session 1: B-Blower and fan

Session 2: C-Compressor

Session 3: H-Hydro-Turbine

Session 4: J-Jet Technology

Session 5: P I-Pump I

Session 6: P II-Pump II

Session 7: PⅢ-PumpⅢ

Session 8: V-Valve and Pipe

8th Asian International Fluid Machinery Conference
October 12-15, 2005 Yichang, China

Paper

PREDICTION OF FLOW CHARACTERISTICS IN MULTI-BLADE FAN

Hidechito Hayashi, Yoshio Kodama
Nagasaki University
Nagasaki, Japan

Takahiro Yamasaki
Daikin Air-Conditioning R&D Laboratory LTD
Sakai, Japan

Souichi Sasaki
Nagasaki University
Nagasaki, Japan

Tohru Fukano
Kurume Institute of Technology
Kurume, Japan

ABSTRACT

In the multi-blade fan, there is the large re-circulation region near the front shroud. Then the flow in the impeller is fairly varied in span direction. We proposed the prediction method for the flow characteristics of the multi-blade fan, in which it is considered the re-circulation region and the flow variation in span-direction. The two-step procedure is proposed. The first step is the potential flow procedure that is included the vortex ring and the source. It can be roughly estimated the meridian flow distribution in span direction and the scale of the re-circulation region. The next step is the correction of the first step. It is proposed the criterion that the static pressure is kept constant in span direction. The blade load, which is estimated with the quasi-one dimensional empirical method, makes the span-wise width modified at the outlet of the impeller. The flow characteristics out of the blade passage were estimated. The predicted results were examined with the experiments and the references. The scale of the re-circulation region is predicted well. The flow characteristics of the impeller are roughly predicted.

Organized and Sponsored by the Chinese Fluid Engineering Institution

1. INTRODUCTION

A multi-blade fan has the large advantage. It can be obtained the high pressure and large flow rate with a small scale. The cost to manufacture it is rather lower than the other type fans because of the simple geometry. But the efficiency of it is fairly low and the noise level is relatively large with the large rotating speed. It is consumed the large power. Then it is required that the fluid characteristics are improved and the performance of the fan is progressed.

The multi blade fan has the special geometry that the ratio of the inlet area to the outlet of the impeller is fairly small. The span length of the impeller is large. The chord length of the blade is short. And the deflection angle is very large. It is known that the flow of the multi-blade fan has the particular characteristics. There exits the large re-circulation region near the front shroud, where the air flows from the outlet of the impeller to the inlet [(1)]. And The main flow is biased toward to the hub-side. This re-circulation flow makes the flow varied in span direction. The separation on the suction surface of the blade is occurred [(2)-(4)] and makes the main flow being complicated [(1)-(5)]. The total pressure rise of the multi-blade fan is mainly produced with the dynamic pressure rather than the static pressure. So it is large the dynamic pressure loss compared to the other type fans. These characteristics increase the loss and decrease the performances of the multi-blade fan. It is required the improvement of this type fan and it is important for predicting the performances of the multi-blade fan to be modeled the re-circulation flow in the shroud region. It is difficult to understand theses characteristics of this type fan.

There are researched for the multi-blade fan to improve these flow characteristics [(5)-(7)]. But it has not succeeded now. It is needed the consideration of the re-circulation flow near the front shroud to improve the performances.

We propose the simple method to estimate the flow characteristics in the multi-blade fan. The method is able to estimate the re-circulation flow. The characteristics of the fan are evaluated in considering this re-circulation region. The method is consisted of the two-steps. First step is based on the potential flow theorem. The vortex ring, which is symmetry to the rotating axis of the impeller, is put the inflow to the impeller from the bell-mouth. The second step is the correction of the potential flow conditions. The criterion is proposed for it. The span-wise length of the passage at the outlet of the impeller is varied by this correction.

2. CALCULATION MODEL

2.1 Potential Flow Model

We proposed the method to reveal the re-circulation flow in the shroud region. The vortex ring in the bell-mouth and the source at a far field, which are based on the potential flow theorem, produces the flow around the impeller (see Fig.1), which includes the re-circulation flow and the span-wise flow distributions.

The static pressure rise in the impeller is not large and the chord length of the blade is short. So it can be considered that the centrifugal effect is not enough influenced to the variation of the flow in span direction. The static pressure does not much varied in the impeller. Then it is supposed that the flow in the impeller is governed with the inflow conditions.

The flow of the fan is predicted with two steps procedure. The first step is the roughly estimation of the meridian flow in the impeller that is obtained with adopting the potential flow theorem at the inlet of the blade and the passage flow. The image method is applied to simulate the flow of impeller, that is, the hub surface of the impeller is considered. The second step is the correction of the 1st step. It is set the criterion to correct it. The criterion is that the static pressure at the outlet of the impeller is put constant in span-direction. It makes the meridian flow being a little varied to keep the uniform static pressure at the outlet in span-direction.

In fig.1 the source q is located at 10 times apart of the radius from the bell-mouth in the rotating axis, which is presented as vector $\boldsymbol{R}q=(0,0,Zq)^T$. The vortex ring is located in the bell-mouth, which is presented as vector $\boldsymbol{R}v=(Rvx, Rvy, Zv)^T$. The center of the vortex is coincided to the rotating axis. The strength of it is Γ. The position and the strength of the vortex ring are decided from the condition at the bell-mouth. The image method is applied to make the radial flow at the hub side.

Then the velocity vector $V(\boldsymbol{R})$ at position $\boldsymbol{R}$ is obtained with the equation (1);

$$V(\boldsymbol{R})=\frac{q}{4\pi}\cdot\frac{\boldsymbol{R}-\boldsymbol{R}q}{\left|\boldsymbol{R}-\boldsymbol{R}q\right|^3}+\frac{q}{4\pi}\cdot\frac{\boldsymbol{R}-\overline{\boldsymbol{R}q}}{\left|\boldsymbol{R}-\overline{\boldsymbol{R}q}\right|^3}+\frac{-G}{4\pi}\int_c\frac{(\boldsymbol{R}-\boldsymbol{R}v)}{\left|\boldsymbol{R}-\boldsymbol{R}v\right|^3}\cdot d\boldsymbol{S}+\frac{G}{4\pi}\int_c\frac{\left(\boldsymbol{R}-\overline{\boldsymbol{R}v}\right)}{\left|\boldsymbol{R}-\overline{\boldsymbol{R}v}\right|^3}\cdot d\boldsymbol{S} \tag{1}$$

where $^-$ is the image vector. The vector $\boldsymbol{R}$v is the position of the vortex ring and $d\boldsymbol{S}$ is the small vector on the vortex ring, which directs to the circumference.

$$\boldsymbol{R}v=\begin{pmatrix}Rv\cdot\cos\theta\\ Rv\cdot\sin\theta\\ Zv\end{pmatrix}\qquad d\boldsymbol{S}=\begin{pmatrix}-Rv\cdot\sin\theta\\ Rv\cdot\cos\theta\\ 0\end{pmatrix}d\theta \tag{2}$$

When the parameters, the source strength q and the strength Γ and the position of the vortex ring are known, the velocity vector at the any location can be calculated by the equation (1).

2.2 Estimation of Source and Vortex Ring

It is required to estimate the source strength and the strength and the position of the vortex ring for calculating the flow characteristics in the impeller. These parameters are estimated with considering the potential flow condition. The potential flow is contacted to the wall of the bell-mouth, that the flow is attached on the bell-mouth surface. The condition is applied at the bell-mouth inlet, the 45-degree position and the throat of it. The vectors of these position are presented as $\boldsymbol{R}n_1$, $\boldsymbol{R}n_2$ and $\boldsymbol{R}$s. The normal vectors at these positions are presented as Nn_1, Nn_2 and Ns. Then the conditions of the flow along the bell-mouth wall are shown as follows.

$$\frac{V(\boldsymbol{R}n_1)\cdot Nn_1}{\left|V(\boldsymbol{R}n_1)\right|}=a_1\qquad\frac{V(\boldsymbol{R}n_2)\cdot Nn_2}{\left|V(\boldsymbol{R}n_2)\right|}=a_2\qquad\frac{V(\boldsymbol{R}_s)\cdot N_s}{\left|V(\boldsymbol{R}_s)\right|}=a_s \tag{3}$$

where the parameters of right side hand should be zero for fitting the bell-mouth.

The flow rate at the throat of the bell-mouth is given by the following equation.

$$Q = \int_0^{d_1/2} V_z(\boldsymbol{R}) \cdot 2\pi R dR \tag{4}$$

where the vector Vz is the axial component of the velocity. The flow rate estimated by this equation has to be equal to the operating flow rate Q_0.

. The source strength q and the strength Γ and the position $\boldsymbol{R}$v of the vortex ring are estimated from these equations (3) and (4). It was applied the RMS method to estimate them. That is, the following function f was introduced and the function f becomes minimum at optimum values of the source strength and the strength and the position of the vortex ring.

$$f(q,\Gamma,Rv,Zv) = a_s{}^2 + k_0\left(\frac{Q-Q_0}{Q_0}\right)^2 + k_1 a_1{}^2 + k_2 a_2{}^2 \tag{5}$$

where the k_0, k_1 and k_2 are the weight coefficients for the influence of the each parameter. These coefficients are set in 100, 0.1, 0.03, respectably. The RMS method is presented as follows.

$$\frac{\partial f}{\partial q} = 0 \quad , \quad \frac{\partial f}{\partial \Gamma} = 0 \quad , \quad \frac{\partial f}{\partial Rv} = 0 \quad , \quad \frac{\partial f}{\partial Zv} = 0 \tag{6}$$

These equations are the nonlinear equations and they are solved by the Newton iteration.

2.3 Calculation of Outlet Flow of Impeller

The flow conditions at the inlet of the blade are calculated from the equation (1) by using the source and vortex ring characteristics, which are obtained above. The effects of the blade load are considered with the modification of the span-wise length of the passage. The flow characteristics in the impeller are governed by the inlet condition of the impeller. Then they are varied in span-direction very much. The span-wise distribution is treated with dividing the span-wise small region to n. And the flow conditions are calculated for each divided region. In the 1st step, the meridian flow condition is roughly estimated by the potential flow condition. The streamlines in meridian direction are calculated by the equation (7) by increasing the time step t.

$$\boldsymbol{R}(t+1)^n = \boldsymbol{R}(t)^n + V\left(\boldsymbol{R}(t)^n\right) * dt \tag{7}$$

The flow characteristics at the n^{th} position at the outlet of the impeller are calculated for each region, where the span-wise scale dz^n is estimated from the equation (7), $dz^n = \left|\boldsymbol{R}^{n+1} - \boldsymbol{R}^n\right|$. The experimental equations by Kind[(8)] are adopted to calculate the flow characteristics at the outlet of the impeller. The flow angle at the outlet β_2 and the deviation angle σ is calculated by the following equations.

$$\beta_2 = \beta_{2b} - \sigma$$
$$\sigma = 15\left(2\frac{s}{c}\right)^{0.6} - 0.13i + 0.0007i^2 \tag{8}$$

where s and c are the pitch and chord length respectively. The β_2 and β_{2b} are the flow out angle

and the outlet angle of blade. The incident angle i is estimated from the no pre-whirl condition and is used in degree unit.
The velocity vectors at the outlet of the impeller are estimated.

$$\begin{aligned} Vr_n &= \frac{dz_{in}}{dz_n} \cdot \frac{d_1}{d_2} Vr_1 \\ Wu_n &= Vr_n / \tan \beta_2 \\ Vu_n &= U_2 - Wu_n \\ Vz_n &= \frac{dz_n}{dR_n} \end{aligned} \tag{9}$$

where dz_{in} is the span-wise scale at the inlet of the blade and is set as $dz_{in} = h/n_s$.
The pressure loss dP_l in the impeller is calculated by the following empirical equations[8]

$$\begin{aligned} k_d &= 1 - \left(\frac{Wu_2}{Wu_1}\right)^2 \\ k_t &= \left(\frac{k_d}{4} + 0.5\right)^{1.7} \left(\frac{c}{s}\right)^{0.7} + 0.35(0.01i)^2 \left\{1 + 12\left(\frac{s}{c} - 0.85\right)^2\right\} + 0.01\frac{c}{s} \\ dP_l &= k_t \frac{\rho}{2} Wru_1{}^2 \end{aligned} \tag{10}$$

where Wu_1 and Wu_2 are the relative velocity at the outlet of the impeller that are not considered the span-wise velocity.
The total pressure rise P_{02} is calculated as follows.

$$\begin{aligned} P_{th} &= \rho \cdot Vu_2 \cdot U_2 \\ P_{02} &= P_{th} - dP_l \end{aligned} \tag{11}$$

2.4 Correction of Pressure Condition (2nd Step)

The flow in the passage between the blades is influenced by the blade load distributions. In this method, it is assumed that the deviation from the potential condition is a little. Because the chord length of the blade is very small and the geometry of the blade is two-dimensional. It can be considered that the blade load is mainly influenced the radial and circumferential distributions and not much to deviate the flow to span-direction. As the flow variation in span direction caused by the blade load is considered by the correction of the pressure condition.
The each nth span-wise position evaluated from the equation (7), which is based on the potential theorem, is not considered the pressure variation with the blade load. When the pressure variation by the blade load exists in span direction, the span-wise flow is varied in span direction. It is assumed that the pressure at the outlet is seemed to be constant in span direction. That is, when the pressure is varied in span-position, the flow will be varied in span position for diminishing the span-wise pressure deferential.
Then it was set the criterion for the second step. The criterion is that the static pressure is constant in span-direction at the outlet of the impeller. The correction of the static pressure is caused the variation of the span-wise length of the small segment, so the flow characteristics at

the outlet of the impeller are varied with the corrected pressure. The averaged pressure in span at the outlet of impeller is used for the criterion.

It is shown the flow chart to calculate the separation region and the flow characteristics in figure 2. At the first step, the source strength and the strength and the positions of the vortex ring with potential theorem are estimated with RMS method. The flow region at the inlet of the impeller is divided to n_z and the streamline is calculated for finding the outlet height of each divided region. The Euler's pressure rise and the flow characteristics is estimated at the outlet of the impeller. The pressure criterion is applied to the outlet regions. The static pressure at each span location is set the averaged one. The span-wise height is corrected by adopting the static pressure criterion. The other flow characteristics are calculated again for the modified height.

3. RESULTS AND CONSIDERRATION

The validity of this method is examined in relating to the flow characteristics with the experimental results by Yamasaki and Satoh(2). And the experiments are made for examining the performance. Four fans were used, which are different in outlet diameter, but the other dimensions of geometry are same each other. The ratio of inlet and outlet diameter is 0.84 and the number of blade is 42(9).

3.1 Separation Region and Flow Characteristics

Fig.3 shows the velocity characteristics at the outlet of the impeller. The experimental data in reference (2) are shown with the black lines. The symbols × are the calculated results with this method. The positions D, E and F in the figure show the circumferential positions. It is noticed that the absolute velocity ratio C_2/U_2 is low about at unity near the shroud. It is indicated that the velocity of the fluid is same to the rotating speed. The region of the low velocity near the shroud is corresponding to the re-circulation region. The flow in the re-circulation region does not have the net flow. The velocity increases near the middle of span-position. At the case of the calculated results, the red line and the symbols × do not varied so smooth compared to the experimental results. The absolute velocity up to x/b_2=0.4 is unity, then the velocity is increased rapidly becomes about 2.0, where the net flow from the bell-mouth goes through downstream. The difference of the velocity distributions between the experiments and the calculated ones is caused by the diffusion and the mixing between the net flow and the re-circulation region. But the calculated velocity is well coincided with the experimental ones in the net flow region.

In the figure of the outlet angle α_2, the distribution of it is well represented the difference between the re-circulation and the net flow regions. The separated flow is shown at the level of α_2 equal to zero (circle symbol in figure). At the middle span, the flow angle is increased to about 10 degree where the flow is relatively attached to the blade. The estimated distribution is well coincided to the experimental distribution.

It is more cleared the difference between the re-circulation and the net flow region in the meridian velocity distributions. The velocity under the x/b_2=0.4 is almost equal to zero, so the flow toward to the downstream from the impeller does not occurred in this region. The net flow is fairly deviated to the hub side. It is cleared that the calculation with this method can be presented the actual flow characteristics.

Figure 4 is the variation of the blockage factor with the span/diameter ratio. The line is the empirical equation from reference (2). The blockage factor presents the non-dimensional scale of re-circulated region. The circle symbols are the results of the calculation. It can be seen that