Study of Half-Ducted Propeller Fan on Aerodynamic Performance and Internal Flow

September 2014

Department of Science and Advanced Technology
Graduate School of Science and Engineering
Saga University

Pin Liu
CONTENTS

Nomenclature ........................................................................... 1

Abstract ................................................................................. IV

Chapter 1 .................................................................................. 1

1.1 Background and motivation ............................................. 1
1.2 3D flow characteristics of a propeller fan ....................... 2
1.3 Literature review ............................................................... 6
   1.3.1 Design method ............................................................ 6
   1.3.2 CFD research and internal flow ................................. 14
   1.3.3 Experimental research ............................................... 21
1.4 Objectives and outline of the dissertation ....................... 26

Chapter 2 ................................................................................. 29

Quasi Three-Dimensional Design Method .......................... 29
   2.1 Design introduction and parameter selection ............... 29
   2.2 Solution of meridional flow ........................................... 30
   2.3 Blade selection on the average stream surface ............ 32
      2.3.1 Equivalent velocity triangle method .................... 34
      2.3.2 Imaginary velocity triangle method ..................... 36
      2.3.3 Expansion of blade element selection theory ........ 38
   2.4 Correction on blade geometry ..................................... 39

Chapter 3 .................................................................................. 43

Numerical Theory and Procedures ........................................ 43
   3.1 Computational procedures ........................................... 43
   3.2 Control volume technique ............................................ 43
   3.3 Basic conservation laws ............................................... 45
      3.3.1 Multiple rotating frame ....................................... 45
      3.3.2 Conservation equations for a rotating reference frame 45
   3.4 Turbulence models and acoustics model .................... 46
      3.4.1 RNG $k$-$\varepsilon$ model ...................................... 46
      3.4.2 LES model ......................................................... 48
      3.4.3 The Ffowcs Williams and Hawkings model .......... 51
   3.5 Boundary conditions and near wall treatment ............ 53
   3.6 Geometry model and grid generation ......................... 55
      3.6.1 Flow field modeling ............................................ 55
7.2 Performance curves .................................................. 105
7.3 Internal flow of designed propeller fan ................................ 106
   7.3.1 Velocity distributions ........................................... 106
   7.3.2 Blockage effect of hub stall ................................... 109
   7.3.3 Aerodynamic noise of designed propeller fan .............. 112

Chapter 8 ........................................................................ 121

Conclusions .................................................................... 121
  8.1 Dissertation conclusions ............................................ 121
  8.2 Research prospects .................................................... 123

Acknowledgment .......................................................... 124

References ...................................................................... 125
Nomenclature

\( A \ [m^2,\cdot] \) surface area or Glauert series coefficient
\( a_0 \ [m/s] \) sound speed of far field
\( C, C_s, [-] \) constant, Smagorinsky constant
\( C, W \ [m/s] \) absolute velocity and relative velocity
\( D_t, D_h \ [m] \) diameter of the rotor tip and hub
\( D_{eq} \ [-] \) equivalent diffusion factor
\( d \ [m] \) blade thickness in \( Y \) direction or distance
\( E_{re} \ [m^2/s^2] \) relative total enthalpy (rothalpy)
\( F, f \ [m^3/s,\cdot] \) flux and circulation parameter
\( G \ [kg/s] \) total flow rate
\( G_k, G_b \ [-] \) generation of turbulence kinetic energies
\( H_b \ [m] \) Eular head
\( \Delta l_{th} \ [m^2/s^2] \) theoretical enthalpy
\( I \ [N/m^2] \) unit tensor
\( k_b \ [-] \) blade blockage coefficient
\( k \ [-] \) kinetic energy
\( L_1, L_2 \ [N/m] \) energy and theoretical power
\( l, L \ [m] \) blade chord
\( N_s \ [-] \) specific speed
\( n / u \ [-/s] \) rotating speed of rotor
\( p_t \ [N/m^2] \) total pressure
\( p' \ [N/m^2] \) sound pressure
\( \Delta p_s \ [N/m^2] \) static pressure rise
\( Q \ [m^3/s] \) volume flow rate
\( q \ [-] \) quasi orthogonal direction
\( q_c, q_h \ [m] \) radii of casing and hub in \( q \) direction
\( q_b \ [-] \) intensity of fixed divergence
\( R, r \ [m] \) radius
\( r^* \ [m,\cdot] \) reference radius or non-dimensional radius
\( S, S_b, S_e \ [-] \) source terms
\( S_{ij} \ [N] \) tensor of strain rate
\( T \ [N/m] \) rotor torque
\( t \ [m,s] \) one pitch length or time
\( U_t \ [m/s] \) rotor speed on tip
\( u, v, w \ [m/s] \) components of velocity vector
\( V' \ [m^2/s,\cdot] \) velocity or volume
\( W_b \ [m/s] \) deduced velocity of fixed vortex and divergence
\( Y_M \ [-] \) fluctuating dilatation
\( y_c, y_d \ [m] \) blade camber and thickness
\( y^+ \ [-] \) dimensionless wall distance
\( Z_b \ [-] \) blade number
\( z \ [-] \) axial coordinate
\[\begin{align*}
\alpha \text{[°]} & \quad \text{attack angle} \\
\alpha_\alpha \alpha_\epsilon \text{[-]} & \quad \text{Prandtl number effecting on } k \text{ and } \epsilon \\
\alpha_\varepsilon \text{[-]} & \quad \text{swirl constant depending on swirl flow density} \\
\beta, \Delta \beta \text{[°]} & \quad \text{flow angle, turning angle} \\
\Gamma, \Delta \Gamma \text{[m}^3/\text{s]} & \quad \text{circulation, variation of circulation} \\
\gamma \text{[°]} & \quad \text{pitch angle or stagger angle} \\
\Delta [-] & \quad \text{local grid scale} \\
\varepsilon \text{[°]} & \quad \text{angle of q line with normal direction of streamline or dissipation} \\
\bar{\varepsilon}_p \text{[-]} & \quad \text{total pressure loss coefficient} \\
\zeta \text{[1/s]} & \quad \text{vortex} \\
\eta [-] & \quad \text{efficiency} \\
\theta_\alpha, \delta \theta \text{[°]} & \quad \text{camber angle, change of camber angle} \\
\kappa [-] & \quad \text{constant of Von Karman} \\
\mu \text{[kg/m}^3\text{s]} & \quad \text{viscosity} \\
\bar{\mu} \text{[°]} & \quad \text{divergence} \\
\xi \text{[-]} & \quad \text{vary ratio of axial velocity or yaw angle} \\
\rho \text{[kg/m}^3\text{]} & \quad \text{air density} \\
\sigma [-] & \quad \text{solidity} \\
\sigma_{ij} \text{[kg/m}^2\text{s}^2\text{-s]} & \quad \text{stress tensor due to viscosity} \\
\sigma_{ij} \text{[-]} & \quad \text{cascade solidity on XY plane} \\
\tau \text{[kg/m}^2\text{s}^2\text{-s]} & \quad \text{stress or torque coefficient or retarded time} \\
\phi, \phi \text{[-]} & \quad \text{flow coefficient or a scalar quantity} \\
\phi \text{[°]} & \quad \text{streamline angle with rotating axis} \\
\chi [-] & \quad \text{parameter of stream surface declination} \\
\psi [-] & \quad \text{pressure coefficient} \\
\Omega [-] & \quad \text{characteristic swirl number} \\
\omega \text{[1/s]} & \quad \text{angular velocity or rotating speed of rotor} \\
\end{align*}\]

**Superscripts**

- average component
- fluctuate quantity
- variation
- defined ratio
- equivalent quantity
- vector
+ relative to \( y^+ \)
Subscripts

1, 2       rotor inlet and outlet
a, r, t     axial, radial and tangential component
i, j       components of tensor
m          mean quantity
m, q       meridional and circumferential components
re         relative quantity
tra        translational quantity
t, h       blade tip and hub
X, Y       components on X, Y axis

Abbreviations

ASD        Acoustic Source Data
BPF        Blade Passing Frequency
DNS        Direct Numerical Simulation
FFT        Fast Fourier Transform
FW-H       Ffowcs Williams and Hawkings
LE         Leading Edge
LES        Large Eddy Simulation
MRF        Multiple Rotating Frame
PS         Pressure Surface
RANS       Reynolds Averaged Navier-Stokes
RNG        Renormalization Group
rms        root-mean-square
SPL        Sound Pressure Level
SS         Suction Surface
TE         Trailing Edge
Abstract

Propeller fan, which is an axial flow fan with no casing and guide vane, not only has large flow rate but also satisfies the requirement of the spatial restriction. In some applications, the propeller with simple structure is installed with a bell-mouth at the fan inlet to make the flow enter into the rotor smoothly. As for the half-ducted propeller fan, the inlet bell-mouth and the short casing are formed by a whole one. Due to its commonly applications for heat-exchange and ventilation in the vicinity of people, the desires to own high performance and low noise level of propeller fans have been strongly demanded from the viewpoint of energy saving and quiet living conditions. In order to understand the internal flow and aerodynamic performance of the half-ducted propeller fan, the following studies have been conducted.

Firstly, the inflow performance of the propeller fan has been investigated by experimental method, and a single I type and a single slant type of hotwire probes have been used to measure the flow field of a propeller fan which is designed by traditional axial design method. Periodic multi-sampling and ensemble average technique have been applied in the data process. As a result, the radial inflow interfere with the flow in axial direction makes the inlet flow field of the propeller fan very complex and three-dimensional. In comparison with semi-open type, the half-ducted propeller fan has relatively good performance and smooth inlet flow with a bell-mouth.

Secondly, according to the particular flow characteristics, a half-ducted propeller fan has been designed by flexible use of the design method of diagonal flow fans. And non-free vortex design method of diagonal flow fans is applied to design a half-ducted propeller fan by taking the radial velocity component into account. In comparison with the ducted design method, the half ducted design can avoid highly twisted blade, obtain better performance characteristics, and improve velocity flow field and blade loading distributions.

Then, a half-ducted propeller fan designed by the second time was made by three dimensional printer and tested in the wind tunnel in order to validate the half-ducted
design method. The pressure performance of the half-ducted propeller fan was improved comparing to the first time. And both of the simulation method and experiment measurement got the same results. In the performance curves, the designed propeller fan is near stall condition at the design flow rate. However, the half-ducted design is still feasible and practicable comparing to the traditional ducted design for a propeller fan.

Finally, the internal flow and aerodynamic performance were analysed by numerical method. Large Eddy Simulation (LES) and the Ffowcs Williams and Hawkings (FW-H) equation are used to compute the unsteady flow field and obtain the acoustic signal, respectively. And the Fast Fourier Transform (FFT) has been applied to process the acoustic signal. As a result, the low energy flow fluid concentrated near hub region blocks flow in the flow passage and makes energy loss. And the high energy fluid with very low axial velocity focused near casing and suction surface are considered to be vortex. The total sound pressure level is 93 dB and 47.7dB at outlet of fan rotor and 1 m away from rotor center. The discrete frequency might be caused by large fluctuation flow especially the hub stall.
Chapter 1

Introduction

1.1 Background and motivation

Propeller fan is an axial flow fan with no casing and guide vane [1]. It sucks the fluid from infinite space upstream, and makes them get out of it in axial direction downstream. It not only produces large flow rate but also satisfies the requirement of the spatial restriction in setup processing. The simple structure and high performance make it used in many applications. Such as, heat-exchange system for engines, cooling system for outdoor unit of air-conditioner and ventilation system for relatively sealed rooms or tunnels. In many of the applications, the propeller is installed with a bell-mouth at the fan inlet to make the flow enter into the rotor smoothly. As for the half-ducted propeller fan, the inlet bell-mouth and the short casing are formed by a whole one.

Since the design of axial flow fan is believable by using two dimensional cascade data, Inoue et al. (1980) [2] have extended this method to design the diagonal flow fan. Some other researchers [3][4] have also investigated in the design method of diagonal flow fan and obtained satisfying results. However, the more effective design of the propeller fan, especially non-ducted, could not follow the design methods of axial fans. The blade shape of propeller fans has long chord length at blade tip and short one at blade hub. The blade tip as the main work part make the opened inlet suck in air at radial direction. The radial inflow interfere with the flow in axial direction make the inlet flow field of the propeller fan very complex and three-dimensional. There are not so many researches involved in this field. The special inlet flow field and the short casing of half-ducted propeller fan make the design different with the traditional design method of axial and centrifugal fans. According to the particular flow characteristics, the design method of diagonal flow fans can be used to design this kind of propeller fans. As so far almost nobody uses diagonal flow fan design method to design axial fan conversely.

Due to its special use in the vicinity of people, the desires to own high performance and low noise level of propeller fan have been strongly demanded from the viewpoint of energy saving and
quiet living conditions. However, a few researches [5] [6] have been conducted in its special inlet flow field and design method. In this dissertation, the inflow performance of the propeller fan is investigated by experimental method, the non-free vortex design method of diagonal flow fans or compressor are applied to design this kind of propeller fan in order to take the radial velocity component into account. The designed fan is also made up and tested on the wind tunnel in order to validate the design method.

1.2 3D flow characteristics of a propeller fan

The propeller fan has simple structure and can produce large flow rate. It does not like other axial flow fans, which is an assembly with the shroud or placed in long duct for getting steady outflow and safety consideration in the installment in many applications.

One of the features in the propeller fan is the inflow, which is not only in axial direction on leading edge but also in the inward radial direction on the rotor tip according to the non-ducted construction. The bell mouth has a strong effect on the inlet flow field and it is important to design the radius of the bell-mouth [7]. Son et al. (2011) [8] studied the effects of bell mouth geometries on the flow rate of centrifugal blowers by using numerical simulation and validated the CFD results by the parallel experimental research. Their results showed that the bell mouth radius has a strong effect on the flow rate. The experimental investigation was carried out by Shiomi et al. (2011) [9] using hot-wire anemometer in order to clarify the inlet flow fields of a semi-opened propeller fan with bell mouth at low flow rate region. Two types of periodical fluctuations without rotor rotating frequency were confirmed in positive gradient performance curve areas and full install areas. A household ventilator of 310mm diameter was tested by LDV method and measured the flow field in low flow rate by hot-wire anemometer to study the vortex flow field in a semi-opened and half-ducted propeller fans [10]. There were a little differences in vortex generation in these two types. The tip leakage vortex affected the leakage flow a lot in half-ducted type, while tip vortex formed in open areas near leading edge of semi-opened type gave a big effect on tip leakage flow. Peak vorticity position of tip leakage vortex in half-ducted type inclined toward more upstream than that in the semi-opened propeller fan. The main flow region at rotor outlet was a limited area except
death areas near the hub in non-stall area[11].

The propeller fan, a rotating machinery, is easy to make out of vortex. Vortex is the main reason to produce aerodynamic noise. Many researchers studied on the noise mechanism in a rotating fan, and pointed out that flow field near tip, where tip vortex and tip leakage vortex happen, is the major flow region to cause aerodynamic noise on the propeller fans.

Jang et al. (2001) [12][13] investigated the three dimensional vortical flow field of a propeller fan used in an outdoor unit for an air conditioner by LDV (Laser Doppler Velocimetry) measurement method and LES method. DANTEC 3D traversing system with resolution of 0.05mm has been used to control the probes for measuring the 3D velocity components on the meridional plane near the rotor tip areas include inside and outside the shroud areas. At each meridional position, 30000 sampling ensemble averaged value data were obtained and the highest uncertainty of three velocity component was 1.7 percent in the radial direction based on the time averaged absolute velocity. The vortex core in the rotor blade passage was identified by a vortex identification method which were used by Sawada (1995) [14], Inoue (1998) [15] and Furukawa (1999) [16] colored by normalized helicity. Three vortex structures of the tip vortex, the leading edge separation vortex and the tip leakage vortex are formed near the rotor tip of propeller fans. The first two vortices are main flow features and formed by boundary layer's rolling-up on suction surface. But the tip vortex is dominant in the flow field near the tip and a tangential vortex ring is formed by all of them around the rotor tip. This tangential structure has a large blockage effect, which causes the high incident condition and leads the onset of the leading edge separation vortex close to the tip. In addition, unsteady nature of the tip vortex has been investigated by large eddy simulation for the propeller fan. The distributions of turbulence intensity and pressure fluctuation measured by LES and LDV have agreed fairly well. The tip vortex, which dominates the flow field in the propeller fan, fluctuating highly with time makes it into the important noise source. The noise of propeller fan was reduced by adopting two types of shroud for the rotor to control this kind of tip vortex [17]. A wavelet transform was introduced to analyze the frequency characteristics of fluctuating pressure on rotor blade in the propeller fan [18]. The dominant pressure frequency at the high pressure fluctuation region by LES agree well with torque fluctuating frequency and the fan noise frequency in the experimental results. The unsteady behaviors of the vortical flow like the tip vortex and the
leading edge separation are the main reason to generate the dominate pressure fluctuation.

Shiomi et al. (2003) [19][20] and Cai et al. (2002) [21] investigated the vortex feature of an open and a half-ducted axial flow fan by the experimental method using a two dimensional LDV system. It is clarified that the tip clearance and flow rate condition have a great effect on the generation, development and declination of tip leakage flow, and the vortex trace inclines to tangential direction.

Wang and Wu (2007) [22] analyzed the tip vortex of the propeller fan used in outside unit for the air conditioner by numerical simulation method. The research model was a lower pressure propeller fan with 3 blades which shroud is covered with 30% of rear region area. The flow could be accelerated in the flow passage of the rotor because the meridian surface is a right trapezoid with the hub inclination of 18.4°. The formation and trajectory of the tip vortex of the propeller fan were illustrated and the effect of flow rates and shroud width on its trajectory were analyzed as well. The tip vortex originates from the blade tip suction side and terminates near rotor outlet without impinging on the pressure surface of the adjacent blade. The tip vortex traces in reverse rotational direction as a ring in the rotor tip region and moves inward in radial direction when enters the coverage of shroud. The effect of flow rate on the trace of tip vortex is significant on the axial position but negligible on the radial position.

Furukawa et al. (2011) [23] analyzed the relationship of flow phenomena and aerodynamic noise in a half-ducted propeller fan designed for the air-conditioner outdoor cooling. Detached Eddy Simulation (DES) on the 3D unsteady flow field of a propeller fan was implemented. Based on Lighthill’s acoustic analogy theory, aerodynamic noise was predicted by Ffowcs Williams-Hawkins (FW-H) equations and the relation with the flow phenomena was quantitatively investigated. The DES results have been validated by the LDV test. The distributions of velocity in three directions downstream the rotor can reach a good agreement. As a result of the DES, the flow structure around rotor and the pressure fluctuation on wall have been presented and discussed. The tip vortex is brought about from the blade tip, turns up as a longitudinal vortex and flows downstream along the suction surface, changes the direction to circumferential direction when comes upon the shroud, but still keeps strong as a longitudinal vortex even downstream. The separate vortex on the shroud rolls up toward the main flow near pressure surface caused by radial
inside flow on pressure surface. This was also illustrated by the LDV test and DES results of the turbulence intensity distribution downstream of the rotor. The secondary flow raised from pressure surface to suction surface on hub and from hub to tip on suction surface gives birth to the corner separate flow on hub near suction surface. This separate flow brings about very violent wall pressure fluctuation on the right place. The pressure gradient is high near the blade tip and along the tip vortex trajectory (low pressure), and their pressure field rotates with the rotor. Based on FW-H, noise level was predicted, and the main noise is from the shroud, that is, the blade loading on the tip and tip vortex interfered with the shroud are the major reason to cause aerodynamic noise on the propeller fans.

Kusano et al. (2011) [24] have investigated 3D structures and unsteady nature of vortical flow fields in a half ducted propeller fan by Detached Eddy Simulation (DES) and the validity was demonstrated by the results of the Laser Doppler Velocimetry (LDV) measurement. The much higher pressure fluctuation caused by the rotating interaction of the blade tip loading and tip vortex was been found on the shroud than that on the blade and hub surface. They also think that the blade loading on the tip and tip vortex interfered with the shroud are the major sources of aerodynamic sound in the half-ducted propeller fan.

Yamakawa et al. (2011) [25] developed a high-efficiency propeller fan to reduce electric consumption of the fan motor for outdoor heat-pump unit. Computational fluid dynamics combined with the multi-objective optimization techniques based on the genetic algorithm. A high-efficiency propeller fan characterized with curled trailing edge tips has been obtained from Pareto optimal solutions. And the ventilation performance has been evaluated in experiment.

As mentioned above, the tip vortex plays an important role on aerodynamic performance and noise produce in half-ducted propeller fans. Aiming at the effective control of it, Kusano et al. (2014) [26] have studied the three dimensional structure of tip vortex in a half-ducted propeller fan by a Detached Eddy Simulation and Laser Doppler Velocimetry measurement. They found that the tip vortex has different behaviors in the opened region upstream of leading edge and in the ducted region covered by the shroud. The tip vortex is advected nearly along streamwise direction in the opened region, nevertheless, it is turned toward the tangential direction by the interaction with the shroud wall in the ducted region. On the other hand, the inception of the tip vortex does not affect
by the flow rate, whereas its trajectory more largely inclines to the tangential direction in the ducted region as the flow rate decreases.

As we all known, many of the propeller fans are specially used in the vicinity of people, such as, the heat-exchange fan for auto-car engine and the cooling fan for personal computers. The desires to own high performance and low noise level have been strongly demanded from the viewpoint of energy saving and quiet living conditions. The higher performance and lower noise can be achieved if the tip vortex is effectively controlled. In order to realize it, it is essential to understand the nature of the flow, the mechanism of the tip vortex and the distributions of velocity and pressure around it.

1.3 Literature review

1.3.1 Design method

In quasi three-dimensional design method of turbomachinery, the swirl velocity is assumed to be constant from hub to casing on the meridional flow before and after the blade row. In the free vortex design analysis, this swirl velocity is considered to be the same as the meridional flow that occurs in the middle of the blade radius. In reality, the flow will vary and the blades themselves are usually tapered and twisted from hub to casing. The quasi three-dimensional design analysis considered the influence of the swirl velocity and combine it with conventional 2D design analysis. The characteristic of the 2D blade element has a great effect on the performance of rotor. The investigation of two dimensional cascade is one of important aerodynamic topics and many data have been got from the wind tunnel test by predecessors. The cascade theory and the experimental data have been validated and can be used to design the rotor satisfied the specification requirement and not bad efficient in common as long as the hub ratio is not extremely small.

The carpet plots of blade chamber as a function of inlet angle, exit angle and solidity, and attack angle as a function of inlet angle, camber and solidity were given by Emery (1957) [27]. Over a range of attack angles at inlet angles of 60°, 70°, 75° and solidity of 0.75, 1, 1.5, 2, low speed cascade tests were made by using 6% thickness of chord blades chambered to have isolated airfoil coefficients of 0, 0.4, 0.8, 1.2 in a 5 inch low-speed porous wall cascade tunnel. Wake total pressure loss and exit angle were surveyed downstream, and blade pressure distributions were measured at
mid-span of the central airfoil at each attack angle. Upstream pressure and flow direction were measure in the same method in reference[28]. According to Herring et al. (1957) [29], the design attack angle is defined as the angle which gives a peak free pressure distribution and was found to be constant for each solidity over a lower range of inlet angles. The design attack angle decreases at high inlet angle, of which the maximum change appears at low solidity and the minimum occurs in non-chambered airfoil at high solidity. This decrease is associated with the variation of section normal force coefficient which is obtained by integration of blade surface pressure distribution. As the inlet angle increases, the normal force coefficient increase, which results in the lift coefficient per blade increases and each blade in the cascade approaches to an isolated airfoil condition. This change in design attack angle is significant because the narrow operation range associates with high pressure-rise condition. Near design attack angle, the thin low-chamber blade sections have low drag and reasonable lift-drag ratio comparing to thicker sections tested at lower inlet angles. In this memorandum, the relationship between exit angle and attack angle for four chamber was illustrated: for a given solidity and inlet angle, the designed exit angle and designed attack angle tend to vary linearity with chamber.

Many investigations have been conducted for potential flow in cascade as the development of electronic computer and computer technology. There are usual two methods: conformal transformation and singularity method. The former is mapping the cascade to unit circle and the latter obtains the performance of cascade by calculating the distributions of vortex and divergence on chord or camber line. Gostelow (1963) [30] primarily concerned the direct problem of application of potential flow theory to cascades and developed the conformal transformation method of Merchant and Collar (1941) [31] in order to obtain an exact solution to the potential flow around a cascade of derived airfoils. Two categories method of conformal transformation and singularity method were proposed to solve the problem of potential flow in a cascade. The solution is used as a check on the accuracy of an approximate method.

For a high performance design of axial fans, it is not only to get high efficient on the design point, but also to obtain a wide working condition and to be able to correctly predict the performance in non-working conditions. In order to supply a system and wide cascade data as a guide in the future investigations, Ikui, Inoue and Kuroumaru tried to build an experimental method to obtain correct
performance of two dimensional cascade. In 1971, they discussed the necessary of contraction effect in the cascade[32]. The development of wall boundary condition brings out the contraction effect in solid wall cascade. When the contraction effect appears, the axial velocity ratio of inlet and outlet in the center of blade span will deviate from 1 and two dimensional cascade theory basic on equal axial velocity will not be suitable. As an important physical quantity in two dimensional cascade, the axial velocity ratio is affected greatly by inlet flow angle and turning angle and but no effect come from the blade shape and solidity. No matter how to increase the axial velocity and turning flow angle, the pressure will not increase if outlet velocity is not retarded, and lower cascade performance will be got than two dimensional cascade performance.

A little variation of attack angle would give a great effect on performance of cascade when there is big inlet flow. In the case of high circumferential velocity in fans, the ratio of axial velocity and circumferential velocity becomes smaller in lower flow rate, and the stagger angle must be large according to designed flow angle. In cascade test, cascade performance cannot be got correctly unless aerodynamic performance in two dimensional flow has been sure. Some researchers tried to obtain the cascade data for high inlet flow rate cascade and Ikui et al. (1971) [33] got satisfying results basic on cascade tests. And performance correction was also discussed when it is difficult to make sure the axial velocity radio equal to 1 in cascade. In the cascade with high inlet flow angle, there are narrow choosing range for the corresponding attack angle (1972) [34]. Basic on cascade performance of known low inlet flow angle and above mentioned high inlet flow angle, Ikui et al. (1972) [35]discussed optimum attack angle by half theory method for NACA 65 system compressor cascade with inlet flow angle 30°-80° and make the carpet diagram for the selection of its blade element. Furthermore, the select coefficient of attack angle was imported and necessary carpet diagram of blade element was completed to select it in the effective working area according to applicative purpose.

Thin and circle arc blade cascade, which is easy to produce, is applied for guide vane of axial flow fan and pump, rotor and stator of axial flow fan with low pressure. However, there is big effect of viscosity in the real flow of it, and evident differences exist comparing with potential theory especially for its retarded cascade. Ikui et al. (1974) [36] made carpet diagram of the thin and circle arc blade cascade basic on cascade test in wide variation range of inlet flow angle and solidity.
The two dimensional design is crucial in the design method. Because in cascade aerodynamic design, the flow passage between blades has to be firstly decided when the design specification of total characteristics is taken into account. Then based on the two dimensional blade element which is designed to satisfy specific performance requirements, the three dimensional blade can be completed. Ikui et al. (1977) [37] reported how to use experimental, two-dimensional cascade data for a quasi three-dimensional design of axial flow compressor blades considering inclined stream surface and varying axial velocity. A theoretical analysis has been performed by a suitable transformation of stream surface into a two-dimensional plane. The meridional and peripheral velocities can be evaluated as the functions of radius and the meridional streamlines are determined by through flow solution in case of specifying enthalpy distributions in advance. In the blade to blade flow, a local flow rate and enthalpy rise coefficients were defined on each stream surfaces for convenient calculation. In order to select the blade element by use of cascade data so as to satisfy the equivalent velocity diagram, the distributed vortices and sources were used to replace the first and second terms in the differential equation. According to singularity method, a flow through a cascade is built up by placing bound vortices along the blade surface. Two dimensionless parameters, which can be estimated from a through flow problem, were introduced to represent streamline inclination and axial velocity variation through the rotor. With these parameters, a camber and a stagger which are selected from the cascade data can be corrected easily by iterating calculations. The method was to be applied to NACA 65 series compressor blade as a calculating example and a reasonable correcting diagrams for NACA 65 series compressor blades was reserved for more convenient design in the future. And the correction charts of the design camber and stagger angle were presented in 1979 [38]. In the controlled vortex design, the axial velocity can be non-uniform and the designed blade circulation can be able to specified non-constant. Most of the axial flow fan are designed by the controlled vortex design.

Inoue et al. (1979) [39] reported the blade element selection method of an axial flow compressor blades with inclined stream surface and varying axial velocity. And then in 1980, they [2] reported a quasi-three dimensional design method of diagonal flow impellers with high specific speed. Firstly, the average stream surface are determined by solving a meridional through flow for given conditions before and behind an impeller and plausible conditions inside of it. The given conditions
such as head and flow rate would be made by the experiment data on the basis of specific speed. The plausible distributions of enthalpy and peripheral velocity along every q-lines are initially given inside the blade row. The meridional flow can be determined by repeat calculation so as to obtain equal mass-flow stream tubes. Then the flow on the average stream surface which is obtained by revolving the meridional streamline about the rotational axis transformed conformally into the 2D plan, and the cascade geometries of airfoil blade can be determined by referring to the experimental 2D cascade data. Finally, the correction of the blade camber is made by considering the effects of streamline inclination and the variation of meridional velocity based on the potential theory. The total pressure loss coefficient of cascade was estimated by use of Lieblein’s work and its extension by Klapproth. The method has also used to design the blade geometry of diagonal flow fan with more inclination of the stream surface and arbitrary blade loading and validated by examining performance tests using nine diagonal fans [40].

In order to research high flow and high pressure diagonal flow fan, Kamada et al. (1986) [41] designed the diagonal flow impeller with high specific speed by extending the vortex design method applied in an axial flow compressor stage. Three kind of vortex flow: free vortex type, constant tangential velocity type with and without correction of the secondary flow effect, have been applied to design the diagonal impeller. They pointed that the blades designed by constant tangential velocity type are better for the blade manufacture which have smaller camber than the blades designed by free vortex type.

Zangeneh (1991) [42] introduced an inverse method of a fully three dimensional compressible flow for the design of radial and mixed flow turbomachinery blades. The blade shape of an initial guess is obtained by the specified rVt and assuming uniform velocity. The three dimensional inverse design method was also applied to design vaned diffusers in centrifugal compressor and centrifugal and mixed flow impellers in order to investigate highly non-uniform exit flow and analyze and minimize the generation of the secondary flows, respectively in reference [43] and [44]. The theory and application of a novel three dimensional inverse method for the design of turbomachinery blade in rotating viscous flow were systematically reported [45].

Most of turbine machines are designed by the inverse methods which make a close link between the intention of designer and the blade geometry. In the inverse methods, free vortex design and
controlled vortex design are often used by researchers [46] [47]. In the controlled vortex design, the axial velocity can be non-uniform and the designed blade circulation can be able to specified non-constant. Most of the axial flow fan are designed by the controlled vortex design. The proper blade loading distributions and reduced loss near the tip can be reached by the controlled vortex design [48]. The controlled vortex design also provides a method for the multistage machinery a reasonable distribution of exit flow angle (2002) [49]. However, all these study are not taking the radial velocity component into account. It is advantageous to take the radial velocity into account in controlled vortex design, which was investigated by Vad and Bencze (1998) [50] based on three dimensional laser Doppler anemometer measurement. The direct proportion of tangential gradient of radial velocity and prescribed span-wise gradient of ideal total head rise in design was pointed out.

Bolcs et al. (1991) [51] developed a quasi-three dimensional characteristics method for a supersonic compressor rotor with an axisymmetric stream surface of variable radius and stream tube thickness. For the supersonic inlet flow, a specified inlet Mach number yields only one flow angle called “unique incidence angle” at the blade leading edge with attaching shock wave. Based on the method, a simple quasi-three dimensional method has been derived from a given stream surface and stream tube thickness for calculating the Mach number distributions and unique incidence condition on the suction side of the profile. To validate the two characteristics methods, the results comparison of different stream surface radius and stream tube thickness distributions was established with Denton’s “time matching” method.

Spring (1992) [52] conveyed the advantages of quasi-three dimensional inverse design method used in the hydraulic design of pump impellers circulation design used for pump impellers. The methodology of this design is to control the different blade angles so as to get a new blade shape according to specified circulation. Two design examples of boiler feed pump impellers were investigated. The exact blade shape could be generated by the selection of a circulation standards. And also the blade loading characteristics could be kept under complete control which can be measured as lift coefficient, maximum vane velocity loading, and the velocity and pressure loading coefficient. It has been shown that inverse design is capable of producing a blade shape which yield the best possible diffusion rates and help increase a power plant’s output through the improvements
to old existing pumps.

Li and Wu (1996) [53] extended the quasi-three dimensional design method of axial flow impeller to a diagonal flow impeller by using the experimental cascade data of circular arc blade. An equal thickness circular arc plate was investigated to substitute for airfoil blade in low pressure operation so as to simplify manufacture and decrease production cost but not reduce the performance of fan. Novak streamline curvature method on equilibrium equation along quasi-orthogonal line was used to calculate the streamline distribution on the meridian surface. The convergence condition of loss distribution was determined by the relationship between entropy function and total pressure loss coefficient. The reasonableness and reliability of this method were verified the performances of diagonal flow fan with outlet guide volute by experiment results in different tip clearance. The flow phenomena of a diagonal rotor designed by this method were analyzed by numerical calculation method using the Reynolds-averaged Navier-Stokes equations coupled with the $k-\varepsilon$ turbulence model[54]. And the calculated streamline distribution on the meridian surface is consistent with the design value.

Demeulenaere et al. (1998) [55] have modified a three-dimensional blade shape by using a physical algorithm based on the transpiration model. The transpiration flux is obtained by imposing the static pressure distribution along the blade surfaces in the modified Euler solver. The boundary conditions on the blade walls are changed and the permeable wall is applied. Non-reflecting boundary conditions are imposed along the inlet and outlet boundary. The result flow field after the convergence of the inverse Euler calculation is used to define the new blade shape. The method shows a rapid convergence to the blade geometry corresponding to the target pressure distribution for subsonic and transonic design. The redesigned blades of transonic compressor rotor have shock-free pressure distributions obtained by subtracting the boundary layer displacement thickness from the designed blade for no account for viscous effects in inverse Euler solver. In redesign of a low aspect ratio turbine nozzle, the smooth acceleration along the suction side can be achieved by considering the effect of lean.
Misaki and Kurokawa (2004) [56][57] have prospected the trend of three dimensional blade stacking technology considering the effect of sweep and dihedral. The sweep and dihedral are defined as the span direction of blade not perpendicular to flow direction and the blade surface not perpendicular to the axisymmetric stream surface between hub and casing, respectively. As shown in Figure 1-1, while the blade leans in tangential and axial direction are tangential lean and axial lean, especially the former one is called skewed angle. Sweep and dihedral have effects on changing local velocity and pressure, decreasing blade loading, suppressing the development of boundary layer and secondary flow, and so on. In the optimization of blade design, when the blade sections are fixed, the selection to sweep and dihedral of stacking line is conducted in design method of deformed polyhedron developed from simplex method. The applications of their control mechanism to turbomachinery have been described. As for subsonic fans, high efficiency and wide operation areas can be obtained when the blades have forward sweep. The reasons are that toward tip flow near leading edge is suppressed and especially the radial flow from tip is strengthen near trailing edge.

In order to improve the performance of a half-ducted propeller fan, Tabata et al. (2009) [58] have conducted the three-dimensional aerodynamic design method by taking non-uniform meridional flow into account. The axisymmetric viscous flow calculation is applied in the process between the blade design and the evaluation of the viscous flow fields around the rotor. According to experiment
results, the higher efficiency can be obtained for the three-dimensional designed fan in comparison with the baseline fan.

Sarraf et al. (2011) [59] studied the blade thickness effects on the overall and local performances of a controlled vortex designed axial flow fan by Laser Doppler Anemometry and two dimensional hot fiber probe. The general viewpoint on the application to low-speed axial flow fans is that the extra thickness may increase the dissipation of energy in the von Karman Street behind the blades so as to decrease the fan performance and increase the pressure fluctuation in its wake. However, this dissertation shows that the application of thick blade makes a good compromise between aerodynamic and acoustic performance on a wider operating rang.

Kinoue et al. (2013) [60] designed a diagonal flow fan of radial outflow type with specific speed of 670 [min⁻¹, m³/min, m] in order to apply the design method of diagonal flow fan based on axial flow design, which could instead the centrifugal flow fan in the economical combination of a centrifugal fan and a scroll used for industrial fan in low specific speed of 600-700 [min⁻¹, m³/min, m]. The experiment investigations of fan characteristics test has been performed to verify the design value. The internal flow surveys is conducted by adopting the five-hole probe and hot wire probe. At the outlet of fan, the characteristic region is observed in features of the decrease of meridional velocity and the increase of tangential velocity near the rotor tip, which is mainly caused by the tip leakage vortex and pressure surface separation. In this dissertation, a half-ducted fan and ducted fan have been designed by quasi-three dimensional design method based on the above mentioned design method of the diagonal flow fan considering radial inflow and outflow.

1.3.2 CFD research and internal flow

Nowadays computational methods are not only used as an analysis tool but also an optimization tool for aerodynamic design problems. The computational fluid dynamics method provides an effective approach to the design, simulation, optimization [61] [62] and flow analysis of fans. It can simulation complex fluid flow and related physical phenomena by a faster total time to solution with reliability and low cost. The advantages make it a prior method to investigate the internal flow through the rotating blades, which was found to be strongly three dimensional and highly turbulence. The internal flow of fans is three dimensional and complex, as shown in Figure 1-2.
there are vortex and separation flow around blades of the rotor. Many parameters affect the internal flow of fans like specific parameters of blade shape. Beiler and Carolus (1999) [63] used a CFD technique to study the 3D flow phenomena of the flow in axial flow fans with skewed blades and validated the numerical results by experimental measurement. Based on elementary airfoil theory, two fans of low pressure rise with different flow and pressure coefficients were designed. Initially two fans were equipped with straight blades and then skewed blades in the circumferential direction were developed. A swept-forward blade increases the lift coefficient near the wall and runs more quietly over a wide flow rates. Various studies on specific parameters like this swept-forward of blade shape have carried out to improve the performance characteristics and reduce noise of fan. Such as, the sweep forward blades can reduce noise and the secondary flow [64] [65] [66], the roundness of trailing edge decreases the broadband noise [67] and so on. In order to improve fan efficiency and reduce noise for personal computers and workstations, Takahiro et al. (2008) [68] in Oriental motor investigated the effects of design parameters in the blade camber ratio, maximum camber point, blade profile, skew angle, and roundness of leading and trailing edges on fan efficiency and noise by computational fluid dynamics analysis. A steady 3D analysis was conducted by using the commercial code Scryu/Tetra based on the Navier-Stokes equations and a standard $k-\varepsilon$ model as the turbulent flow model. As a result, a most suitable camber ratio was pointed out and the discrete tone noise could be increased by moving the maximum camber point forward. Blades swept forward in rotational direction and roundness of the blade trailing edge were relaxing the interference with the spokes so as to reduce the discrete tone noise.

Except specific parameters of blade shape, the tip clearance is another element to the influence of internal flow, especially referring to the tip vortex. Lakshminarayana (1970) [69] reported the prediction method of the tip clearance effects on axial flow turbomachinery, and an expression is derived for the decrease in stage efficiency due to tip clearance. Inoue et al. (1985) [70] clarified the structure and behavior of tip leakage vortex and discussed the validity of Lakshminarayana's vortex model by experimental study. Hutton (1955) [71], Cumpsty (1989) [72] and Furukawa et al. (1999) [73] reported that the pressure rise increases and the onset of rotating stall moves to lower flow rate as the clearance decreases. An enlarged tip clearance results in the increase of broadband noise levels reported by Longhouse (1978) [74] and Fukano et al. (1986) [75].
Hoying et al. (1999) [76] reported that a short length-scale (or spike) type of stall inception in a compressor is linked to the tip clearance vortex by the numerical simulation using $k$-$\varepsilon$ turbulence model. A central feature observed during the rotating stall inception was the tip clearance vortex moving forward the blade leading edge. Based on the simulations, a local stall inception criterion for the short length-scale phenomena, namely tip vortex trajectory perpendicular to the axial direction, has been identified for axial compressors. He also suggested that the short length-scale phenomena be able to predict for tip-critical compressors single blade passage calculations, rather than computations of the entire annulus.

As Inoue (1997) [77] said that the turbomachine is a generator of vortices, there must be some vortices generated in the rotor and moreover they dominate the flow field. It is well known that the vortex produces the noise, so it is necessary to study the vortex in the internal flow of fans and pursue the methods to decrease noise [78][79][80]. Furukawa et al. (1999) [81] have investigated the breakdown of tip leakage vortex on a low speed axial compressor rotor with moderate blade loading by Navier-Stokes flow simulations and visualization techniques. It has found that the vortex breakdown plays a major role in performance characteristics near stall condition. The breakdown regions grow rapidly as the flow rate decreasing from the operation condition in peak pressure rise, which causes the blockage effect and loss to increase drastically. These region interact with blade suction surface and make its boundary layer separating, thus leading to three dimensional stall.

Gong et al. (1999) [82] presented a computational model for simulating axial compressor stall inception and development via disturbances with length scales on the order of several blade pitches. The formation and evolution of Spikes type stall inception which observed in experimental phenomena were captured by using the model.

Deguchi and Miyake (2002) [83] used unsteady numerical simulations to reveal the construction of stall cell and flow in stall situation for axial flow fans. Although the sub-grid scales (SGS) model of Large Eddy simulation was introduced, numerical viscosity produced by convection term had turned out to be larger and the calculated results were not big different with the simulation results without SGS. In a certain rotor, stable rotating stall cells present as observed in the corresponding experiments, but in another rotor they do not appear. Under the effect of rotating stall cell, a strong vortex tube is formed in several flow passages. The uppermost part is a stable, nearly two...
dimensional vortex perpendicular to casing wall and underneath the vortex tube bends down to the hub to make foots. The vortex body has different moving velocity depending on its portion. The flow induced by the vortex attributes to the distortion of whole flow field having rotating stall cell. Shiomi et al. (2003) [84] the outlet flow fields of a high specific speed diagonal flow fan in rotating stall condition by experimental method. The behavior of a stall cell was captured by a single slanted hot wire using a double phase-locked averaging technique. Hah et al. (2006) [85] reported the criteria and mechanisms of short length-scale rotating stall inception in a transonic axial compressor. A large oscillation of the tip clearance cortex as the rotor operate near the stall condition and inception of spike-type rotating stall were measured in the transonic axial compressor by experimental test.

Yamada and Furukawa (2000) [86] investigated three dimensional vortex structure and its unsteady characteristics of one stage axial compressor rotor at a near stall operating condition by RANS simulation. The nature of rotating stall inception has been captured and the formation of the tornado type vortex has been revealed. In order to obtain the rotating stall inception, the tested rotor is calculated in all circumference. The mixed mesh is used to reappear the flow field of the blade tip area correctly. The main flow region and the blade tip region are meshed by H grid. The turbulence model of Baldwin & Lomax is adopted in the RANS unsteady simulation. Near the stall point, the increasing load and stronger tip leakage vortex can be identified in the distributions of non-dimension helicity of vortex core. The adverse pressure gradient increases and makes the TLV breakdown in spiral form. Then, the interaction with adjacent blade leading edge results in focus-type separation from suction surface near leading edge tip. And the spiral TLV caused by low flow rate breakdown and interact with the LE of adjacent blade, which result in the information of tornado type vortex. The tornado type vortex having foots on casing which has low pressure and suction surface develops intensely and the span rotating stall cell is formed.

Komerath et al. (2004) [87] investigated the formation and decay of rotor blade tip vortices based on other researchers’ results on measurements and analysis until the year of 2004. A conceptual model for the development and decay of the tip vortex was put forward. A relative data from the literature has shown that just 40% of the tip vortex strength comes from the peak bound circulation for the blade with sharply-cut straight-edged tip in a wide range of tip Mach and Reynolds Number.
The reasons about this drop which are relative to both counter-intuitive and in conflict with the measurements of turbulent fluctuations are given from turbulent models. The substantial differences in vortex evolution are traced to the tip surface condition that the rougher tip blade generated the vortex with the thinner and persisting longer shear layer. The upper shear layer ends up going into the inboard vortex sheet is the genesis of the discrepancy between vortex strength and peak bound circulation. The spillage of tip vorticity into the inboard vortex sheet is also the explanation of the loss of the tip circulation.

Jang et al. (2001) \cite{88} have investigated the three dimensional vortical flow structure in a semi-opened propeller fan by a Reynolds-averaged Navier-Stokes (RANS) flow simulation. The vortical flow structure especially the vortex core is identified by a semi-analytical method based on the critical point analysis. Three types of vortex structure, tip vortex, leading edge separation vortex and tip leakage vortex, are found near the rotor tip region. And the tip vortex is found to be closely related to an aerodynamic performance of the propeller fans.

Lampart and Yershov (2002) \cite{89} have optimized the blade shape of turbines numerically by using a three dimensional RANS solver of viscous compressible flows. Direct optimization where the final blade shape is obtained from evaluating an objective function is carried out using Nelder-Mead method of deformed polyhedron, which is relatively easy to work with constraints imposed on reaction, the mass flow rate and exit angle. The minimized objective function is the total losses of the stator-rotor stage with the exit energy. The penalty function is imposed on the mass flow rate if it changed by more than 1%, compared to the original geometry. The optimized parameters are stator and rotor blade numbers, stagger and twist angles, stator sweep and lean, both straight and compound. The optimization gives new 3D stacking lines of optimized stages with increased efficiencies.

David and Ronan (2008) \cite{90} studied the effect of Reynolds number on micro-axial flow fan performance by a novel linear scaling method derived from dimensional analysis. The experimental results show that the fan performance progressively reduces in decrease of Reynolds number when \( \text{Re} \) is less than 1980 and is insensitive to Reynolds number when \( \text{Re} \) is larger than 1980. Three operating regimes, the abrupt stall, progressive stall and normal operating regime, are defined according to the fan performance curve. And Reynolds number effect was analyzed on the fan
performance curve.

Sarkar (2009) [91] investigated the influence of wake structures on the evolution of the boundary layer on suction surface by Larger Eddy Simulation (LES) for a high-lift low-pressure turbine in a Reynolds number $Re=7.8 \times 10^4$. A wake-generating cylinder upstream of a cascade was used in extracting the wake data of different characteristics instead of a moving bar. No small-scale fluctuations have been found in the span-wise direction in the wake data obtained from a 2D precursor simulation. And velocity deficit and 3D small-scale eddies in a separation-induced transition over the suction surface of a high-lift LPT blade (T106 profile) were examined by the data from a realistic 3D simulation. An inflectional boundary layer over the suction surface of a high-lift LPT blade is subject to the periodic wake, which has a mean effect and the effect of small-scale fluctuation. The Kelvin-Helmholtz instability of the separated shear layer is triggered by the low-frequency mean effect of the convective wake. Apart from the wake kinematics, the large pressure oscillations and rollup of the separated shear layer along the rear half of the suction surface depend on the length scale of the convective wake. The transition of this rolled-up shear layer is influenced by the wake turbulence and the small-scale motion.

Liu et al. (2010) [92] developed downstream flow resistance method (DFR method) to study the flow field and performance characteristics of an axial flow fan in order to improve the low prediction accuracy of the fan performance induced by the conventional methods and reveal characteristic flow behaviors of inducing the stall phenomena. This new approach engages the downstream flow resistance as a source term to the computation of the fan performance. The distributed resistance in each volume discretized from the porous region is expressed as a pressure drop per unit length in flow direction. Both the moving grid and static grid are adopted by DFR method in the platform of the commercial code, STAR-CD, for solving coupled governing equations. The prediction accuracy of the pressure difference can be dramatically improved in comparison with those obtained by the conventional method. The DFR method with the static-grid scheme is enough to obtain high precision for the case in primary consideration of the fan performance.
Shigemitsu et al. (2010) [93] proposed the application of contra-rotating rotors instead of increasing rotating speed to improve the performance of the small-size axial flow fans. The numerical analysis was conducted to study the performance and the internal flow condition of a small-size axial flow fan as the first step for the application of contra-rotating structure. The performance of a contra-rotating axial flow fan was calculated in unsteady condition, and the flow conditions were investigated at the inlet and outlet of the front rotor and rear rotor [94]. As a result, it is pointed out that considering the unsteady flow condition of each front and rear rotor is necessary for the design of a contra-rotating axial flow fan.

Sasajima and Kawaguchi (2011) [95] have studied the flow around blades of axial flow fans for cooling electronic device by Large Eddy Simulation with Dynamic Smagorinsky Model and revealed the phenomena of pressure fluctuation, separating flow and tip vortex.

Ali et al. (2011) [96][97] used Fluid dynamics solver Ansys-CFX to analyze viscous turbulent flow field of the ducted fan and especially complicated flow field near tip clearance for hover condition. The computation analysis obtained from RANS solutions for the tip treatment development were performed on 3 connected domains. A stage type interface model and general grid interface was used in rotating and stationary interfaces surfaces. The stationary inlet and outlet regions was given an opening Type Boundary condition, while the rotating fan rotor region was
simulated by adding source term and the counter rotating wall was assigned at the shroud. Stage type interface model was used to deal with modeling frame change. With respect to grid refine study, the medium mesh was used for all prediction and the non-dimensional wall distance less than 2 was achieved near the shroud and hub region. Total pressure distribution near the tip region of the ducted fan reached a very good agreement between a steady RANS simulation and experiment measurement.

The France Company Valeo has involved in engine cooling fan system design for many year. Tannoury et al. (2012) [98] described tonal noise prediction of automotive engine cooling fan by CFD technique of analytical models to make comparison for different fan geometries, and a hybrid approach was adopted for predicting tonal noise. The noise sources were extracted from an Unsteady Reynolds-Averaged Naviers-Stocks simulation and propagated into the far and free field using the Ffowcs-Williams & Hawkings\textsuperscript{\textregistered} acoustic analogy. The simulation confirmed that the tonal noise is radiated by the guide vanes mainly impacted by the wake of the rotor.

In this dissertation, the studied propeller fans have been numerically simulated in the commercial software Ansys Fluent for getting the three-dimensional internal flow. One flow passage has been modeled and calculated by periodicity method with RNG $k-\varepsilon$ visous model.

### 1.3.3 Experimental research

While computational fluid dynamics have been becoming powerful tools in recent development of computer resources, the accuracies also could not be estimated without corresponding experimental studies. The internal flow of the turbomachinery is complex three dimensional flow, include the blade affected by centrifugal force and Coriolis force, the three dimensional boundary condition wall, the collection of low energy flow caused by secondary flow, the tip leakage vortex and so on, of which they give a great effect on the performance of the rotor. As so far the development of quasi-three dimensional analysis and computer technology, there are no correct prediction methods of internal flow for any rotor. If the measurements of internal flow can be conducted easily, the flow loss will be analyzed and the goal of improvement for the performance can be finished. It is beneficial for the development of performance prediction and improvement that these experimental data accumulate.
There are two methods in measurement for the internal flow of blade rotor, the laser anemometer and the measurement sensor which is rotating with rotor. As for the laser anemometer, Particle Image Velocimetry (PIV) and Laser Doppler Velocimetry (LDV) are usually applied in the internal flow measurement of turbo-machines, whereas the measurement sensors are Hot Wire Anemometry (HWA) and Multi-Hole Probe (MHP) in common. LDV, MHP and HWA are point instruments which require to traverse along the interest domain to make out a velocity field. Nevertheless, PIV can obtain information along a cutting plane across the velocity field.

Measurement sensor has complex transverse apparatus which just can be used for the special rotor and is difficult to extend to other rotors. However, a sensor with high response fixed in the inertial frame at right outlet of rotor can be used to predict the internal flow easily if the instantaneous output is related with corresponding position on rotor. Hot wire anemometer, one of measurement sensors, is used to measure the velocity by applying a relationship of the flow speed with the resistance of the wire which is changing when the flow through heated hot wire. Generally, there are three kinds of Hot Wire anemometers: CCA (constant current anemometer), CVA (constant voltage anemometer) and CTA (constant temperature anemometer). The voltage output from these anemometers follows Ohm's law. Hot-wire anemometers have extremely high frequency-response and fine spatial resolution compared to other measurement methods, so they have been employed [99] for the detailed study of turbulent flows and any flow of rapid velocity fluctuations. Some researches [100] also use the five-hole probe, one kind of Multi-Hole Probes (MHP)[101], in studying the internal flow of fans. The five-hole probe can measure the value and direction of velocity in space, the total pressure and static pressure.

Gostelow (1977) [102] reported a method to process the samples in statistics, which were obtained from output of sensor in synchronous rotation with rotor. In 1970s, this method was used by Evans [103] to measure the boundary layer on an axial flow compressor stator blade, but just one point data would be sampled in one rotation. On one hand, a lot of time was needed to obtain all the data in a section so that relative higher error of hot-wire appeared caused by long using time. On the other hand, the variation of rotating numbers in measurement make the measurement position move because the delay times of synchronous pulse with rotor and sample pulse are fixed.

Kuroumaru et al. (1982) [104] reported periodic multi-sampling method of a slanted hot-wire and
sample pulse generator was developed based on TTL for generating correct pulse. The periodic multi-sampling measuring system could be used to measure a three-dimensional flow field and turbulent characteristics behind an impeller. A typical experiment has been carried out to show the availability of this measuring system for measuring a flow field at downstream of an axial flow impeller.

Kameier (1997) [105] measured the fluctuations of pressure and velocity near blade tip of a low-speed high-pressure axial fan with outlet guide vane using a microphone mounted flush with the casing wall and constant temperature anemometers of hot wire probe respectively. It is revealed that the tip clearance noise is associated with a rotating flow instability which only occurs under reverse flow conditions in tip clearance region. The reverse flow makes the thickness of the casing wall boundary layer increase at fan inlet and eliminates the axial component of absolute velocity in the gap, which result in vortex separation similar to part span stall.

Jang et al. (2003) [106] used constant temperature hot wire anemometer and FFT analyzer to analyze the effect of the tip clearance on vortical flow and its relation to noise in an axial flow fan. 3D vortical flow near the tip region in the axial flow fan with two different tip clearance has been investigated by analysis of experimental results and numerical calculation. Three dimensional velocity and its fluctuation were measured inside and downstream of the rotor blade by a constant temperature hot wire anemometer. The data of real-time velocity fluctuation were obtained in the relative coordinate system by controlling the hot-wire probe with traversing system installed inside of the hub with traverse resolution of 0.3 mm. ONOSOKKI FFT analyzer is used to measure the spectrum of the velocity fluctuation in order to understand the frequency characteristic of the rotor blade.

Shiomi et al. (2003) [107] analyzed the effect of zero clearance on internal flow in a high-speed diagonal flow fan by the experiment method. A high response pressure transducer and a single slant hot-wire probe were used to measure the pressure and velocity fields under the stall free and rotating stall condition by phase-locked average technique and double phase-locked average technique, respectively. They (2011) [108] use five hole Pitot tube and hot wire anemometer with a slant hot wire probe to measure the internal flow fields of a low specific speed diagonal flow fan designed by a quasi-three-dimensional aerodynamics design method. The test fan designed showed
a good performance at the design point. Averaged velocity distributions for meridional and tangential velocity near design flow rate are in agreement with design condition. The region with low meridional velocity and high tangential velocity in blade passage is recognized by phase-lock averaged velocity distributions.

Unsteady pressure on the rotating blade gives us very useful information to understand the fan flow and its noise level. Unsteady pressure measurement, a very simple of which has a parallel rotating axis to the gravity direction, has been conducted by Hirata et al. (2008) [109]. Fukaki et al. (2010) [110] have developed measuring technology for minute fluctuating pressure on blade surface with an arbitrary rotational axis direction as well as flow velocity measurements with flow visualization on propeller blade surface. A pressure transducer is flush-mounted on the pressure surface or suction surface of a fan with the radius of 235mm (5mm blade thickness). The Re effect is more remarkable on the pressure surface than the suction surface and can be negligible for large Re. The centrifugal force effect and gravitational force effect and other leading error effects have been corrected for more accurate results.

Wisler and Mossey (1973) [111] used Laser Doppler Velocimeter to measure the gas velocity within a compressor rotor for the first time. The next year, Schodl (1974) [112] exploited laser dual-beam method for velocimeter in the application of internal flow measurement in turbo-machines. Henceforth, laser velocimetry owing the advantages of no touch measure has been widely used in the measurement of internal flow. Hayami (1995) [113] summarized the Laser Doppler Velocimetry use in the flow measurement in turbomachinery. Laser Doppler Velocimetry (LDV), also known as Laser Doppler Anemometry (LDA), is the technique of using the Doppler Effect in a laser beam to measure the velocity in fluid flows. Laser Doppler Velocimeter can be outside of the flow being measured and therefore has no effect on the flow. Some researchers [114] have studied the internal flow of fans by LDV method.

Tsutsui et al. (1988) [115] have applied a two-color four-beam LDV system to measure the instantaneous flow distributions around a propeller-type wind turbine. The measured instantaneous velocity distributions on a rotor plane where the blades pass through, so the velocity could not be measured by a hot wire anemometer and a pitot tube. Three components of axial, radial and tangential velocity could be obtained from the measurement data on a horizontal plane and a
vertical plane. The LDV measurement shows that the axial velocity increases gradually after a blade passed and decreases suddenly with the next passing blade which is induced by the blade bound vortex. Jang et al. (1997) [116] measured the phase-locked flow fields inside and outside of propeller fans using a Laser Doppler Velocimetry system, and indicated that the tip vortex dominates the complicated flow field near the rotor tip pf the propeller fans. Tanikuchi et al. (2011) [117] have investigated effect of a narrow space on the noise and flow field of a small axial flow fan by experiment method for meeting miniature and contractive requirement of electrical device. The analysis of sound pressure level for several gap cases and ensemble averaged velocity was carried out by noise test in an anechoic chamber and the three dimensional Laser Doppler Velocimetry respectively.

Adrian (1991) [118] reported the Particle Image techniques for experimental fluid mechanics. Particle Image Velocimetry (PIV) is an optical method of flow visualization by which instantaneous velocity is obtained from the tracer of sufficiently small particles in the fluid. The main difference of PIV from those techniques is that PIV produces two-dimensional or even three-dimensional vector fields, while the other techniques measure the velocity at a point. PIV and LDV require optical access to the flow field, which means no interruption to the flow field but is a limitation to the experiment device as well as the range of working media.

Im et al. (2001) [119] have pointed out that PIV system proves detailed space and time resolved experimental data for understanding and controlling of flow field, and analyzed the complex three dimensional flow structure of an axial flow fan by the measurement using a high resolution stereoscopic PIV system. The three dimensional instantaneous velocity and phase-averaged velocity, instantaneous and averaged vorticity distributions and turbulent intensity have measured on typical planes of the flow field. The three dimensional flow structures and features around an axial flow fan have been shown clearly by the Stereoscopic PIV test data. Wernet et al. (2005) [120] made phase-locked three dimensional Digital Particle Image Velocimetry (DPIV) measurements near the tip region of a low speed compressor to characterize the behavior of the rotor tip clearance flow. Ali and Cengiz (2010) [121] used the Stereoscopic Particle Image Velocimeter (SPIV) to observe the three dimensional mean flow near the blade tip. The once per revolution pulse provides a phase locked triggering of the SPIV data collection system. Several novel blade tip treated fans have been
introduced by adding one blade extension of pressure side bump areas to minimize the tip leakage flow. Particle Image Velocimeter is high in cost but low in reliability and also it is hardly to measure the flow near the wall. Despite all this, the PIV techniques are still developing quickly in the visualization of internal flow in turbo-machines.

1.4 Objectives and outline of the dissertation

There are three objectives of this dissertation:

First, clarify the effect of inlet flow on performance of propeller fans.

Second, based on the first objective, design a half-ducted propeller fan by quasi three-dimensional design method considering inlet and outlet flow angle. Compare its aerodynamic performance and internal flow with the ducted design of propeller fans.

Third, make a propeller fan based on objective 2, then conduct the experimental study to validate the quasi three-dimensional design method considering inlet and outlet flow angle extending from the design method of the diagonal flow fan.

In order to achieve the above objectives, research has been carried out on the propeller fan. **Figure 1-3** presents the research procedure in this dissertation. The outline of this dissertation from Chapter 2 is described as following:

In Chapter 2 and 3, quasi three-dimensional design method and computational method applied in this dissertation will be described in detail.

In Chapter 4, experiment apparatus and measurement methods will be illustrated, furthermore, periodic multi-sampling and ensemble average technique for more accuracy velocity measurement will be introduced.

In Chapter 5, the inlet flow of propeller fans with various inlet bell-mouth will be investigated by experimental study. And the effect of bell-mouth on the performance and internal flow of propeller fans will be analyzed.

In Chapter 6, the design process and numerical simulation results of half-ducted design and ducted design will be presented. Aerodynamic performance and the distributions of velocity and pressure at the internal flow of half-ducted and ducted propeller fans obtained by numerical
simulation will be compared with the designed data.

According to the analysis in Chapter 6, a half-ducted propeller fan has been made by the three dimensional printer. The experimental study on its performance and internal flow will be conducted in Chapter 7.

In the last Chapter 8, the research of this dissertation will be concluded and the study on the inlet flow and quasi three-dimensional design of propeller fans in the future will be prospected.
Fig. 1-3 Research procedure

1. Start
2. Experimental Study & Design Specification
3. Calculation of Meridional Flow
4. Obtain Blade Geometry
5. Calculation of B2B Flow
6. Mesh in Gambit
7. Flow Field Simulation in Fluent
8. Result Analysis
9. Satisfaction with Design Specification
   - No
   - Yes
10. Blade Manufacture
11. Test in Experiment Apparatus/CFD
12. Results Analysis
   - Bad
   - Well

DESIGN

CFD

EFD/CFD
Chapter 2

Quasi Three-Dimensional Design Method

2.1 Design introduction and parameter selection

In conventional quasi three-dimensional design method, the swirl velocity is assumed to be constant from hub to casing on the meridional flow before and after the blade row. In the free vortex design analysis, this swirl velocity is considered to be the same as the meridional flow that occurs in the middle of the blade radius. In reality, the flow will vary and the blades themselves are usually widened and twisted from hub to casing especially the propeller fans.

In the quasi three-dimensional design method, the complex three dimensional flow in the real is presumed to be the composition of two dimensional flow: the meridional flow and blade to blade flow. The meridional flow and the blade to blade flow are affecting each other to consist of the real three dimensional flow. Therefore, to obtain the solution of the quasi three-dimensional flow, the effect relationships between them are taken into account in the iterative computation of the two dimensional flow. According to the flow characteristics of the half-ducted propeller fan, the design method of a diagonal flow fan is adopted in this dissertation. The design of a diagonal flow fan extended from the design method of an axial flow fan [122] will be described to inversely apply to design the axial flow propeller fan owning to the special flow characteristics at inlet flow and the casing coverage structure.

In this dissertation, the swirl velocity is prescribed to be varying with the radial, that is to say, the non-free vortex is applied as vortex design in which circumferential velocity is constant. The fundamental design parameters that volume flow rate is 10 m$^3$/min, static pressure rise is 200 Pa, rotating speed is 3000 rpm and rotor geometry parameters of radius, hub ratio and blade number are 200mm, 0.6 and 5, are prescribed so that the corresponding non-dimensional form flow coefficient, pressure coefficient and specific speed can be obtained by the following equations.

$$\phi = \frac{4Q}{\pi(D_t^2 - D_h^2) \cdot U_t}$$  \hspace{1cm} (2-1)
\[ \psi = \frac{2 \Delta p_s}{\rho U_t^2} \]  

(2-2)

\[ \eta = \frac{\Delta p_s \cdot Q}{T \cdot \omega} \]  

(2-3)

\[ N_t = \frac{n \cdot Q^2}{\frac{1}{3} \pi h^3} \]  

(2-4)

Where \( U_t \) is speed on the rotor tip. \( D_t \) and \( D_h \) are the diameter of the rotor tip and hub respectively. \( Q \) is the volume flow rate. \( \Delta p_s \) is the static pressure rise. \( \rho \) is atmospheric density. \( T \) is the rotor torque and \( \omega \) is the rotating speed of the rotor. The efficiency is presumed to 50%. According to Eular equation, theoretical enthalpy \( \Delta I_{th} \) can be described in terms of the variation in swirl velocity \( r C_{\theta} \). The total pressure is assumed to be equal to static pressure rise. Based on the presumption of zero circumferential velocity at inlet and efficiency, the relationship of outlet swirl velocity and static pressure rise is presented in Equation (2-6).

\[ \Delta I = \eta \cdot \Delta I_{th} = \eta \cdot (u_2 \cdot c_{\theta 2} - u_1 \cdot c_{\theta 1}) = \frac{P}{\rho} \approx \frac{\Delta p_s}{\rho} \]  

(2-5)

\[ \Delta I_{th} = u_2 \cdot c_{\theta 2} - u_1 \cdot c_{\theta 1} = u_2 \cdot c_{\theta 2} = \omega \cdot r_2 \cdot c_{\theta 2} \]  

(2-6)

\[ r_2 \cdot c_{\theta 2} = \frac{\Delta p_s}{\rho \cdot \eta \cdot \omega} \]  

(2-7)

As shown in Equation (2-5) and Equation (2-6), the distributions of enthalpy from hub to shroud at outlet is determined only by radius in the blades flow passages.

### 2.2 Solution of meridional flow

In calculation of meridional flow, the following force balance equation according to Novak’s [123] streamline curvature computing procedures was evaluated at the quasi orthogonal direction on meridional plane:

\[ \frac{dC_{m}}{dq}^2 + A(q) \cdot C_{n}^2 = B(q) \]  

(2-8)

Here the compressibility of the fluid is ignored.
\[ A(q) = 2 \cdot \left\{ \frac{1}{r_m \cdot \cos \varepsilon} + \frac{\sin \varphi \cdot \sin\varepsilon}{r} + \frac{d\varphi}{dq} \cdot \tan \varepsilon \right\} \]  
\[ (2-9) \]

\[ B(q) = 2 \cdot \left\{ \frac{1}{\rho \cdot dq} \cdot \frac{\frac{dP_t}{dq} - C_{\theta} \cdot d(rC_m)}{r} \right\} \]  
\[ (2-10) \]

\( C_m \) and \( C_{\theta} \) is the velocity in meridional and circumferential directions. \( q \) is the quasi orthogonal direction which is the same with the leading edge of the blade at inlet. \( \varphi \) is the streamline angle with the rotating axis. \( \varepsilon \) is the angle of \( q \) line with the normal direction of streamline. \( P_t \) is the total pressure. \( \rho \) is the density of working medium (air). Equation (2-8) can be solved with the following equation of the total flow rate which is given as:

\[ G = 2\pi k_B \int_{q_i}^{q_f} \rho r C_m \cos \varepsilon dq \]  
\[ (2-11) \]

\( k_B \) is the blade blockage coefficient which is taken as 0.96. \( q_c \) and \( q_h \) are the radii of casing and hub in \( q \) direction, respectively. In this dissertation, circumferential velocity is constant in radial direction that is the \( q \) direction either. The energy getting though rotor outlet at unit time can be evaluated as following:

\[ E_z = \frac{G \cdot \Delta l_{th}}{k_B} = 2\pi \int_{q_i}^{q_f} \rho \cdot r_c \cdot C_{m2} \cdot \cos \varepsilon \cdot u_z \cdot C_{\theta2} dq \]  
\[ (2-12) \]

According to the assumption that the total pressure is equal to static pressure rise in section 2.1, the total pressure distribution along \( q \) line can be expressed by the substitution of Equation (2-6):

\[ \frac{1}{\rho} \cdot \frac{dP_t}{dq} = \frac{1}{\rho} \cdot \frac{d\rho}{dr} = \frac{1}{\rho} \cdot \frac{d(\eta \cdot \rho \cdot r \cdot \omega \cdot C_{\theta})}{dr} = \eta \cdot \omega \cdot C_{\theta} \]  
\[ (2-13) \]

Therefore,

\[ B(q) = 2 \cdot \left\{ \eta \cdot \omega \cdot C_{\theta} \cdot \frac{C_m^2}{r} \cdot dq \right\} \]  
\[ (2-14) \]

In the meridional calculation, the streamlines on hub and tip are prescribed in splines and the spline inclinations are based on the experimental data. According to specified enthalpy and circumferential velocity along every \( q \) line before and behind the blades described in section 2.1, the meridional velocity can be obtained.
According to Lieblein [124] in the loss analysis of compressor cascades, the total pressure loss coefficient of the cascade under the effects of the streamline inclination and the variation of meridional velocity is given as:

$$\zeta_p = \frac{0.008}{1 - 1.17 \ln D_{eq}} \left( \frac{L}{t_m} \left( \frac{r_1 b_1 \cos \beta_1}{r_2 b_2 \cos \beta_2} \right)^2 \right)$$

(2-15)

Where, $D_{eq}$ is the equivalent diffusion factor defined as:

$$D_{eq} = \frac{\cos \beta_2}{\cos \beta_1} \frac{C_{n1}}{C_{n2}} \left[ 1.12 + 0.61 \left( \cos ^2 \beta_1 \left( \tan \beta_1 - \frac{r_2 C_{w2}}{r_1 C_{w1}} \tan \beta_2 - \frac{r_2 \omega}{C_{w1}} \left( 1 - \frac{r_2^2}{r_1^2} \right) \right) \right) \right]$$

(2-16)

The subscripts 1 and 2 denote the inlet and outlet of blades. $r$ and $b$ are radius and the thickness of revolutionary stream surface. $l$ is the chord of blade. $t_m$ is one pitch length on the average radius $r_m = 0.5(r_1 + r_2)$. $\beta$ and $\omega$ are flow angle and angular velocity.

### 2.3 Blade selection on the average stream surface

The meridional flow has been decided in the above section, the blade to blade flow on revolving stream surface will be discussed and blade element on average stream surface will be selected in this section. According to the flow characteristics of the propeller fan, the design method of the diagonal flow fan extended from the axial flow fan is adopted. Rotor of the axial flow fan carries the fluid through the blades on the cylindrical surface. The streamline getting through the rotor and rotating along the axis makes into a surface called the stream surface. The blade having little thickness on this surface is the blade element. When the stream surface is cylindrical, the blade element can be chosen on the opened cylindrical surface, and the three dimensional blade can be obtain by piling it up in span direction.

Therefore, the characteristic of the 2D blade element has a great effect on the performance of rotor. The investigation of two dimensional cascade is one of important aerodynamic topics and many data have been obtained from the wind tunnel test by predecessors. The cascade theory and the experimental data have been validated and can be used to design the rotor satisfying the specification requirement as long as the hub ratio is not extremely small. In order to use the cascade data, the through flow in cylindrical or cone surface must be considered on an averaged stream.
surface. Therefore, two conditions have to be pointed out before the solution. First, the flow is assumed to be axial symmetry. Second, the physical parameters must to be averaged in circumferential direction. The stream surface of revolution considering declination has been presented in Figure 2-1. And the velocity triangle is shown on $m$-$r$-$\theta$ plane in which $C$ and $W$ are absolute velocity and relative velocity respectively. The subscripts mean the components of them in that directions. The flow on revolving stream surface is projected to $XY$ plane in the following function:

\[
\frac{dX}{dm} = \frac{r^*}{r}, \quad \frac{dY}{d\theta} = -r^*
\]

$r^*$ is arbitrary radius for reference radius. $X$ is axial flow direction and $Y$ is blade rotating direction on revolving stream surface. Using this function, the flow on revolving stream surface can be projected on linear cascade. Relative axial velocity $W_X$ and velocity in rotating direction $W_Y$ on linear cascade are expressed as follows:

\[
W_X = \frac{r}{r^*} \cdot C_m, \quad W_Y = \frac{r}{r^*} (u - C_m)
\]

Local theoretical pressure coefficient and flow coefficient are defined as follows:

\[
\Psi_{th} = \frac{(u_2 C_{m2} - u_1 C_{m1})}{(u^*^2 / 2)} = 2\Phi \left[ 2(W_{x1} - W_{x2})/(W_{x1} + W_{x2}) + \chi \right]
\]

\[
\Phi = (r_1 W_{m1} + r_2 W_{m2}) / 2r^* u^* = (W_{x1} + W_{x2}) / 2u^*
\]

Here, $u^* = r^* \omega$ is circumferential velocity on the reference plane. The subscripts 1 and 2 denote the rotor inlet and outlet. $\chi$ is the parameter of stream surface declination, and it is defined as:

\[
\chi = (r_2^2 - r_1^2) / (4r^2)
\]

If these parameters are known, the velocity triangle at blade inlet and outlet can be drawn and then blade element selection could be conducted. For the propeller fan design in this dissertation, two methods are considered to perform on the blade element selection.
2.3.1 Equivalent velocity triangle method

Figure 2-2 presents the velocity triangles at blade inlet and outlet on XY plane. In order to use linear cascade data, averaged velocity in axial direction is taken as \( W_{x0} = (W_{x1} + W_{x2})/2 \) and velocity in \( Y \) direction does not change. On equivalent velocity triangle, the flow angles have the following relationship:

\[
\tan \beta_1 - \tan \beta_2 = \psi/2\Phi - \chi
\]

\[\tan \beta_1 = W_{y1}/W_{x0} = r_i (u_i - C_{in})/(\Phi \cdot r' \cdot u^*) \]  

If stagger angle and blade number are \( \gamma \) and \( Z_b \), respectively. Cascade solidity \( \sigma_{XY} \) on XY plane can be given as,

\[
\sigma_{XY} = Z_i (X_2 - X_1)/(2\pi \cdot r' \cdot \cos \gamma)
\]  

According to the above three functions, blade element on XY plane can be selected from linear cascade data. However, the obtained two dimensional cascade has to be corrected to satisfy design point because of the effect from declination and thickness of stream surface. Therefore, the singular point method is used to make the correction.

The singular point method is using to solve Laplace's equations of potential function and stream function. Let singular points of vortices and divergence distribute in the flow field and they will produce the velocity. The produced velocity can be decided by the intensity of the vortices and divergence which satisfies the boundary condition of solid surface. The flow along the blade surface

![Fig.2-1 Revolution surface (m-rθ)](image-url)
will be obtained by the velocity composition of averaged flow velocity and produced velocity. In
the linear cascade, the camber and thickness of blade are assumed to be evaluated by the flow field
of the resultant velocity. Generally, there are two representative methods to solve the distributions
of vortices and divergence on the blade surface: Schlichting method and vortex point method. In
Schlichting method, the camber line follows the tangential direction of resultant velocity and the
variation of blade thickness satisfies the local continuity condition according to the boundary
condition of thin blade. While for the vortex point method, the velocity along the blade surface is
presented by the function of blade profile and solved in the boundary layer of the blade.

The effect of declination and thickness of stream surface are considered into the distributions of
cambrer and thickness of blade are assumed to be evaluated by the flow field of the resultant velocity. Generally, there are two representative methods to solve the distributions of vortices and divergence on the blade surface: Schlichting method and vortex point method. In

Schlichting method, the camber line follows the tangential direction of resultant velocity and the
variation of blade thickness satisfies the local continuity condition according to the boundary
condition of thin blade. While for the vortex point method, the velocity along the blade surface is
presented by the function of blade profile and solved in the boundary layer of the blade.

The effect of declination and thickness of stream surface are considered into the distributions of
vortex \( \zeta = \text{rot}(W_x, W_y) \) and divergence \( \mu = \text{div}(W_x, W_y) \) on \( XY \) plane respectively. Figure 2-3 presents
the distributions of vortex and divergence, deduced velocity at inlet and outlet of cascade. The
circulation of the dashed closed line on left side in Figure 2-3, which is consist of blade leading and
trailing edge, two streamlines with one pitch distance, can be expressed:

\[
t \cdot W_{xe} (\tan \beta_i - \tan \beta_l) = \Gamma_0 + \Delta \Gamma + \int_{Y1}^{Y2} (-\zeta) dXdY
\]

\( \Gamma_0 \) is blade circulation when there are no vortex and divergence. \( \Delta \Gamma \) is the variation of blade
circulation. After calculation of the integral on right side of above equation, it can be written as
follows:

\[
\tan \beta_i - \tan \beta_l = \frac{\Gamma_0}{tW_{xe}} + \frac{\Delta \Gamma}{tW_{xe}} - \chi
\]

(2-26)

The only difference of Equation (2-26) with Equation (2-22) is the second and third items on
right side. If using Equation (2-22)-(2-24) to make blade selection from two dimensional cascade
data, it has to make the right side of Equation (2-26) and Equation (2-22) same to correct the blade
shape.
2.3.2 Imaginary velocity triangle method

According to Equation (2-26) derived from Equation (2-22), pressure rise of diagonal cascade is caused by the difference between turning quantity of relative flow \((\tan \beta_1 - \tan \beta_2)\) and centrifugal effect of stream surface declination \(\chi\). When the stream surface declination is larger than blade loading, that is \(\psi_0/2\phi < \chi\), the cascade data of speed increase is necessary for design. However, there
are almost no very suitable speed increase cascade data for the blade shape of fans. Therefore, the imaginary velocity triangle is considering for the application of speed reduction cascade data in this situation.

$$\Psi_{a_b} = 2\Phi (\tan\overline{\beta}_1 - \tan\overline{\beta}_2 + \chi)$$  \hspace{1cm} (2-27)

As shown in Figure 2-3, if the vortex \( \zeta = 2\alpha (r/r')^3 \, dr/dm \) is distributing on XY plane, the velocity in Y direction \( \pm 0.5 \chi W_{\infty} \) at inlet and outlet of cascade is deduced. While the velocity in X direction \( \mp 0.5 \Delta W_x \) is deduced by the divergence distribution between two blade. For excluding the deduced velocity from vortex, the imaginary velocity is considered in this section. The same with this method, for excluding the deduced velocity from thickness variation of stream surface, the equivalent velocity is made for selecting blade element as explained in section 2.3.1.

Figure 2-4 shows imaginary velocity and equivalent imaginary velocity triangles in cascade. At inlet and outlet of cascade, absolute velocity is \( C_1 \) and \( C_2 \), rotating speed of rotor is \( u_1 \) and \( u_2 \). Because of \( u_2 - u_1 = \chi W_{\infty} \), the circumferential velocity is \((u_1 + u_2)/2\) in imaginary velocity triangles in which the relative velocity is shown in dashed lines. Therefore, the imaginary velocity is relative velocity which is corresponding to circumferential velocity at averaged radius of inlet and outlet in diagonal flow cascade. Due to the imaginary velocity in axial direction is changeable, the equivalent imaginary velocity in axial direction is adopted to calculate the flow angles in the same consideration with equivalent velocity theory mentioned at above section,

$$\tan\overline{\beta}_1 = \tan\overline{\beta}_2 + \chi / 2, \quad \tan\overline{\beta}_2 = \tan\overline{\beta}_1 - \chi / 2$$ \hspace{1cm} (2-28)

$$\tan\hat{\beta}_1 - \tan\hat{\beta}_2 = \Psi_{a_b} / 2\Phi > 0$$ \hspace{1cm} (2-29)

Therefore, cascade data of speed reduction is suitable in this situation,

$$\tan\hat{\beta}_1 - \tan\hat{\beta}_2 = \frac{\Gamma_0 + \Delta\Gamma}{\iota W_{\infty}}$$ \hspace{1cm} (2-30)

When use two dimensional cascade data to make blade element selection, the performance change of diagonal flow cascade is just decided by the circulation variation \( \Delta\Gamma \) on the right side of Equation (2-30). Therefore, it has to make the right side of Equation (2-29) and Equation (2-30) equal to correct the blade shape.
Actually for the blade element selection based on real velocity triangle, existing vortex in relative velocity flow field produces slip making the turning angle decrease. However, turning angle is enlarged in blade element selection of imaginary velocity method due to the neglect of head rise from centrifugal force effect. In the case of blade element selection of imaginary velocity method, it is necessary to carry out experimental correction due to over high evaluation of viscosity effect.

2.3.3 Expansion of blade element selection theory

In section 2.3.1 and section 2.3.2, equivalent velocity method and imaginary velocity method are discussed as two methods of blade element selection. They are derived from the same function but the selected blade elements are different in two dimensional non-rotating flow field. However, the selected blade elements would be the same blade shape if their correction of blade element selection are conducted only in the way of neglecting viscosity effect. That is to say, the only distinction of the two methods is different initial value for the correction calculation of cascade shape. Therefore, it is possible to give arbitrary initial value by mixing geometry shape of two methods in proportion. Then, the flow angle of imaginary velocity can be expressed by Equation (2-31).
\[ \tan \hat{\beta}_1 = \tan \bar{\beta}_1 + k\chi / 2, \quad \tan \hat{\beta}_2 = \tan \bar{\beta}_2 - k\chi / 2 \]  

(2-31)

Here, \( k \) is arbitrary constant, therefore,

\[ \tan \hat{\beta}_1 - \tan \hat{\beta}_2 = \Psi_{in}/2\Phi - (1-k)\chi \]  

(2-32)

\[ \tan \hat{\beta}_1 - \tan \hat{\beta}_2 = \frac{\Gamma_0 + \Delta \Gamma}{i\dot{W}_{two}} - (1-k)\chi \]  

(2-33)

It will be the expansion method of blade element selection to use Equation (2-31) to make blade element selection from cascade data, and carry out cascade shape correction by making the right side of Equation (2-32) and Equation (2-33) equal. When \( k=0 \) and \( k=1 \), they are corresponding to equivalent velocity triangle method and imaginary velocity triangle method respectively. Generally, the value of \( k \) is in the range of \((0, 1)\).

\[ k = 1 - \frac{\Delta \Gamma}{i\dot{W}_{two}} \]  

(2-34)

If the above equation is satisfied, the correction calculation is not necessary. However, it is necessary to conduct the calculation repeatedly, and when

\[ k = 1 - \frac{\Psi_{in}}{2\Phi \chi} \]  

(2-35)

The cascade data of zero turning angle can be in application. However, the above blade element selection theory is inviscid flow. Viscosity effect in cascade is relative to speed reduction ratio of relative velocity, the larger speed reduction ratio \( k\chi \) is in case of imaginary velocity method, the more difference it is far away from the one in real. Best performance condition in test of designed fans by above theory are different with design point. So it is necessary to evaluate the change of circulation parameter \( f \) by the following equations in case of imaginary velocity method:

\[ f = \frac{\tan \bar{\beta}_1 - \tan \bar{\beta}_2}{\Psi_{in}/2\Phi - \chi} \]  

(2-36)

\[ f \approx 0.1k\chi \]  

(2-37)

### 2.4 Correction on blade geometry

Three dimensional effect of declination and thickness variation of stream surface on flow field is substituted by the distributions of vortex and divergence on \( XY \) plane respectively. Thus, if blade element selection is conducted on \( XY \) plane by using linear cascade of two dimensional non-rotating
flow field, the cascade performance is changed by the effect of the vortex and divergence. Therefore, the correction for cascade shape has to be carried out and need repeated calculation by the application of the potential theory. Moreover, Schlichting’s singular point method is used in calculation. In order to suit three methods explained in section 2.3.1-2.3.3, $\beta_1$ and $\beta_2$ in Equation (2-31) and $\sigma_{xy}$ in Equation (2-24) are given to make blade selection. The deduced velocity caused by vortex $\dot{\zeta} = \text{rot}(W_x, W_y) = 2\omega \frac{r}{r^2 + \rho^2} \frac{dr}{dm}$ is

$$V_{x0} = \frac{u}{r} \left[ \frac{r^2 - \rho^2}{2r^2} \right]$$

(2-38)

And the deduced velocity caused by divergence $\dot{\mu} = \text{div}(W_x, W_y)$ is

$$V_{ax} = \frac{1}{2\pi} \int_0^\pi \int_0^{2\pi} \frac{\rho(X', Y') \sinh \tilde{X} \cosh \tilde{Y} - \cosh \tilde{X} \sinh \tilde{Y}}{\cosh \tilde{X} - \cos \tilde{Y}} dY' dX'$$

$$V_{ay} = \frac{1}{2\pi} \int_0^\pi \int_0^{2\pi} \frac{\rho(X', Y') \sinh \tilde{X} \cosh \tilde{Y} - \sinh \tilde{X} \cosh \tilde{Y}}{\cosh \tilde{X} - \cos \tilde{Y}} dY' dX'$$

(2-39)

Here, $\tilde{X} = 2\pi(X - X')/t$, $\tilde{Y} = 2\pi(Y - Y')/t$, $d$ is blade thickness measured in $Y$ direction. The right side of above equation cannot be integrated due to unknown $W_x$ and $W_y$. Thus, no divergence effect on $Y$ direction is assumed according to the first order approximation and the deduced velocity is evaluated from one dimensional continued function. That is, in the cases of divergence exists or not, the averaged axial velocities are respectively,

$$\bar{W}_x = \frac{b \rho}{k b \rho} W_{x0}$$

(2-40)

$$\bar{W}_{x0} = \frac{1}{k} W_{x0}$$

(2-41)

Here, the blockage coefficient $k_{bg} = (t - d)/t$ considering the effect of blade thickness. Deduced velocity of the first order approximation is

$$V_{\mu x} = \bar{W}_x - \bar{W}_{x0} = \frac{1}{k} \left[ \frac{b \rho}{b \rho} (1 - \frac{\xi}{2}) - 1 \right] W_{x0}$$

(2-42)

$$V_{\mu y} = 0$$

Here, the vary ratio of axial velocity $\xi = (W_{x2} - W_{x1})/W_{x0}$.

If to find the solution in high order approximation, the integral has to be conducted in Equation (2-39) until $V_{\mu x}$ and $V_{\mu y}$ get convergence in repeat calculation. As a matter of fact, it is enough to
use the solution in the first order approximation in design problems. In order to make the flow field full of vortex and divergence suitable for Schlichting’s singular point method, the suit conditions of blade and flow are expressed as follows,

\[
\begin{align*}
\frac{dy_c}{dx} &= \frac{W_{c_\alpha} + W_{c_\beta} + V_x}{W_{c_\alpha} + W_{c_\beta} + V_x} \\
\frac{dy_d}{dx} &= \frac{\hat{q}_b}{W_{c_\alpha} + W_{c_\beta} + V_x}
\end{align*}
\] (2-43)

Here, \(x\) is in blade chord direction and \(y\) direction is perpendicular to \(x\). Subscripts of them are the components in that directions. \(y_c\) and \(y_d\) are blade camber and thickness. \(W_b\) is the deduced velocity of fixed vortex and divergence distributing on blade chord, and \(\hat{q}_b\) is intensity of fixed divergence.

The deduced velocity caused by vortex and divergence distributing in the flow field are

\[
\begin{align*}
V_x &= (V_{c_\alpha} + V_{c_\beta}) \cos \alpha + (V_{a_\alpha} + V_{a_\beta}) \sin \alpha \\
V_y &= -(V_{c_\alpha} + V_{c_\beta}) \sin \alpha + (V_{a_\alpha} + V_{a_\beta}) \cos \alpha
\end{align*}
\] (2-44)

As the vortex and divergence distributing on blade chord have Glauert series, see Glauert series coefficient as unknown number, Equations (2-43) become first order simultaneous equations. The numbers of series term equal to coefficient of \(x\) with the same order in Equations (2-43) to evaluate unknown coefficient. Circulation around blade is

\[
\Gamma_b = 0.5 \pi W_c \cos \alpha_s [(A_{00} + 0.5 A_{01}) + (A_{00} + 0.5 A_{01}) \tan \alpha_s]
\] (2-45)

Here, \(A_{00}, A_{10}, A_{01}(A_{11})\) is Glauert series coefficient of fixed vortex on blade chord. \(\alpha_s\) is attack angle of averaged velocity vector. Thus,

\[
\tan \beta_1 - \tan \beta_2 = \frac{\sigma_{y1} \cos \alpha_s}{2 \cos \beta_s} [(A_{00} + 0.5 A_{01}) + (A_{00} + 0.5 A_{01}) \tan \alpha_s] - \chi
\] (2-46)

Which is corresponding to Equation (2-26). \(\alpha_s = \beta_1 - \gamma, \quad \tan \beta_s = 0.5(\tan \beta_1 + \tan \beta_2)\) and together with Equation (2-24), the above equation will become

\[
\tan \beta_1 - \tan \beta_2 = \frac{\pi}{2} \sigma_{y1} (S_d + T_d \tan \beta_s) - (1-k) \chi
\]

Here, \(S_d = (A_{00} + 0.5 A_{01}) \cos \gamma - (A_{00} + 0.5 A_{01}) \sin \gamma\)

\[T_d = (A_{00} + 0.5 A_{01}) \sin \gamma + (A_{00} + 0.5 A_{01}) \cos \gamma\]

Based on above equation, blade camber is corrected as the following steps:
Firstly, when there are no vortex and divergence distributing in flow field, $V_x=0$, $V_y=0$, $\chi=0$ and $\hat{f}_0$ equal to the value of Equation (2-46).

Then, giving the values of $V_x$, $V_y$, $\chi$ and $k$, solve cascade flow and the value of Equation (2-46) equal to $\hat{f}_0$.

Finally, when $\hat{f} \neq \hat{f}_0$, the change of turning angle is given as $\delta(\Delta \beta) = (\hat{f} - \hat{f}_0) \frac{\partial(\Delta \beta)}{\partial \hat{f}}$, which is just equal to the change of blade camber angle, that is

$$
\delta \hat{\theta} = -\delta(\Delta \beta) = -0.5(\hat{f} - \hat{f}_0)\left[\frac{1}{(\tan \beta_x + 0.5 \hat{f})^2 + 1} + \frac{1}{(\tan \beta_x - 0.5 \hat{f})^2 + 1}\right].
$$

(2-47)

Just let camber angle change and solve cascade flow to get $\hat{f}$. The calculation need conducted repeatedly until $\hat{f} = \hat{f}_0$.

Blade camber line is corrected by above mentioned method. If blade thickness adds to the blade camber line, the blade shape on $XY$ plane will be obtained. Then, project it to physical plane as expressed in Equation (2-18), the blade shape on revolution stream surface will be obtained.
Chapter 3
Numerical Theory and Procedures

3.1 Computational procedures

Numerical simulation has been used to check the performance characteristics, internal flow field and dynamic noise of studied propeller fans. The steady flow field of a ducted designed propeller fan and a half-ducted designed propeller fan are simulated using turbulence model of RNG $k-\varepsilon$ in commercial software Fluent 6.3 and Fluent 14.5. Through the comparison of calculated results with design data for the two designed propeller fans, it is known that the half-ducted designed propeller fan has better dynamic performance. Therefore, the half-ducted design method is applied to design a new propeller fan and the design parameters are modified according to the numerical results. Then, the new designed propeller fan is checked in the calculation with the same approach above and as a result, it is a better design than prior ones. Finally, the last designed propeller fan is made by three dimensional printer. Moreover, its performance characteristics and steady flow field are discussed by RNG $k-\varepsilon$ turbulence model and its dynamic noise is calculated by Large Eddy Simulation (LES).

3.2 Control volume technique

Control volume technique is used in Fluent to convert a general scalar transport equation to an algebraic equation that can be solved in computer. Using the control volume approach, the conservation principle can be expressed as that the quantity generation inside the control volume and the flux of a quantity through the control volume surface must equal to the time accumulation of that quantity within the volume. In each control volume as shown in Figure 3-1, $dA$ is a pointing outwards vector normal to a small area on surface A. Transport equations are integrated to yield a discrete equation that express the conservation law on a control volume.

$$\frac{d}{dt} \int \phi dV = -\oint (\vec{F} + \vec{\phi} \cdot \vec{v}) \cdot d\vec{A} + \int S dV$$

(3-1)

$\vec{v}$ and $\vec{A}$ are velocity and surface area vector. $\vec{F}$ and $S$ are flux and per unit volume source of
scalar quantity $\phi$. The negative sign means that a positive outward flux corresponds to a negative rate change of the integral in the left side. Using Gauss's theorem, the closed surface integrals in above equation can be replaced as:

$$\oint (\vec{F} + \phi \vec{n}) \cdot d\vec{A} = -\int \nabla \cdot (\vec{F} + \phi \vec{n}) dV$$  \hspace{1cm} (3-2)

Furthermore, the sum of the local time derivative can instead of the time derivative because the surface and volume are fixed in an inertial frame.

$$\frac{d}{dt} \int_V \phi dV = \frac{\partial \phi}{\partial t} dV$$ \hspace{1cm} (3-3)

Substituting above two equations into Equation (3-1) yields

$$\int \left( \frac{\partial \phi}{\partial t} + \nabla \cdot (\vec{F} + \phi \vec{n}) - S_\phi \right) dV = 0$$ \hspace{1cm} (3-4)

The volume $V$ is arbitrary shape and size, therefore,

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\vec{F} + \phi \vec{n}) - S_\phi = 0$$ \hspace{1cm} (3-5)

This is the general form of conservation laws.

In Fluent, discretization of the governing equations can be expressed by the unsteady conservation equation for transport of a scalar quantity $\phi$, which can be written in integral form for an arbitrary control volume $V$:

$$\int \frac{\partial \rho \phi}{\partial t} dV + \int \rho \phi \vec{v} \cdot d\vec{A} = \int \Gamma \nabla \phi \cdot d\vec{A} + \int S_\phi dV$$ \hspace{1cm} (3-6)

$\Gamma$, $\nabla \phi$ and $S_\phi$ are diffusion coefficient, gradient and per unit volume source of scalar quantity $\phi$. 

![Control volume](image-url) Fig.3-1 Control volume
3.3 Basic conservation laws

3.3.1 Multiple rotating frame

The moving reference frame are used in the flow around the rotor which can be modeled as a steady state problem with respect to the moving frame. Before discussing the conservation equations, the moving reference frame used in this dissertation will be explained.

The Multiple rotating Frame (MRF) was used to separate the moving zone of the rotor and stationary zones around it. Governing equations in each subdomain are expressed with respect to the reference frame in that subdomain. Steady state flow conditions are assumed at the interface between the two reference frames and the absolute velocity at the interface are the same. The meshes at interface are not slide. The scalar quantities such as pressure, density, temperature, etc. are the same and obtained from adjacent cells. The relative velocity are computed relative to the motion of rotor. Velocity and its gradients are converted from a moving reference frame to the inertial frame using the following equations:

\[
\bar{v} = \bar{v}_{re} + (\bar{\omega} \times \bar{r}) + \bar{v}_{tra} \quad (3-8)
\]

\[
\nabla \bar{v} = \nabla \bar{v}_{re} + \nabla (\bar{\omega} \times \bar{r}) \quad (3-7)
\]

\(\bar{v}\) is absolute velocity, and the subscripts re and tra mean relative velocity and translational velocity respectively.

3.3.2 Conservation equations for a rotating reference frame

In the zone set rotating reference frame, it rotates steadily with angular velocity. The angular velocity \(\bar{\omega} = \omega \times \bar{r}\) is relative to the inertial reference frame and located by a position vector \(\bar{r}\). The velocity can be transformed from the inertial frame to the rotating frame using the following equation:

\[
\bar{v}_{re} = \bar{v} - \bar{u}_{re} = \bar{v} - \bar{\omega} \times \bar{r} \quad (3-9)
\]

\(\bar{u}_{re}\) is the whirl velocity due to the moving frame.

In this dissertation, Fluent’s pressure-based solvers and relative coordinate system are chosen so
that the governing equations of mass, momentum and energy conservation for relative velocity formulation in a steadily rotating frame are provided below.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}_r) = 0
\]

\[
\frac{\partial (\rho \vec{v}_r)}{\partial t} + \nabla \cdot (\rho \vec{v}_r \vec{v}_r) + \rho (2\omega \times \vec{v}_r + \ddot{\omega} \times \vec{v}_r) = -\nabla p + \nabla \tau + \vec{F}
\]

\[
\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho \vec{v}_r H) = \nabla \cdot (k \nabla T + \tau \cdot \vec{v}_r) + S_b
\]

\( \tau \) is the stress tensor

\[
\tau = \mu [\nabla \vec{v} + (\nabla \vec{v})^T] - \frac{2}{3} \nabla \cdot \vec{v} I
\]

\( \mu \) is molecular viscosity, \( I \) is unit tensor, and the second term on the right side is the effect of volume dilation. Two additional acceleration terms, the Coriolis acceleration \( 2\ddot{\omega} \times \vec{v}_r \) and centripetal acceleration \( \ddot{\omega} \times \ddot{\omega} \times \vec{r} \), are included in the momentum equation. The energy equation is presented in terms of the relative total enthalpy (rothalpy) and the relative internal energy, which are defined as following respectively:

\[
H_{re} = E_{re} + \frac{p}{\rho}
\]

\[
E_{re} = h - \frac{p}{\rho} \frac{1}{2}(v_{re}^2 - u_{re}^2)
\]

### 3.4 Turbulence models and acoustics model

#### 3.4.1 RNG \( k-\epsilon \) model

RNG \( k-\epsilon \) model has been used to calculate the steady flow field of studied propeller fans. The Reynolds averaged turbulence model of RNG \( k-\epsilon \) is derived from the instantaneous Navier-Stokes equations using a mathematical technique called renormalization group (RNG). The method is got refinements from standard \( k-\epsilon \) model. It adds a term in \( \epsilon \) equation to improve the accuracy for rapidly strain flow and the effects of swirl flow and low Reynolds number are included either. An analytical formula for turbulent Prandtl Numbers is also provided in RNG \( k-\epsilon \) model.

The transport equations of RNG \( k-\epsilon \) are

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \alpha_t \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_i + G_t - \rho \epsilon - Y_M + S_k
\]

(3-16)
\[
\frac{\partial\rho c}{\partial t} + \frac{\partial (\rho u c)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \alpha_s \mu_{eff} \frac{\partial c}{\partial x_i} \right) + C_{te} (G_k + C_s G_b) - C_{2e} \rho \frac{e^2}{k} - R_e + S_e
\]

(3-17)

Here, \(G_k\) and \(G_b\) represents the generation of turbulence kinetic energies due to mean velocity gradients and buoyancy respectively. \(Y_M\) is the fluctuating dilatation contributed to the overall dissipation rate in compressible turbulence. \(\alpha_t\) and \(\alpha_e\) are Prandtl number inversely effecting on \(k\) and \(e\), respectively. \(S_k\) and \(S_e\) are user defined source terms. \(C_{1e}\) and \(C_{2e}\) are model constants derived by the RNG theory which \(C_{1e} = 1.42\) and \(C_{2e} = 1.68\) are used by default in Fluent.

The effective viscosity is modeled by a differential equation resulting from the scale elimination procedure in RNG theory:

\[
d\left(\frac{\rho^2 k}{\sqrt{\mu}}\right) = 1.72 \frac{\tilde{E}}{\sqrt{\tilde{E} + C_t}} d\tilde{E}
\]

(3-18)

Where, \(\tilde{E} = \mu_{eff}/\mu\)

\[C_t \approx 100\]

Integrating Equation (3-18) accurately describes the turbulent transport varies with Reynolds Number which is better to handle low Reynolds Number and near wall flows. When it is in high Reynolds Number situation, Equation (3-18) yields

\[\mu_t = C_{\mu} \rho \frac{k^2}{\varepsilon}\]

(3-19)

Here, \(C_{\mu} = 0.0845\) derived using RNG theory. It is very close to empirical value 0.09 in the standard \(k-\varepsilon\) model. The effective viscosity is computed by Equation (3-19) by default in Fluent. However, the differential relation in Equation (3-18) is an available option when low Reynolds Number effects is needed.

In general, turbulence is affected by rotating or swirl in the mean flow. RNG model gets the effects of swirl or rotation by modifying the turbulent viscosity appropriately. It takes the following function:

\[\mu_t = \mu_{ref} f(\alpha_s, \Omega, \frac{L_a}{L_s})\]

(3-20)

\(\mu_{ref}\) is turbulent viscosity value without the swirl modification calculated by Equation (3-18) or Equation (3-19). \(\Omega\) is a characteristic swirl number in Fluent. \(\alpha_s\) is a swirl constant which depends
on the swirl flow density. The default swirl is mildly swirling flows with $\alpha_s=0.07$. The stronger the swirl flow is, the higher the value is. The swirl modification of RNG model always takes effect for axisymmetric, swirl flow and three dimensional flow.

The inverse effective Prandtl Numbers $\alpha_i$ and $\alpha_e$ are computed by the following formula:

$$
\begin{vmatrix}
\alpha - 1.3929 \\
\alpha_i - 1.3929
\end{vmatrix}^{0.6321} = \frac{\mu_{mod}}{\mu_{eff}}
\begin{vmatrix}
\alpha + 2.3929 \\
\alpha_e + 2.3929
\end{vmatrix}^{0.3679} = \mu_{mod}
$$

(3-21)

Where $\alpha_0=1.0$, in the case of high Reynolds number ($\mu_{mod}/\mu_{eff} << 1$), $\alpha_i = \alpha_e \approx 1.393$. The additional term in $\varepsilon$ equation is

$$
R_{\varepsilon} = \frac{C_{\mu} \eta^{1/2}(1-\varepsilon/\varepsilon_h)}{1+\beta \eta}
$$

(3-22)

Where, $\eta = S_k/\varepsilon$, $\varepsilon_h = 4.38$, $\beta = 0.012$, use this equation to rearrange Equation (3-17), then the $\varepsilon$ equation can be rewritten as

$$
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_j} (\alpha_s \mu_{eff} \frac{\partial \varepsilon}{\partial x_j}) + C_{\mu_s} \frac{\varepsilon}{k} (G_i + C_{\mu_s} G_h) - C_{\mu_s} \beta \frac{\rho^{1/2}}{k}
$$

(3-23)

Here,

$$
C_{\mu_s} = C_{\mu_s} + \frac{C_{\mu_s} \eta^{1/2}(1-\varepsilon/\varepsilon_h)}{1+\beta \eta}
$$

(3-24)

The $R$ term gives a positive contribution when in the region of $\eta < \eta_0$, and $C_{\mu_s}$ is larger than $C_{2\varepsilon}$. For example, in the logarithmic layer $\eta \approx 3.0$ gives $C_{\mu_s} \approx 2.0$ which is close to the value of $C_{2\varepsilon}$ in the standard $k-\varepsilon$ model. While the $R$ term gives a negative contribution when $\eta > \eta_0$ in regions of large strain rate, and $C_{\mu_s}$ is less than $C_{2\varepsilon}$. The RNG model yields a lower turbulent viscosity than the standard $k-\varepsilon$ model. Thus, the RNG model is more responsive to the effects of rapid strain and streamline curvature than the standard $k-\varepsilon$ model.

3.4.2 LES model

Turbulent flows are featured by eddies of which the largest scales are comparable in size to the characteristic length of the mean flow and the smallest scales contributes to the dissipation of
turbulence kinetic energy. Direct numerical simulation (DNS) is an approach to resolve the whole spectrum of turbulent scales directly in theory. However, the approach of DNS meets some problems in practical engineering involving high Reynolds number flows. Large eddy simulation (LES) falls between DNS and Reynolds averaged Navier-Stokes (RANS), in which small eddies are modeled and large eddies are resolved directly.

Time dependent Navier-Stokes equations are filtered in Fourier space or configuration space to obtain the governing equations of LES. In filtering process, the eddies smaller than the filter width or grid spacing used in the computations are filtered out. Thus the dynamics of large eddies are governed by these equations.

A filtered variable is defined as:

\[ \tilde{\phi}(x) = \int \phi(x') G(x,x') dx', \quad x' \in V \]  

(3-25)

Where \( V \) is the volume of computational fluid domain. \( G \) is the filter function to determine resolved eddies scale and it is

\[ G(x,x') = \begin{cases} 
\frac{1}{V} & x' \in V \\
0 & \text{otherwise}
\end{cases} \]  

(3-26)

For incompressible flows, filtered Navier-Stokes equations can expressed as follows:

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho \bar{u}_i)}{\partial x_j} = 0 \]  

(3-27)

\[ \frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = \frac{\partial (\sigma_{ij})}{\partial x_j} - \frac{\partial p}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} \]  

(3-28)

Where \( \sigma_{ij} \) and \( \tau_{ij} \) are the stress tensor due to viscosity and the subgrid-scale stress and defined as:

\[ \sigma_{ij} = \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial \bar{u}_k}{\partial x_k} \delta_{ij} \]  

(3-29)

\[ \tau_{ij} = \rho u_i u_j - \rho \bar{u}_i \bar{u}_j \]  

(3-30)

The filtering operation results in subgrid-scale stresses which are unknown and require modeling. In Ansys Fluent, the Boussinesq hypothesis is employed to get subgrid-scale turbulent stresses.

\[ \tau_{ij} - \frac{1}{3} \tau_{kk} \delta_{ij} = -2 \mu_S \tilde{S}_{ij} \]  

(3-31)
\( \mu_t \) is the viscosity of subgrid-scale turbulence. \( \tau_{kk} \) is the isotropic part of subgrid-scale stress and added to the term of filtered static pressure. \( \mathbf{S}_{ij} \) is the tensor of strain rate defined as:

\[
\mathbf{S}_{ij} = \frac{1}{2} \left( \frac{\partial \mathbf{u}_i}{\partial x_j} + \frac{\partial \mathbf{u}_j}{\partial x_i} \right)
\]

(3-32)

For compressible flows, the density-weighted (or Favre) filtering operator is introduced conveniently.

\[
\tilde{\phi} = \frac{\rho \phi}{\bar{\rho}}
\]

(3-33)

The Filtered Navier-Stokes equation of Favre takes the same form as Equation (3-28). The subgrid stress tensor in compressible form is expressed as isotropic and deviatoric parts:

\[
\tau_{ij} = \tau_{ij}^{\text{dev}} - \frac{1}{3} \tau_{kk} \delta_{ij} + \frac{1}{3} \tau_{kk} \delta_{ij}^{\text{isotropic}}
\]

(3-34)

The Smagorinsky model in compressible form is used to model the deviatoric part of the stress tensor of subgrid-scale.

\[
\tau_{ij} - \frac{1}{3} \tau_{kk} \delta_{ij} = 2 \mu_t (\mathbf{S}_{ij} - \frac{1}{3} \tau_{kk} \delta_{ij})
\]

(3-35)

For incompressible flows, \( \tau_{kk} \) can be simply neglected or added to the filtered pressure. Indeed, it can be rewritten as \( \tau_{kk} = \rho M_a^2 \frac{\partial \mathbf{u}}{\partial x} \), where \( M_a \) is Mach number of the subgrid. This \( M_a \) can be expected to be small when Mach number of the flow is small.

Four models are offered for the turbulent viscosity of subgrid-scale \( \mu_t \) in Ansys Fluent. They are: Smagorinsky-Lilly model, dynamic Smagorinsky-Lilly model, dynamic kinetic energy subgrid-scale model and Wale model. In subgrid-scale turbulence, a scalar flux is modeled using turbulent Prandtl number of subgrid-scale.

\[
q_j = -\frac{\mu_t}{\sigma_t} \frac{\partial \tilde{\phi}}{\partial x_j}
\]

(3-36)

Where, \( q_j \) is the subgrid-scale flux. In the dynamic models, Prandtl number or Schmidt number of subgrid-scale turbulence is obtained by the dynamic procedure proposed by Germano.

Smagorinsky-Lilly model is first proposed by Smagorinsky. In this model, the eddy viscosity is defined as:
\[ \mu_t = \rho L_s^2 |\vec{S}| \]  

(3-37)

Where, \( |\vec{S}| = \sqrt{2 L_s} \), \( L_s \) is the mixing length for subgrid scales and can be computed by

\[ L_s = \min(\kappa d, C_s \Delta) \]  

(3-38)

Where, \( \kappa \) is the constant of von Karman, \( d \) is the distance to the closest wall, \( C_s \) is the Smagorinsky constant, \( \Delta \) is the local grid scale and can be computed by the volume of the computational cell:

\[ \Delta = \nu^{1/3} \]  

(3-39)

In the inertial subrange, \( C_s = 0.17 \) is derived by Lilly for homogeneous isotropic turbulence. However, this value has to be reduced due to excessive damping of large-scale fluctuations caused in the presence of mean shear and in transitional flows as near solid boundary. In a word, \( C_s \) is the most serious shortcoming of this simple model and is not a universal constant. Nevertheless, \( C_s = 0.1 \) is the default value in Fluent which has been found to yield the best results for a wide range of flows.

For the LES model, the fluctuating velocity model of no perturbation is applied at velocity inlet boundary. The instantaneous velocity components are simply equal to their mean velocity counterparts and the turbulence level at the inflow boundary is negligible.

### 3.4.3 The Ffowcs Williams and Hawkings model

In Fluent, the noise can be computed from unsteady pressure fluctuations in two ways. Broadband noise source models estimates acoustic sources based on the results of steady state simulation. While the Ffowcs Williams and Hawkings can be used to model the propagation of acoustics sources for various objects including rotating fan blades. Transient LES predictions for surface pressure can be converted to a frequency spectrum using the built in Fast Fourier Transform (FFT) tool. The Ffowcs Williams and Hawkings (FW-H) equation is essentially an inhomogeneous wave equation and can be written as:

\[
\frac{a_0^2 \partial^2 p'}{\partial t^2} - \nabla^2 p' = -\frac{\partial^2}{\partial x_i \partial x_j} T_{ij} H(f) - \frac{\partial}{\partial x_i} \left[ \rho_0 n_j + \rho u_i (u_n - v_n) \right] \delta(f) + \frac{\partial}{\partial t} \left[ \rho_0 v_n + \rho (u_n - v_n) \right] \delta(f)
\]

(3-40)
Where, \( u_i \) and \( u_n \) are fluid velocity components in \( x_i \) direction and normal direction to the surface \( f=0 \) respectively. \( v_i \) and \( v_n \) are the corresponding surface velocity components. \( \delta(f) \) and \( H(f) \) are dirac delta function and Heaviside function, respectively. \( p' = p - p_0 \) and \( a_0 \) are sound pressure and sound speed of far field, respectively. \( f=0 \) denotes a mathematical surface which corresponds to a source surface and can be made coincident with a body surface or a permeable surface off the body surface but is not required to coincide with body surfaces or walls. \( f>0 \) is an exterior region in an unbounded space. \( n_i \) is a unit normal vector pointing to the exterior region. \( T_y \) is the Lighthill stress tensor and defined as:

\[
T_y = \rho u_i u_j + P_y - a_0^2 (\rho - \rho_0) \delta_y
\]  

\( P_y \) is the compressive stress tensor and is given as follows for a Stokesian fluid:

\[
P_y = p \delta_y - \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right)
\]

The subscript 0 denotes the free-stream quantities. The free-space Green function \( \delta(g)/4\pi r \) is used to obtain the solution to FW-H equation which consists of surface integrals and volume integrals. The quadrupole sources in the region outside the source surface is represented by the volume integrals while the surface integrals represent the contributions from monopole and dipole acoustic sources and partially from quadrupole source. In low subsonic flow and the source region enclosed by the source surface, the volume integral contribution becomes small. Thus,

\[
p'(\vec{x}, t) = p'_I(\vec{x}, t) + p'_L(\vec{x}, t)
\]

Where, \( 4\pi p'_{\mathcal{R}}(\vec{x}, t) = \int_{f=0} \left[ \frac{\rho_0 (U_i + U_n)}{r(1-M_r)^2} \right] dS + \int_{f=0} \left[ \frac{\rho_0 U_n (rM_r + a_0 M_r - a_0 M_r^2)}{r^2(1-M_r)^3} \right] dS \]

\[
4\pi p'_{\mathcal{L}}(\vec{x}, t) = \frac{1}{a_0} \int_{f=0} \left[ \frac{L_r}{r(1-M_r)^2} \right] dS + \int_{f=0} \left[ \frac{L_r - L_m}{r^2(1-M_r)^3} \right] dS
\]

\[+ \frac{1}{a_0} \int_{f=0} \left[ \frac{L_r (rM_r + a_0 M_r - a_0 M_r^2)}{r^2(1-M_r)^3} \right] dS \]

Where, \( U_i = v_i + \frac{P}{\rho_0} (u_i - v_i) \)

52
When the integration surface coincides with an impenetrable wall, \( p'_i(\mathbf{x},t) \) and \( p'_i(\mathbf{x},t) \) are thickness and loading term, respectively. In their equations, appearing subscripts are the inner products of a vector with a unit vector, such as \( U_n = \hat{U} \cdot \hat{n} = U_j n_i \) and \( L_r = \hat{L} \cdot \hat{r} = L_r r_i \), where \( \hat{n} \) and \( \hat{r} \) denote the unit vectors in wall-normal and the radiation directions, respectively. A dot over a variable denotes differentiation of that variable in source time. The square brackets in their equations denote that the integral kernels are computed at the corresponding retarded times \( t \) which is defined as follows:

\[
\tau = t - \frac{r}{a_0}
\]  

(3-44)

\( t \) and \( r \) are the observer time and the distance to observer, respectively.

The FW-H equation can handle rotating surfaces as well as stationary surfaces. The formulation permits source surfaces to be permeable. When a permeable source surface is placed at a certain distance away from the body surface, the integral solutions include the contributions from the quadrupole sources within the source surface enclosing region. In the volume enclosed by the permeable surface, the mesh resolution needs to be fine enough to resolve the transient flow.

### 3.5 Boundary conditions and near wall treatment

The flow inlet boundary condition is given uniform velocity in axial direction at absolute reference frame. While outlet boundary condition is prescribed gauge pressure as static pressure which is the same with atmosphere pressure. The pressure outlet boundary condition results in a better rate of convergence when backflow occurs during iteration.

Wall boundary conditions are used to bound solid and fluid zones. Non slip boundary condition is applied at walls by default which uses adjacent flow properties to predict the shear stress, meanwhile a slip wall also can be modeled by specifying shear and the tangential velocity will be computed. In this dissertation, moving reference frame is used for rotating fluid zone. Blade wall is moving wall with zero speed relative to adjacent cell zone rotating fluid and in shear condition of
no slip. The walls in inertial reference frame are moving walls with reverse speed of rotating fluid zone and specified shear.

The steady flow field of studied propeller fans is assumed to be axisymmetric. In order to save mesh number, one fifth flow passage has been made to simulate the whole flow passage in the propeller rotor by specified the periodic boundary condition.

RNG $k-\varepsilon$ model provides a differential formula for viscosity accounting for the effect of low Reynolds number. Therefore, an appropriate near wall treatment will make effective use of this feature. In this dissertation, enhanced wall treatment has been applied to the near wall treatment. Enhanced wall treatment combines a two-layer model with enhanced wall functions that can be used with coarse meshes as well as fine meshes. In enhanced wall functions, linear and logarithmic wall laws are blended to extend the application throughout the near wall region including laminar, buffer and full turbulent sublayers.

\[
    u^+ = e^+ u_{tum}^+ + e^+ u_{turb}^+
\]  

(3-45)

Where the blending function is

\[
    \Gamma = \frac{a(y^+)^i}{1 + by^+} 
\]  

(3-46)

Here, $a=0.01$ and $b=5$. Therefore,

\[
    \frac{du^+}{dy^+} = e^+ \frac{du_{tum}^+}{dy^+} + e^+ \frac{1}{dy^+} \frac{du_{turb}^+}{dy^+} 
\]  

(3-47)

this formula corrects asymptotic behavior for big change of $y^+$ and gives reasonable velocity profiles when $y^+$ falls inside buffer region near wall ($3 < y^+ < 10$). Enhanced wall approach makes the fully turbulent law easily modified and takes pressure gradients or variable properties into account.

The enhanced turbulent wall law for compressible flow with pressure gradient and heat transfer is

\[
    \frac{du_{turb}^+}{dy^+} = \frac{1}{k} [S'(1 - \beta u^+ - \gamma (u^+)^2)]^2 
\]  

(3-48)

Where,

\[
    S' = \begin{cases} 
    1 + \alpha y^+ & (y^+ < y_s^+) \\
    1 + \alpha y_s^+ & (y^+ \geq y_s^+) 
    \end{cases} 
\]  

(3-49)

Here, \( \alpha = \frac{v_u}{\tau_{m} u} \) \( \frac{dp}{dx} = \frac{\mu}{\rho S^2 (u^+)^3} \frac{dp}{dx} \)
\[ \beta = \frac{\sigma_{i} q_{i} \mu^{'}}{c_{p} \tau_{w} T_{w}^{}} = \frac{\sigma_{i} q_{v}}{\rho x_{p} \mu^{'}} \]

\[ \gamma = \frac{\sigma_{i} (u')^{2}}{2 c_{p} T_{w}^{}} \]

By default \( y^{'\text{f}} = 60 \), at which the slope of logarithmic law remains fixed. The coefficient \( \alpha \) denotes the effect of pressure gradients while the coefficients \( \beta \) and \( \gamma \) represent the thermal influences. If they are all equal to zero, the analytical solution is in the case of classical turbulent wall log-law.

The laminar wall law is

\[
\frac{du^{'}}{dy^{'}} = 1 + \alpha y^{'
}
\]

(3-50)

This formula expresses the effects of pressure gradients but neglected variable properties affected caused by heat transfer and compressibility. Integration of this formula results in

\[
u^{'\text{w}} = y^{'} (1 + \frac{\alpha}{2} y^{'})
\]

(3-51)

Enhanced thermal wall functions follow the same approach with velocity profile, and here it is neglected. The enhanced wall treatment is available for both \( k-\varepsilon \) models and LES models.

### 3.6 Geometry model and grid generation

#### 3.6.1 Flow field modeling

The studied propeller fans have five blades and the flow through them is assumed to be axisymmetric. In order to reduce the calculation work, one fifth flow passage is built as shown in [Figure 3-2](#). The inlet and outlet regions are extended domains for the flow field of the propeller fan which are one fifth of the cylinder with the diameter of 880mm and the length of 440mm. The rotating area in the middle of the flow field modeling consists of blade flow passage between the suction surface and pressure surface, tip clearance area and 5 mm transitional regions to the extended domains. The opened blade flow passage is shown in [Figure 3-3](#). The meshed surfaces are suction surface and pressure surface abbreviated to \( \tilde{SS} \) and \( \tilde{PS} \), while the solidified surfaces are casing and hub, respectively. Tip clearance is the gap between blade and casing marked as \( \tilde{TC} \) in the figure. The rotating direction is counter clock wise and the flow is from left to right as shown in
arrows.

**Fig. 3-2** Modeling flow field of propeller fans

**Fig. 3-3** Blade flow passage
3.6.2 Grid generation

The numerical solution of partial differential equations are approximated to the solution of algebraic equations building on the discretization of the physical domain into a collection of points and element volume. The grid generation is the practice of discretizing the domain into a finite number of elements.

In this dissertation, triangular and quadrilateral cells in two dimension, and tetrahedral, hexahedral and wedge cells in three dimension are used to generate the unstructured and structured grid for rotating area and extended domains, respectively. The meshes are presented in Figure 3-2.

The mesh quality plays a significant role in the accuracy and stability of the numerical simulations. The shape of the cell including skewness and aspect ratio has been checked. Skewness of equivolume cell is defined as the ratio of difference between equilateral cell with the same circumradius and the cell size with equilateral cell size. The aspect ratio is a measure of the stretching of the cell which is defined as the ratio of the maximum distance between the cell centroid and face centroids to the minimum distance of the cell nodes. A general rule is to avoid aspect ratio in excess of 5:1.

For near wall mesh in turbulent flow, the distance from the wall at the wall adjacent cells should be considered to satisfy the log-law, that is $y^+ > 30-60$, due to wall function validation in this condition. For near wall mesh in LES, a very fine near wall mesh space is necessary for the best results even though no restrictions on near wall mesh space is required.

3.6.3 Convergence and mesh independence

For a steady state simulation, convergence is satisfied when the solution meets the following three conditions: First, root-mean-square error values of residual have reduced to a specified value typically $10^{-3}$. Second, the values of flow parameters at interesting monitor points have reached a steady solution. Third, the overall imbalances for all variables on the calculated domain is less than 1%. 

57
When numerical solution satisfies the convergence conditions, it is also important to make sure that the solution is independence of the mesh resolution. It is a common reason for error results in CFD that the mesh independence is not conducted. In this dissertation, the mesh independence check process has carried out twice due to the use of flow passages for two propeller fans. As a results, the mesh independence sizes are almost the same because the studied propeller fans have almost the same dimensions. In this section, just one time of the mesh independence check processes will be shown.

Step one: Run the initial simulation on the initial mesh and make sure the convergence satisfy the three conditions mentioned above. Otherwise, refine the mesh and repeat.

Step two: Once the first simulation has met the convergence criteria, refine the mesh and repeat step one. At this time, compare the results of monitor point values from step one and two. If they are the same within the allowable tolerance, then the mesh at step one is accurate enough to obtain the results. Otherwise, the solution is changing because of mesh resolution. In this case, refine the mesh more, and repeat this process until the mesh independence is achieved.

**Figure 3-4** shows the mesh independence study in this dissertation. The abscissa is the number of cells and the ordinate is mass weighted averaged static pressure at outlet.

![](image.png)

Fig.3-4 Mesh independence study
Chapter 4

Experimental Methods

4.1 Experimental apparatus

The propeller fan with hub-ratio of 0.51, having five blades and the diameter of 180mm has been studied in the dissertation, as shown in Figure 4-1. The rear half of its blade tip is covered with casing and the tip clearance is kept in 1mm. The rotor has been tested in the experiment equipment system in Figure 4-2 with an 880mm×880mm square cross section and a length of 3m. The rotor speed can arbitrarily set on the range of 0 to 3100rpm driven by the direct-current electric motor. In order to get smaller static pressure rise, the centrifugal fans were set in the outlet of the system. The flow rate of the system can be well controlled by altering the output of the rotating speed. In order to clarify the unsteady flow field of the propeller fan, it is significantly important to make clear its characteristics, such as pressure rise, axial torque and efficiency. The flow rate, pressure rise and shaft power are represented with non-dimensional form of flow coefficient, pressure rise coefficient and axial torque coefficient which are defined as:

\[ \varphi = \frac{4Q}{\pi(D_t^2 - D_h^2) U_t} \]  \hspace{1cm} (4-1)

\[ \psi = \frac{2\Delta p_s}{\rho U_t^2} \]  \hspace{1cm} (4-2)

\[ \tau = \frac{8T \cdot \omega}{\pi(D_t^2 - D_h^2) \cdot \rho U_t^5} \]  \hspace{1cm} (4-3)

And efficiency

\[ \eta = \frac{\Delta p_s \cdot Q}{T \cdot \omega} \]  \hspace{1cm} (4-4)

Where \( U_t \) is speed on the rotor tip, \( D_t \) and \( D_h \) are the diameter of the rotor tip and hub respectively, \( Q \) is the volume flow rate, \( \Delta p_s \) is the static pressure rise, \( \rho \) is atmospheric density, \( T \) is the rotor torque and \( \omega \) is the rotating speed of the rotor.
The static pressure of the propeller fan is obtained by averaging the data from 16 static pressure taps setting at downstream of rotor as shown in Figure 4-2. The flow rate is calculated by the differential pressure between the upstream and downstream of the nozzle where eight pressure taps are set respectively. The higher flow rate, corresponding to larger differential pressure, can be obtained by elevating the rotating speed of two centrifugal booster fans, and the rotor speed is set by 3000 rpm in this dissertation.

4.2 Measurement methods

4.2.1 Working principle of hot wire

Figure 4-3 shows the configuration of slanted hot-wire probe. On plane A perpendicular to the probe axis, $x$-axis is the reference coordinate and $y$-axis is the projected direction of hot-wire. The...
velocity vector and the slanted hot-wire sensor make angle $\gamma$ (pitch angle) and $\lambda$ with plane $A$. And their projection on plane $A$ make angle $\zeta$ (yaw angle) and angle $\Delta \zeta_i$ with x-axis. $\zeta_i$ denotes the angle between the $y$-axis and the projection of velocity vector on Plane $A$. Then, the angle $\chi$ between the slanted hot-wire sensor and velocity vector is expressed as follows:

$$\cos \chi = \cos \lambda \cos \gamma \cos \zeta_i + \sin \lambda \sin \gamma$$

(4-5)

In the calibration process, the projection of the velocity vector on plane $A$ is on x-axis, that is $\zeta=0$, $\zeta_i=\Delta \zeta_i$. According to the linearizer in the hot-wire anemometer, the linearized output $E$ is a function of velocity vector, pitch angle and $\chi$, which can be given as:

$$E(\gamma, \chi, v) = F_1(\gamma, \chi)F_2(\gamma)K|v|$$

(4-6)

Where, $F_1(\gamma, \chi) = E(\gamma, \chi, v)/E(\gamma, 90^\circ, v) = -\cos[A(\gamma)(\chi-90^\circ)]$

$$A(\gamma) = A_0 + A_1 \gamma + A_2 \gamma^2$$

$$F_2(\gamma) = E(\gamma, 90^\circ, v)/E(0, 90^\circ, v) = 1 + B_1 \gamma + B_2 \gamma^2$$

The outputs $E$ are measured by changing angle $\zeta_i$ in the way of revolving the probe axis with the pulse motor for getting various values of velocity $v$ and pitch angles $\gamma$. The constant coefficients $A_0, A_1, A_2, B_0, B_1, B_2$ and $K$ are determined by the generalized least squares method using Equation (4-5) and Equation (4-6). The output value is ignored when $\zeta_i=0$ because of a large standard variance in output owning to the effect of the prong. Pitch angle $\gamma$ is prescribed seven values in (-30°, 30°) every 10° in calibration.

The measuring procedure at downstream of the axial flow fan are expressed as follows. Firstly, find the orientation of the probe when the anemometer indicator shows the minimum value. This can be finished by revolving the probe about its axis with a computer. Note that orientation of the probe is $\Delta \zeta_m$. Please pay attention that $\Delta \zeta_m$ does not always correspond to $\zeta_i=0$ for any sampling points because the indicator output is circumferential average value of the periodic flow. Next, the sampling is conducted by revolving the probe to prescribe orientations $\Delta \zeta_i$. Twenty five orientations are chosen at interval of 14.4° in one revolution. At each orientation, the ensemble average and variance are obtained by periodic multi-sampling and ensemble averaging technique which will be introduced in the next section.
\[
\tilde{E}_i = F_1(\gamma, \chi_i)F_2(\gamma)K\left|v\right| 
\]

(4-7)

Where,

\[
\cos \chi_i = \cos \lambda \cos \gamma \cos(\tilde{\xi} + \Delta \xi_i) + \sin \lambda \sin \gamma
\]

(4-8)

The unknown averaged values of yaw angle, pitch angle and velocity are determined by a group of twenty five equations corresponding to different values of \(\Delta \xi_i\) by the generalized least square method.

### 4.2.2 Periodic multi-sampling and ensemble average technique

The periodic multi-sampling and ensemble average technique are used to acquire wall pressure and velocity measurement. Application of this technique requires the measured flow periodically regular. In this dissertation, this technique are applied to measure the velocity with a hot wire in the internal flow of propeller fans. Thus, the data processing method of this technique need to give a detail explanation.

**Figure 4-4** presents the process of periodic multi-sampling. The outer trigger signal is used to trigs the beginning of sampling. It is a position signal of the rotor shaft rotation which can locate the beginning measured point corresponding to a blade position in the circumference. Then this beginning point of sampling will be phase-locked in the same circumferential position in rotor. The trigger signal is conducted 26 times and there are 500 sampling points which cover one circle of
rotor in each trigger measurement. Channel 0 and Channel 1 are signals from anemometer and digital computer respectively. 10 microsecond between two channels is the A/D converter time. The sampling time of instantaneous velocity is 40 microsecond.

**Figure 4-5** shows schematic of hot wire measuring system. The single sensor hot-wire probe is connected directly to the pulse motor. The pulse motor is instructed by computer to make the hot-wire probe revolve in prescribed orientation. The scanning pulse generator generates a trigger pulse at the peak of timing signal from the photo interrupter and the sample pulses correspond to the correct positions of the sampling points in every revolutions. According to the instruction of the scanning pulse, the linearized signals are obtained by a linearizer incorporated in a constant temperature hot-wire anemometer, and then converted to digital values by an analog-digital converter. The converted digital values being transferred to computer, the sum and the sum of squares are figured out for each distinct sampling points to yield the averaged value and the variance. The same sampling and averaging process are carried out for every orientations of the probe axis. All the sampling points on the same circumference are scanned in every revolution.

![Figure 4-4 Process of periodic multi-sampling](image)
4.3 Uncertainty analysis

No measurement is completely accurate so that uncertainty analysis is necessary for experiment studies. Kline (1985) [125] distinguished the experiment into four classes: quick-sort, acceptance test, report of research and calibration test and reported that uncertainty analysis should be applied as a part of publication procedure for the last three experiments. And the combination method about propagation of uncertainties into results discussed by Kline will be adopted in this dissertation.

Uncertainty analysis can reduce error in experiment work systematically. However, an uncertainty is not the same with an error which is a fixed difference between the true value and the measured value. It is a statistical variable. In general sense, an uncertainty is usually an averaged value describing the central tendency like standard deviation. Uncertainty analysis has no universal procedure but some procedures are useful for getting an uncertainty value.

In this dissertation, the performance tests and hot wire anemometer measurements are carried out.
for the studied propeller fans. For the performance tests of propeller fans, the functional relationship of pressure rise coefficient and efficiency with flow coefficient are reported after measurement. These parameters are defined in Equation (4-1), (4-2), (4-4) in Section 4.1. To know the uncertainty of them, the measured parameter: flow rate, blade tip speed, air density, static pressure, rotor torque and rotor speed should be made uncertainty analysis, as shown in Table 4-1. The first four uncertainties static pressure, rotor torque, rotor speed and the three below them are supposed according to test certification of meters and their fluctuation in measurement process. Uncertainty of flow rate could be obtained in propagation processing below. Squaring both sides of the equation of flow rate in the last second one of Table (4-1), it can be written as

\[ Q^2 = A^2 \cdot \frac{2 \cdot \Delta p}{\rho} \]  

Differentiating Equation (4-9), then

\[ 2QdQ = 2A^2 \cdot \left( \frac{d\Delta p}{\rho} - \frac{\Delta p}{\rho^2} d\rho \right) \]

Dividing Equation (4-10) by Equation (4-9) gives

\[ \frac{dQ}{Q} = \frac{1}{2} \cdot \left( \frac{d\Delta p}{\Delta p} - \frac{d\rho}{\rho} \right) \]

Replace the differentials in Equation (4-11) with the uncertainties of \( \Delta p \) and \( \rho \) as shown in Table (4-1), and use the combination method of root-mean-square (rms) to obtain the uncertainty of \( Q \) in the results.

\[ \frac{dQ}{Q} = \frac{1}{2} \cdot \sqrt{(0.2\%)^2 + (0.1\%)^2} = 0.112\% \]

In this dissertation, the rms method as presented above has been applied to calculate the propagations of uncertainties into indirectly measured physical parameters in results providing negligible uncertainties in areas calculation in Equation (4-1) and Equation (4-3). For example, after dealing with Equation (4-2) as above mentioned processing, it will be written as

\[ \frac{d\psi}{\psi} = \frac{dp_p}{p_p} - \frac{d\rho}{\rho} - \frac{2dU_t}{U_t} \]
Then,

\[
\frac{d\psi}{\psi} = \sqrt{(0.2\%)^2 + (0.1\%)^2 + (2 \times 0.019\%)^2} = 0.225\%
\]  

(4-14)

Therefore, the uncertainties of pressure rise coefficient, efficiency and flow coefficient are 0.225\%, 0.359\%, 0.114\%, respectively.

| Table 4-1 |
|-----------------|-----------------|-----|-------|
| Terms          | Specification                                           | Unit | Uncertainty |
| Static Pressure| Manometer monitor reading                               | Pa   | 0.2\%   |
| Pressure Difference | Manometer monitor reading                               | Pa   | 0.2\%   |
| Rotor Torque   | Torque meter reading                                     | Nm   | 0.2\%   |
| Rotating Speed | Torque meter reading                                     | rpm  | 0.03\%  |
| Saturated Vapor Pressure | \( P_i = g(0.405T_i^2 - 0.85T_i + 93) \) | Pa   | 0.1\%   |
| Atmospheric Pressure | \( P = P_0g(136 - 0.0025T_i) \)                         | Pa   | 0.1\%   |
| Air Density    | \( \rho = \frac{P - 0.378 \times P \times H}{100 \times 29.3 \times g(T_i + 273)} \) | kg/m³ | 0.1\%   |
| Volume flow Rate | \( Q = A \cdot \frac{2 \cdot \Delta p}{\rho} \) | m³/s | 0.112\% |
| Blade Tip Speed | \( U_i = r \cdot \omega \)                              | m/s  | 0.019\% |

For hot wire anemometer measurements, three components of averaged and fluctuation velocities are measured simultaneously in real time. The calibration is conducted for the slanted hot wire probe every time before real tests. In calibration, the velocity and pitch angle are calibrated in linearity relationship with output voltage. Suppose a precision uncertainty of 1.0\% and negligible bias in calibration. In real tests, the slanted hot wire probe is sampling 500 data numbers for every 14.4° revolution. The measured data are absolute velocity and velocity fluctuation. The absolute velocity is an averaged datum which is defined as:

\[
\bar{V}_{(j)} = \frac{\sum_{i=1}^{N} V(i, j)}{N}
\]  

(4-15)
Here, \(i=(1, N), N=26, j=(1, N_p), N_p=500\). \(v(i, j)\) is measured velocity and \(\overline{V}_{(j)}\) stands for averaged velocity. The deviation of data fluctuation is defined as:

\[
D_{(j)} = \frac{1}{N} \sum_{i=1}^{N} (\overline{V}_{(j)} - v(i, j))^2
\]

Three components of velocity are evaluated by the pitch angle and yaw angle, as shown below:

\[
\begin{align*}
\overline{V}_{r(j)} &= \overline{V}_{(j)} \sin \gamma \\
\overline{V}_{a(j)} &= \overline{V}_{(j)} \cos \gamma \cos \xi \\
\overline{V}_{t(j)} &= \overline{V}_{(j)} \cos \gamma \sin \xi
\end{align*}
\]

Here, \(\gamma\) and \(\xi\) are pitch angle and yaw angle respectively as shown in Figure (4-3). Suppose their uncertainty is \(1^\circ\). Then

\[
\left| \frac{\sin(\gamma + 1^\circ) - \sin \gamma}{\sin \gamma} \right| = \cos 1^\circ + \cot \gamma \sin 1^\circ - 1 \approx \cot \gamma \sin 1^\circ \approx 1.745\% \quad (\gamma = 45^\circ)
\]

When pitch angle is within \(45^\circ\), tangential velocity will have large uncertainty,

\[
\left| \frac{\cos(\gamma + 1^\circ) - \cos \gamma}{\cos \gamma} \right| = \left| \cos 1^\circ - \tan \gamma \sin 1^\circ - 1 \right| \approx \tan \gamma \sin 1^\circ \approx 1.745\% \quad (\gamma = 45^\circ)
\]

When pitch angle is larger than \(45^\circ\), meridional velocity will have large uncertainty. Then based on the rms propagation method, the uncertainties of tangential velocity, axial velocity and radial velocity are \(2.663\%, 2.663\%, 2.011\%\) respectively in the condition of \(45^\circ\) pitch angle and \(45^\circ\) yaw angle. This means that uncertainty of axial velocity or meridional velocity is less than \(2.663\%\) or \(2.011\%\) when pitch angle is within \(45^\circ\). Radial velocity is highly effected by pitch angle especially near zero degree and yaw angle has the same influence on tangential velocity. However, the pitch angle is about in the range of \((10^\circ, 30^\circ)\), and the yaw angle is almost in \((30^\circ, 75^\circ)\).
Chapter 5
Experimental Study of Inflow Effect on Performance of Propeller Fans

5.1 Introduction on inlet flow of propeller fans

A lot of daily using axial fans are propeller fans, which are classified into several types using the relationship of relative location between blade rotor and outer casing. They are classified into four types: ducted type on common, half-ducted type, semi-opened type, and open type, as shown in Figure 5-1. However, most applications of propeller fans are in non-ducted types for the space limitation. In order to clarify the effect of rotor inlet geometry of the propeller fan on performance and velocity fields at rotor outlet, the experimental investigation was carried out using a hotwire anemometer in this chapter. Fan test and internal flow measurement at rotor outlet were conducted for two types of inlet geometry. At the internal flow measurement, a single slant hotwire probe was used and a periodical multi-sampling technique was adopted to obtain the three-dimensional velocity distributions.

In section 5.2, two types of inlet geometry were analyzed to clarify the effect of rotor inlet geometry of the propeller fan on performance and internal flow fields. The first type is half-ducted that the rotor blade tip is fully covered by a casing. The second type is semi-opened included two kinds that consist of opened front one-third or two-third part of blade tip and covered rest part.

![Fig.5-1 Types of axial fans and image of inlet flow pattern](image-url)
In cases of a ducted and a half-ducted type fans, tip clearance is between rotor blade tip and outer casing. Tip leakage flow occurs over the blade tip which is caused by the flow from blade pressure surface to suction surface. The tip leakage vortex is produced by tip leakage flow which interacts with main flow. It has been found that tip leakage vortex brings out the blockage effect and makes noise by the interaction with some structures at downstream region. In case of an open type fan, there is no tip leakage flow but strongly radial inflow from blade tip areas. It has found that this radial inflow becomes strong near blade suction surfaces. Radial inflow near blade suction surface interacts with the main flow makes the formation of tip vortex. In case of semi-opened type fan, the radial inflow makes a part of blade tip working as blade leading edge. Therefore, tip leakage vortex and tip vortex are both exit in semi-open type fans. In order to study these internal flow characteristics, it is very important to clarify the effect of inlet geometry on fan performance. In section 5.3, the performance and flow field near blade tip of semi-open type fan with different bell-mouth radius will be analyzed by experimental method.

5.2 Study on half-ducted and semi-open inlet structures

In this section, half-ducted type and semi-open type of propeller fan will be investigated by experimental method. Figure 5-2 shows the schematic views of inlet geometries for the tested fans and the details of measuring stations. Three propeller fan rotors called ṇType A, ṇType B and ṇType C, which are half-ducted fan in Figure 5-2 (a), one third semi-opened fan in Figure 5-2 (b) and two thirds semi-opened fan in Figure 5-2 (c) respectively, will be discussed. The three-dimensional velocity fields at rotors outlet are measured by a single slant hotwire probe in the survey stations as shown in Figure 5-2 (a). The effect of inlet geometry on fan characteristics will be discussed.

The hotwire probe is traversed at 5mm downstream of rotor as shown in Figure 5-2 (a). The measuring stations on each survey line are 16 stations. The velocities are measured by a single slant hotwire probe and periodic multi-sampling technique developed by Kuroumaru et al. (1982) [102] is applied. At each measuring station, the hotwire probe is revolved about its axis at 14.4 degree interval for each revolution. The hotwire probe is a tungsten wire with 5 μm diameter and 1mm
sensitive length. Measuring data are processed by the use of phase-locked averaging technique based on the rotor blade phase. The similar method was used by Inoue et al. (1985) [70], (1986) [125]. The rotor blade phase was obtained from the output signal of the photo-sensor mounted on rotor shaft shown in Figure 4-2. Periodic multi-sampling and ensemble average technique were introduced in Chapter 4.

At the experiment, blade rotating speed is 3000rpm, and Reynolds number based on blade tip velocity and blade chord length at mid-span is 1.32×10^5. Performance characteristics of the propeller fans and velocity measurement at rotor outlet using a hot-wire probe have been carried out in a wind tunnel as shown in Figure 4-2. The distributions of time-mean velocity components along radius and phase-lock averaged velocity components are shown in Section 5.2.2 and Section 5.2.3. The effect of inlet geometry on fan characteristics and flow fields will be discussed based on those results.

5.2.1 Analysis on performance of tested propeller fans

Figure 5-3 shows the pressure flow rate curves of three types of tested propeller fans. The abscissa is flow rate coefficient \( \phi \) and the ordinate is pressure-rise coefficient \( \psi \). The lines with circle, diamond and triangle symbols in this figure are corresponding to propeller fans of Type A, Type B and Type C, respectively. The definitions of flow rate and pressure coefficients are presented in Equation (4-1) and Equation (4-2), respectively.

The pressure coefficient becomes small as the rotor shifts to high flow rate as shown in Figure 5-3. Generally, the pressure difference between blade suction surface and pressure surface becomes large at the blade tip for ducted fan. The tested propeller fans can obtain higher pressure rise in the same flow conditions as the open areas decrease over casing, especially after stall point conditions (\( \text{SP}_0 \) in Figure 5-3). That is because the tested propeller fans work mainly at blade tip region due to the application of their design method for ducted fan. At the same time, the maximum flow rate of the ducted propeller fan is the widest while possible flow rate is less than 0.36 for Type C. At the low flow rate region, the stall point moves to large flow rate condition, that is, the stall margin becomes small, as the rotor shifts from Type A to Type C. Also, the pressure curves of Type A and
Type B have almost the same tendency at all flow rate condition, while it shows different tendency for Type C at low flow rate region.

The internal flow measurements at rotor outlet are mainly carried out at $\phi = 0.290$ for three propeller fans which is seen as the reference flow condition. For the study of small flow rate region, $\phi = 0.230$ for Type A and Type B, $\phi = 0.270$ for Type C fan are taken in the measuring procedure. At large flow rate region, the flow fields at $\phi = 0.350$ for all types of fan will been measured.

![Diagram of inlet geometry and hotwire survey stations](image)

Fig.5-2 Inlet geometry of tested propeller fans and hotwire survey stations at rotor outlet

![Graph of performance curves](image)

Fig.5-3 Performance curves

71
5.2.2 Time-averaged velocity distributions along radius

Figure 5-4 shows the distributions of time-averaged velocity components along radius for three types of inlet geometries at rotor outlet. Figure 5-4 (a-c) show the measurement results of Type A, Type B and Type C in three flow rate conditions, respectively. For each figure from left to right is the distributions of non-dimensional axial velocity component $C_{a^*}$, circumferential velocity component $C_{\theta^*}$ and radial velocity component $C_{r^*}$, respectively. They are normalized by the blade tip speed $U_t$. The x-axis $v^*$ denotes the non-dimensional velocity components $C_{a^*}$, $C_{\theta^*}$ and $C_{r^*}$, while the y-axis $r^*$ represents the non-dimensional radius, and $r^*=1.0$ is the location of exit inner radius of outer casing. $\bar{\text{Tip}}$ and $\bar{\text{Hub}}$ indicate the location of rotor blade tip and hub at rotor trailing edge.

In Figure 5-4(a) for Type A, the axial velocity decreases and the location of maximum velocity moves to radial inwards when the flow rate reduces. And the decrease rate of axial velocity along inwards radius becomes large. At outside of blade tip ($r^*>1.0$), the velocity becomes almost zero at all flow rate conditions. The circumferential velocity and radial velocity become large in small flow rate. Especially at $\phi = 0.230$, the circumferential velocity becomes dramatically large but decreases near hub. At $\phi = 0.350$, the radial velocity inside the blade tip region ($r^*<1.0$) is minus value which means the air flow turning to radial inwards and the contraction of flow fields. At $\phi = 0.290$, the radial velocity is near zero which indicates the air flow at rotor outlet almost parallel to rotor axis. At $\phi = 0.230$, the air flow turns to radial outwards as seen from the distributions of radial velocity and the radial velocity is almost zero near hub. Due to the three components of velocity becoming very small at $\phi = 0.230$, it is considered that the dead flow region is formed near hub which is similar to hub-corner stall in axial compressor reported by Lei et al. (2008) [126].

In Figure 5-4(b) for Type B, the tendency profile of axial velocity is almost the same with Type A except the increased axial velocity near blade tip region at $\phi = 0.230$. The circumferential velocity gradually increases as the flow rate reduces and relative large circumferential velocity is near blade tip ($0.8<r^*<1.0$). The similar velocity distributions which is due to the tip leakage vortex have been reported by Inoue et al. (1986) [127] in axial compressor rotor. The tendency of radial velocity is almost the same with Type A. The negative radial velocity inside the blade tip at $\phi = 0.350$ is a little smaller than the one of Type A which means smaller contraction than the results of Type A. The
value of radial velocity becomes plus for $\phi = 0.290$ at the region of $r^* > 0.8$ and $\phi = 0.230$ at all radial measurement stations.

Fig. 5-4 Distributions of time-averaged velocity along radius at rotor outlet
In Figure 5-4(c) for Type C, the axial velocity distributions have local decline near blade tip region and the circumferential velocity distributions have large value at \( r^* < 1.0 \) at all flow rate condition. The radial velocity distribution at \( \phi = 0.290 \) shows positive value at all radial location which indicates the flow field turning to radial outward and it is similar to near stall condition for Type A and Type B.

Figure 5-5 presents the distributions of time-averaged velocity components along radius at rotor outlet at \( \phi = 0.290 \). The distributions of non-dimensional axial, circumferential and radial velocity components are shown from left to right in the figure. The symbols have the same means in Figure 5-4. For the axial velocity, the maximum value becomes larger near \( r^* = 0.85, 0.8 \) and \( 0.7 \) for Type A, Type B Type C respectively which indicates the flow in radial direction involving more and main flow region becoming small as open casing area enlarges. For the circumferential velocity, the large value region is near blade tip where the axial velocity rapidly decreases especially remarkable for Type B and Type C. It is considered that the strong vortex exists near blade tip for Type B Type C. For the radial velocity, its distribution of Type B is similar to Type A near hub and similar to Type C near blade tip.

For the three kinds of inlet geometry, ducted type can obtain the most uniform axial velocity in the widest main flow region along radius. In design flow rate condition, the radial velocity of Type A is the closest to zero. When flow rate is larger than design flow rate \( \phi = 0.290 \), the air flow turns to radial inward and the flow fields becomes contraction. The inlet geometry gives strong influence.
on circumferential velocity in three velocity components. As the open region of blade tip enlarges, the circumferential velocity is influenced at least by flow rate and its distributions become uniform along radius which is benefit for obtaining energy at outlet of rotors. However, the circumferential velocity of the ducted type becomes large in low flow rate 0.23 near the tip regions which has been affected by vortex in that region. Type C almost has the same distributions of circumferential velocity for three flow rate except near the hub region in high flow rate.

5.2.3 Phase-lock averaged velocity distributions

Figure 5-6, Figure 5-7 and Figure 5-8 show the contour maps of axial, circumferential, radial velocity component and velocity fluctuation at rotor outlet for Type A, B and C at design flow rare $\phi = 0.290$, respectively. The abscissa is in pitch-wise direction and the ordinate is in radial direction. The value of velocity in each figure is normalized by the blade tip speed $U_t$. Blade rotating direction is from left to right as shown at top-right side of figures. $\bar{r}_{SS}$ and $\bar{r}_{PS}$ in each figure mean blade suction surface and pressure surface, respectively.

In Figure 5-6, the flow field for Type A at $\phi = 0.290$ is considered to be an appropriate flow field because it is similar to the one expected. In Figure 5-6 (a) and (b), the distributions of axial velocity and circumferential velocity inside the blade passage are almost uniform. The span-wise extent of uniform axial velocity area for blade SS is a bit larger than the one for blade PS. While the circumferential velocity becomes clearly large at blade wake region. For radial velocity distributions in Figure 5-6 (c), the flow near blade pressure surface and suction surface tend to radial inward and radial outward, respectively. And it is almost zero inside the blade passage. For velocity fluctuation in Figure 5-6 (d), it is small inside the blade passage but becomes a little higher at blade wake region. Observing the flow region marked $\bar{r}_{AA}$, the circumferential velocity and radial velocity become a little large, the contour line value of axial velocity slightly decreases, and the velocity fluctuation becomes higher. All those indicate that the effect of vortex on the flow field is hardly observed at rotor outlet for Type A fan at $\phi = 0.290$. 

75
(a) Axial velocity component

(b) Circumferential velocity component

(c) Radial velocity component

(d) Velocity fluctuation

Fig. 5–6 Contour maps of velocity and velocity fluctuation at rotor outlet of Type A at $\phi = 0.290$
Fig. 5-7 Contour maps of velocity and velocity fluctuation at rotor outlet of Type B at $\phi = 0.290$
In Figure 5-7, the uniformity of flow field is a little distorted for Type B comparing with Type A. The axial velocity becomes small near hub-blade SS corner marked "B" in Figure 5-7 (a)
comparing to Figure 5-6 (a), which is due to the effect of the hub-corner separation. In Figure 5-7 (b)-(d), circumferential velocity and radial velocity become large and velocity fluctuation becomes higher in the region marked \( \text{\textcircled{B}} \) comparing to the result of Type A, so as they do in the region marked \( \text{\textcircled{C}} \) on blade PS near blade tip in which the flow pulls the blade rotating. A counter clock-wise vortex exists in this region and is not broken completely at rotor outlet. Jang et al. (2001) [12, 13] reported that tip vortex or tip leakage vortex exists on a blade SS near blade tip which moves to downstream during the flow diagonally across the blade passage. However, the vortex interacts with the next blade of PS and then breaks down before it reaches to the location of blade trailing edge. The similar phenomenon that vortex breakdown occurs in axial compressor rotor was reported by Furukawa et al. (1999) [16].

In Figure 5-8, the flow for Type C at \( \phi = 0.290 \) is quite different from Type A and Type B, although the measurement flow rate is the same. From the analysis result of averaged velocity distribution, the flow field of Type C at \( \phi = 0.290 \) is similar to the one near stall condition. The axial velocity becomes small near blade tip and the main flow region is limited around blade mid-span region. In Figure 5-8 (b)-(d), circumferential velocity enlarges and velocity fluctuation becomes high in the region near blade tip from blade PS to blade mid pitch in which something blocks the axial flow.

5.2.4 Result discussion

The performance test and the internal flow measurement at rotor outlet were carried out for three kinds of inlet geometry for a propeller fan designed by only considering ducted structure. The effect of these inlet geometries on fan performance and three dimensional velocity fields at rotor outlet were discussed. From the results of fan test, the performance curve cannot be improved by opening the casing area of inlet geometry. Half-ducted type propeller fan can obtain the most uniform axial velocity in the widest main flow region along radius in the three kinds of inlet geometry, and its radial velocity is the closest to zero in design flow rate condition.

On the other hand, the inlet geometry gives circumferential velocity the heaviest influence in three velocity components. As the open area of blade tip enlarges, the circumferential velocity is influenced at least by flow rate and its distributions become uniform along radius which is benefit
for obtaining energy at outlet of rotors. However, the flow with a strong radial velocity component flows into the blade passage passing through blade tip, and the pressure-rise becomes small and the stall point shifts to large flow rate condition. The basic mechanism to make vortex is almost the same, that is, the flow interaction between the main flow and the radial inflow passing through blade tip. The vortices are formed by both of half-ducted and semi-opened fan, and they are tip leakage vortex and tip vortex respectively. The circumferential velocity of the half-ducted type becomes large in low flow rate 0.23 near the tip regions which has been affected by vortex in that region. As the open area of blade tip enlarges, the blade loading increase due to the circumferential velocity increasing, and the tip vortex strengthens at outlet because increasing radial inflow interacts with the main flow strongly when flow rate is no less than design flow rate. When flow rate is larger than design flow rate, the air flow turns to radial inwards and the flow fields becomes contraction.

5.3 Performance and flow fields of semi-open propeller fan

In this section, the experimental investigation has been carried out for a semi-open type propeller fan in order to clarify the effect of inlet bell-mouth size on fan performance and flow fields around rotor. As mentioned above, semi-open axial fan is the one that it doesn't have the duct upstream and downstream of the fan, but only has the casing at blade tip. And moreover, one part of rotor blade tip is covered and the other part is open. The rotor blade of semi-open fan has long chord length at blade tip and short in the hub which dedicate that the main work is done at the blade tip areas. On one hand, the semi-open structure will cut down the amount of material use which also meets the requirement of compact structure of fans. On the other hand, since the part of the rotor is not covered, the radial velocity component is included in the inflow that will not only complex internal flow of the fan but also has a significant effect on the tip leakage flow. The bell-mouth is equipped at fan inlet to suck air into the fan more smoothly. As the simple structure and the merits of semi-open type, the flow rate is relatively larger. So it is mainly applied for ventilation and cooling systems. The semi-open propeller fan is different from the full-ducted compressors and fans which have many researches, but it owns lots of merit make itself competitive. Recently, for the sake of
energy conservation and noise reduction in house, high performance and low noise have been strongly desired on this kind of fan. In order to investigate directly effect of the inlet shape on its flow field, six types of R-size bell-mouth were measured by an I-type hot wire and a slanted hot wire at inlet and outlet regions respectively.

![Fig.5-9 Schematic of hot wire survey locations](image)

![Fig.5-10 Performance curves](image)
Figure 5-9 shows the schematic diagram of the hot wire test, and the distribution of test points of the propeller fan at upstream and downstream are illustrated. The tested propeller fan is covered by the rear part of the blade tip area with half chord length in axial direction of rotor, while the front part are set six types of bell-mouth as the special inlet geometry to lead the inflow. The bell-mouth curves are circular-arc with the radius of 0 (R/L=0) to 25mm (R/L=0.5, which is relative to 50% of blade chord) at the interval of 5mm. Ensemble average data at inlet is obtained from the single I-type hot wire probe which is set at two crossed directions of one direction normal to meridional plane. The test points were set from RF0-RF3 in front region of rotor and RO1-RO5 on top region of the blade with 5mm interval and also r01-r17 in radial direction. While the hot wire anemometer with slanted probe was used to get the outlet data of three-dimensional velocity component by periodical multi-sample technique. The probe rotates 7.2° once a time and 25 times with the rotor trigger signal so that the velocity can be obtained in different direction. Fifteen measurement points r01-r15 in radial direction are set 3mm downstream of the rotor. In this section, according to 180° rotating range of the probe, the reverse flow can be observed and it will be discuss in the later context. The rotating speed of the rotor is 3000 rpm and the Reynolds Number based on the circumferential velocity at the rotor tip and the chord length at the mid span is 1.32×10⁵.

5.3.1 Performance curves of tested fans

The performance test of a propeller fan with six types of bell-mouth was conducted by the experiment method as explained in Chapter 4 in the wind tunnel in Figure 4-2. The performance test results in Figure 5-10 show the relationship of pressure rise, axial torque and efficiency with flow rate which are all non-dimensional value normalized following Equation (4-1)-Equation (4-4). According to torque curve, it is insensitive to the condition change of inlet flow due to the variation of bell-mouth construction within the measuring range of flow rate. For the ψ-φ curve, the pressure rise becomes lower mainly at the non-stall condition, the stall point shifts to higher flow rate side as the openness of the bell-mouth increases, and the stall point could be hardly observed when the bell-mouth radius is 10% of blade chord (R/L=0.1) or zero. For the η-φ curve, the best efficiency becomes lower and the corresponding flow rate shifts to lower flow rate side, which is in agreement with pressure rise curve. The propeller fan with the smallest openness (R/L=0.5) reached the best
efficiency of 42%. The best efficiency could be kept up 40% for the smaller openness with bell-mouth radius more than 0.3L. While the best efficiency of the propeller fan with bell-mouth $R/L=0.2$ and $R/L=0.1$ rapidly decreased and the propeller fan with the largest openness obtains the best efficiency within the stall region.

In the following section, the internal flow fields are measured in four flow rate, $\phi=0.32$ (in super flow region), $\phi=0.265$ (the design point), $\phi=0.23$ (in the partial flow region) and $\phi=0.21$ (in the stall flow region) according to the measuring method introduced in Chapter 4. And the rotating speed of the propeller fan is 3000 rpm.

5.3.2 Circumferentially averaged flow field at fan inlet

Figure 5-11 and Figure 5-12 show the contour maps of circumferentially averaged velocity (left side) and its deviation (right side) for the rotor inlet region on the meridional plane measured by hot wire anemometer. The contour maps only show the measurement results for bell-mouth size of 0.1L, 0.3L, 0.5L. The flow direction of the propeller fan is from left to right of the maps, while the radial direction is in vertical direction. The mark $r^*$ is defined as the ratio of the radial distance to the blade span and the position of $r^*=1$ is on the blade tip. All the velocity presented in this section are normalized into non-dimensional form by the blade tip speed. Figures 5-11 presents the contour maps of the resultant velocity $V_{ar}$ of velocity components in axial and radial direction named meridional velocity and its deviation, respectively. And Figure 5-12 shows the contour maps of the resultant velocity $V_{tr}$ of velocity components in tangential and radial direction and its deviation, respectively.

On the meridional plane shown in Figure 5-11, the meridional velocity outside of blade tip becomes large and the one ahead of rotor becomes small as the bell-mouth size decreases. It is possible that the velocity is divided by regions and the proportion largely depend on the inlet geometry. See the contour line of meridional velocity valued 0.11, the distribution region more than 0.11 is continuous from hub to the onset of the bell-mouth, which indicates that the main inflow regions not only include the region ahead of rotor but also involve the tip bell-mouth region. The contour line valued more than 0.13 in meridional velocity contour has been gradually restricted in a smaller and smaller region as the bell-mouth size decreases, which means that the averaged inflow
velocity makes a drop in the same flow rate condition. The corresponding deviation on the right side has synchronous tendency with a little strong variation in larger velocity region, and the stronger region is above the rotor rather than in front the rotor.

Fig.5-11 \( V_{\text{var}} \) (left side) and its deviation (right side) distributions at fan inlet (\( \phi = 0.265 \))
Fig. 5-12 $V_{tr}$ (left side) and its deviation (right side) distributions at fan inlet ($\phi=0.265$)
In **Figure 5-12**, the velocity components in circumferential and radial direction is small ahead the rotor which indicates that uniform inflow in axial direction has occurred here. However, these velocity components become much larger near the bell-mouth region, that is, the fluid is imported into the rotor in circumferential direction above the rotor. And the fluid will be imported starting from the onset place of the bell-mouth and be along its arc line. Compared to meridional velocity distributions in **Figure 5-11**, the velocity $V_{tr}$ is relatively weak near the bell-mouth region. Above the rotor, the resultant velocity $V_{tr}$ become smaller with the bell-mouth radius decreasing, and its distribution areas tend to be gradually limited near the bell-mouth arc. For the deviation distributions of $V_{tr}$, the intense fluctuation of $V_{tr}$ is near the corner region between the rotor tip and the bell-mouth, where is the right place of the intersection region of axial and radial inflow. In a word, the bell-mouth plays a crucial role not only leading the flow but also controlling the beginning position and intensity distributions of inflow.

### 5.3.3 Phase locked average velocity field at fan inlet and outlet

**Figure 5-13** shows the contour maps of phase locked average velocity on the section surface of RF0 (as shown in **Figure 5-9**) at flow rate of 0.265. It illustrates the distributions of resultant velocity $V_{ar}$ (left side) and $V_{tr}$ (right side) for the propeller fan with bell-mouth size of 0.1L, 0.3L, 0.5L, respectively. The rotating direction of the propeller fan is from left to right, while the vertical direction is in the radial direction. The mark $r^*$ has the same mean as explained for **Figure 5-11**.

The same conclusions can be achieved by the analysis of the distributions of phase locked average velocity and circumferentially averaged velocity. Near the blade leading edge, the velocity of $V_{ar}$ fiercely congregated near the tip area which indicates that the main flow region is formed here and the inflow runs parallel to axial direction, and also $V_{ar}$ decreases as the bell-mouth radius decreases. While $V_{tr}$ approaches to really small values near the blade surrounding areas and relatively large above the central location of the blade passage. It is essentially necessary to reach more detailed discussion of $V_{tr}$ into decomposition velocity in tangential and radial direction components for profound comprehension of inflow and its most interesting place near blade tip region. Furthermore, the distributions of $V_{tr}$ increasingly concentrate on the leading edge near suction surface as the bell-mouth size decreases, which will have an effect on main flow.
The distributions of corresponding velocity fluctuation keep the same tune with contour maps of velocity so that the figures are omitted in the text. As it is observed above, the bell-mouth plays a crucial role of leading in the fluid, and effects the starting position and intensity distributions of fluid import. The arc length of bell-mouth has a main influence on distributions of resultant velocity $V_r$ that the smaller curvature makes it uniformly distribute along the blade cascade. Furthermore, the fluid enters into the rotor in radial direction and meets with the main flow which will cause acute fluctuation and loss of axial velocity. However, the radial velocity component doesn't directly

Fig.5-13 Contour maps of phase locked average velocity at RF0 plane at $\phi=0.265$ ($V_{ar}$ left side / $V_r$ right side)
affect the loss of axial velocity largely.

![Contour maps of phase locked average velocity and deviation at outlet at $\phi=0.265$ (R/L=0.5)](image)

**Fig.5-14** Contour maps of phase locked average velocity and deviation at outlet at $\phi=0.265$ (R/L=0.5)

![Contour maps of phase locked average axial velocity at outlet at $\phi=0.265$ (R/L=0.1)](image)

**Fig.5-15** Contour maps of phase locked average axial velocity at outlet at $\phi=0.265$ (R/L=0.1)
**Figure 5-14** shows the distributions of phase locked average velocity and deviation at outlet section plane shown in **Figure 5-9** in the flow coefficient of 0.265 for the propeller fan with bell-mouth $R/L=0.5$. In **Figure 5-14 (a)**, $V_r$ has both positive and negative values which means the fluid flows inwards on pressure surface and flows outwards on suction surface along radial direction. $V_t$ is obviously larger than $V_a$ at outlet as shown in **Figure 5-14 (b-c)** which means rotating flow dominates the outlet flow field especially near upper span areas. As Inoue [77] said the turbomachine is a generator of vortices, there must be some vortices generated near the rotor tip and moreover they dominate the outflow field. The tip leakage vortex and the tip vortex might contribute to this phenomenon. It can also refer to the distribution of deviation at outlet in **Figure 5-14 (d)**.

**Figure 5-15** shows axial velocity of the propeller fan with bell-mouth $R/L=0.1$ at outlet which is remarkably larger comparing to **Figure 5-14 (b)**. And the axial velocity of the propeller fan with bell-mouth between them is less than the former and more than the latter, that is, the axial velocity inside main flow region at outlet becomes higher even though the flow rate is the same as the bell-mouth size decreases. It indicates that more static energy can be obtained by rotor with larger bell-mouth which is in an agreement with the performance curves shown in **Figure 5-10**, and also
the reversed flow may occur at lower axial velocity region around main flow region. Figure 5-16 shows the distributions of $V_t$ for the propeller fan with bell-mouth $R/L=0.3$ and $R/L=0.1$. $V_t$ for the propeller fan with bell-mouth $R/L=0.3$ is especially larger whatever compares to Figure 5-16 (b) ($R/L=0.1$) or Figure 5-14 (c) ($R/L=0.5$). This unique phenomenon may be caused by the mix between the tip leakage vortex and the passage vortex induced by the secondary flow.

5.4 Summary on effect of inlet geometry

The effects of the inlet geometry on fan performance and three dimensional internal flow fields have been analyzed in this chapter. From the results of performance test, the performance of the propeller fan cannot be improved by opening the casing area of inlet geometry. Ducted type propeller fan can obtain the most uniform axial velocity in the widest main flow region along radius in the three kinds of inlet geometry, and its radial velocity is the closest to zero in design flow rate condition.

On the other hand, the inlet geometry gives circumferential velocity the heaviest influence in three velocity components. As the open area of blade tip enlarges, the circumferential velocity is influenced at least by flow rate and its distributions become uniform along radius which is benefit for obtaining energy at outlet of rotors. However, the flow with a strong radial velocity component flows into the blade passage passing through blade tip, and the pressure-rise becomes small and the stall point shifts to large flow rate condition. The basic mechanism to make vortex is almost the same, that is, the flow interaction between the main flow and the radial inflow passing through blade tip. The vortices are formed by both of ducted and semi-opened fan, and they are tip leakage vortex and tip vortex, respectively. As the open area of blade tip enlarges, the blade loading increase due to the circumferential velocity increasing, and the tip vortex strengthens at outlet because increasing radial inflow interacts with the main flow strongly when flow rate is no less than design flow rate. When flow rate is larger than design flow rate, the air flow turns to radial inwards and the flow fields becomes contraction.

The experimental measurement of velocity field has been carried out at inlet and outlet of a semi-open type propeller fan by an I-type hot wire and a slanted hot wire. Six types of bell-mouth
size have been investigated in order to understand the effect of inlet geometry on the performance and flow fields of a propeller fan. The results are summarized as follows.

For the performance characteristics of a propeller fan, the torque curve is almost invariant, the pressure rise becomes lower in the non-stall condition, and the best efficiency becomes lower as the bell-mouth size decreases.

For the internal flow, although the most intensity inflow still in front of rotor, the main inflow regions extend to the tip blade region and it is in axial and circumferential inflow. As openness of bell-mouth increased, the inflow proportion of the tip blade region increases, meanwhile the averaged inflow velocity makes a drop at the same flow rate. The velocity fluctuation intensifies near the clearance between the blade tip and the bell-mouth where is the right place of the intersection region of axial and radial inflow. At outlet, axial velocity inside main flow region become higher even though the flow rate is the same as the bell-mouth size decreases.

The bell-mouth leads the fluid entering the rotor in radial direction and meeting with the main flow, which causes acute fluctuation and the decrease of axial velocity. The bell-mouth size controls the casing covered length which would affect the formation and development of tip leakage vortex. The bell-mouth arc length effects the starting position and intensity distributions of fluid import.
Chapter 6

Comparison of Half-Ducted Design with Ducted Design

6.1 Half-ducted design and ducted design

As we known, most applications of propeller fans are in non-ducted types for the space limitation or cost saving, such as the applications in the ventilation and cooling systems. However, the corresponding design methods for non-ducted types of propeller fans are not studied by many researchers. According to the experimental study in Chapter 5, the inlet geometry and bell-mouth use have some merits for improving the internal flow of propeller fans. But these advantages could not play a big role due to the ducted design method. In this chapter, a comparison will be made between the ducted design method and the half-ducted design method of propeller fans. The controlled vortex design method has been applied for half-ducted and ducted propeller fan design by specifying the constant tangential velocity at both inlet and outlet of the fan rotor. For the ducted design, uniform inflow and outflow have been prescribed as if it were placed in a straight pipe. Taking the real flow into account in the application of no pipe situations, the half-ducted design have considered the radial flows at both inflow and outflow by specifying the flow angles.

6.1.1 Design parameters and procedures

The quasi three-dimensional design method addressed in Chapter 2 has been applied to obtain the blade profile of a ducted and half-ducted propeller fans. The meridional flow and the blade to blade flow are calculated by the method of streamline curvature. Based on the theory, the meridional flow is calculated by adopting the radial balance Equation (2-7), while the calculation of the blade to blade flow is obtained by two dimensional cascade data as explained in Section 2.3. And the blade profile is corrected based on a potential flow theory by considering the axial flow velocity change and the inclination of the flow surface as addressed in Section 2.4.

On the solution of meridional flow, the meridional velocity is evaluated at the quasi-orthogonal direction on meridional plane on the condition of ignoring the compressibility of the fluid. The
energy per second getting through the outlet of the rotor can be calculated by the following equation on the assumption of constant velocity component in tangential direction \( V_t \) at inlet and outlet (zero at inlet):

\[
E_2 = \frac{m}{k_B} \Delta I_{sh} = \int_{q_3}^{q_4} \rho \cdot r_2 \cdot V_{w2} \cdot \cos \varepsilon \cdot u_2 \cdot V_{t2} dq
\]  

(6-1)

Therefore, \( V_{t2} \) can be obtained. The total pressure rise is presumed to be able to calculate by the Euler equation as follows:

\[
P_t \simeq \Delta P = \rho \cdot \eta \cdot u_2 \cdot V_{t2}
\]  

(6-2)

As so far, the meridional velocity and the tangential velocity can be evaluated so that the calculation of meridional flow is finished. The blade profile on the revolving stream surface is selected by referring to the diagram of circular arc carpet added at the end of this Chapter. The circular arc blade with equal thickness and quadrilateral blade on the meridional plane is adopted.

### Table 6-1 Design Parameters

<table>
<thead>
<tr>
<th>Designed propeller fans</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Tip diameter</td>
<td>( D_t )</td>
</tr>
<tr>
<td>Hub-tip ratio</td>
<td>( D_h/D_t )</td>
</tr>
<tr>
<td>Number of blades</td>
<td>( Z )</td>
</tr>
<tr>
<td>Flow coefficient</td>
<td>( \Phi )</td>
</tr>
<tr>
<td>Pressure coefficient</td>
<td>( \Psi )</td>
</tr>
<tr>
<td>Rotational speed</td>
<td>( n )</td>
</tr>
<tr>
<td>Specific speed</td>
<td>( N_s )</td>
</tr>
<tr>
<td>Efficiency</td>
<td>( \eta )</td>
</tr>
<tr>
<td>Blockage coefficient</td>
<td>( k_B )</td>
</tr>
<tr>
<td>Air density</td>
<td>( \rho )</td>
</tr>
</tbody>
</table>

Table 6-1 shows the design parameters which are same specification for the half-ducted and ducted design of propeller fans. For the half-ducted design, the leading edge and trailing edge near...
blade tip are modified into spline curve, which can be seen in Figure 6-7(a). The flow rate, pressure rise and efficiency are shown in non-dimensional form by Equation 4-1, Equation 4-2 and Equation 4-4.

6.1.2 Blade profiles and streamlines of designed propeller fans

Figure 6-1 shows the streamlines on meridional plane and blade profiles of the designed half-ducted and ducted propeller fans. The designed blade shapes on top view are presented on the right side of figures. The highly twisted blades can be avoided in half-ducted design method of the controlled vortex. For the ducted fan, the streamlines on the meridional plane are uniform and parallel to each other, as shown in Figure 6-1(a) on left side. While the streamlines of half-ducted fan are obviously different due to the inclinations of inflow and outflow shown in Figure 6-1(b) on left side. That is, the radial velocity component has been taken into account in half-ducted design by specifying the flow angles according to the former experimental data [10, 11]. The flow angles of the streamlines are 63°, 16° and 38° for tip streamline at inlet, tip streamline at outlet, hub streamline at outlet, respectively. The geometrical shapes of blade on meridional plane, the flow passage and the design parameters are referring to the investigated propeller fan in Chapter 5 for the sake of easily carrying out measuring test after design. The same rotating speed 3000 rpm is adopted which is relatively high speed. Therefore, the hub ratio has to be set a little larger to avoid the flow focus on the blade tip area.

The blade profile data of designed propeller fans are obtained by making program in Fortran 77. The analysis of the three-dimensional internal flow fields are conducted comparing to the numerical computation results in the commercial software. The geometrical creation and grid generation for the numerical simulation have been described in Chapter 3. The numerical calculation also take the mesh dependence into account. More than 2 million mesh has been applied to obtain the flow in general accuracy. The ratio between the energy $L_1$ obtained by the propeller fan and the theoretical power $L_2$ can reach above 0.96. The expressions of $L_1$ and $L_2$ are as follows,

$$L_1 = T_D \cdot \omega = T_D \cdot 2 \cdot \pi \cdot n$$

$$L_2 = Q_D \cdot g \cdot H_{th}$$

(6-3)  
(6-4)
Here, $T_D$ and $Q_D$ are the values of torque and mass flow rate from CFD calculation. $n$ is the rotating speed. $H_{th}$ is Eular head evaluated in Equation (6-5) in which $\langle u_1 v_1 \rangle_{ave}$ and $\langle u_2 v_2 \rangle_{ave}$ are the averaged sum of circumferential velocity multiplying tangential velocity on eleven positions from rotor hub to casing at inlet and outlet, respectively.

### 6.2 CFD performance of propeller fans

As a result of the numerical simulation, the efficiencies of ducted fan and half-ducted fan are 51.31% and 49.96% according to Equation (4-4), respectively. Both of the design methods have nearly approached to the designed efficiency value 50%. However, the static pressure rise in half-ducted design is much higher than that of the ducted design propeller fan. The difference between them could reach to 22.75 Pa, which is a relatively substantial pressure rise for a propeller fan with a diameter of 200mm. Therefore, the half-ducted design which is considering the radial velocity for inflow and outflow is much better than the ducted design for the improvement of the propeller fan performance. In order to make it more believable, the three-dimensional flow field of half-ducted design propeller fan will be investigated in comparison with that of ducted design propeller fan.
The following sections will analyze the velocity and pressure distributions of internal flow in designed ducted and half-ducted propeller fans. The discrepancy of flow field caused by these two design methods will be clarified by the numerical analysis.

6.3 Internal flow of designed propeller fans

6.3.1 Velocity distributions of designed propeller fans

Figure 6-2 and Figure 6-3 present the distributions of circumferentially averaged meridional velocity and tangential velocity of half-ducted and ducted propeller fans at inlet and outlet in terms of numerical results and design data, respectively. The abscissa is in the radial direction, and 0.6, 1 are at hub and tip of blade. The circled lines and the lines with diamonds denote the data from design and numerical calculation, respectively. And solid and hollow types of circle and diamond stand for half-ducted and ducted propeller fan, respectively. The meridional velocity of the ducted propeller fan is specified nearly uniform both at inlet and outlet, however, it has smaller value near mid-span at inlet and decreases at hub and tip region at outlet according to the calculated results shown in Figure 6-2. The meridional velocity of the half-ducted propeller fan is prescribed not uniform as the ducted propeller fan but increases from hub to tip. Although the calculated meridional velocity is a little lower than the design value, they are almost going in the same tendency.

In Figure 6-3, tangential velocities of half-ducted and ducted propeller fans are prescribed as constant, the one of half-ducted propeller fan in terms of numerical calculation is much closer to the constant prescribed value both at inlet and at outlet than that of the ducted propeller fan. Take the prescribed constant as the average of tangential velocity, then calculate the standard deviation of computational value in CFD to judge the degree of divergence from design. The standard deviations of the tangential velocity at inlet in Figure 6-3(a) are 0.0726, 0.0322, and the ones at outlet are 0.1021, 0.0851 in Figure 6-3(b), respectively for the ducted and half-ducted propeller fans. Tangential velocity of the half-ducted propeller fan at inlet is uniform and near zero in the numerical calculation which follows the design. While it deviates from design value at outlet especially near the hub. According to Euler equation as shown in the Equation (6-2), the tangential
velocity especially at outlet has a significant effect on pressure rise.

In order to find the reason that the tangential velocity is divergent from the design value, the distributions of flow angles which are the intersection angle of relative velocity and meridional plane at inlet and outlet are investigated as shown in Figure 6-4. The lines with hollow circles and hollow diamonds are the distributions of the inlet flow angle $\beta_1$ in terms of design and numerical calculation, respectively. And the lines with solid circles and solid diamonds show the distributions of the outlet flow angle $\beta_2$ in terms of design and numerical calculation, respectively. The deviations of the flow angles at inlet and at outlet are below 4.5 degree except the points near hub and casing at outlet. Thus the difference between the tangential velocity of the half-ducted fan and its design value at inlet and outlet is relatively smaller, especially at inlet and the standard deviation is 0.0322. However, the situation is a bit difference for the ducted fan, the inlet flow angles does not differ so much from the designed condition of uniformed axial inflow as seen in Figure 6-4 (b), which improves the outflow angles near the hub. However, in the dominate flow region the outflow angles are divergent from designed data, which also make tangential velocity at outlet have the standard deviation of 0.1021.

Figure 6-5 and Figure 6-6 present the distributions of velocity vector on a meridional plane and a section plane perpendicular to flow axis. The meridional plane is set between blade and blade, and locates near the leading edge of pressure surface and the trailing edge of next suction surface. The section plane is set at 1mm upstream of the blade leading edge. The inflow of half-ducted fan is shown more uniform than that of the ducted fan which is in agreement with the analysis of velocity distribution at inlet. The vortex are marked by arrows in Figure 6-5 (b) and Figure 6-6. The blade is rotating from suction surface to pressure surface, while the clockwise vortex rotates in reverse due to the pressure difference between the two blade surfaces. The counter clockwise vortex indicates a big reverse flow produced at inlet of ducted fan. They cause the tangential velocity to increase as shown in Figures 6-3 (a) and the meridional velocity to decrease as shown in Figures 6-2 (a), respectively.
Vt/U -

Fig.6-2 Circumferentially averaged velocity on meridional plane

Vt/U -

Fig.6-3 Circumferentially averaged tangential velocity component

Fig.6-4 Distributions of flow angles
6.3.2 Pressure distributions of designed propeller fans

Figure 6-7 and Figure 6-8 show the distributions of static pressure on the suction surface and the pressure surface, respectively. For energy generating turbo-machinery rotor, the static pressure on suction surface is negative pressure and increases from the leading edge to the trailing edge. However, the static pressure on suction surface of the ducted design propeller fan decreases from...
the leading edge to the one third of chord length and increases from the mid chord to trailing edge, and it produces a high pressure center near leading edge as shown in Figures 6-7 (b). In Figures 6-8(b), the static pressure on pressure surface of the ducted design propeller fan decreases to negative near the leading edge which weakens the suction performance of suction surface and makes the flow twist at inlet region as seen in Figures 6-5 (b) and Figures 6-6. While the performance of the half-ducted design propeller fan is better than that of the ducted design propeller fan, and the static pressure uniformly increases on pressure surface from hub to tip in radial direction which follows the assumption in design in Equation (6-2).

**Figure 6-9** presents circumferentially averaged total pressure of half-ducted and ducted design propeller fans at outlet along radial direction. The total pressure of the half-ducted design propeller fan is larger than that of the ducted design propeller fan except near casing and hub regions. As seen in Figures 6-2 (b), the meridional velocity dominating the flow field at outlet makes the mid-span region the most important part of the flow, which is beneficial to the energy acquisition. Therefore, the static pressure rise of the half-ducted design propeller fan would be able to surpass that of the ducted design propeller fan even they are designed by specifying the same flow parameters.

Half-ducted and ducted design propeller fans have been numerically analyzed for investigating the effect of the consideration of radial flow in design on the overall performance and the three dimensional internal flow of propeller fans. The half ducted design could avoid higher twisted blade in design process. Even though the two kinds of designed propeller fan can reach the specified efficiency, the half-ducted design propeller fan is able to obtain 16.6% more static pressure rise than the ducted design propeller fan. For three dimensional internal flow, the half-ducted design also has a certain number of advantage in the design of propeller fans as mentioned above. Therefore, half-ducted design with considering radial inflow and outflow, which can realize the improvement for the flow angle at inlet and outlet, the distributions of velocity in the flow field and the pressure distributions on the blade surfaces, is feasible and valid in comparison with ducted design for propeller fans.

However, the static pressure rise of the half-ducted design propeller fan is 159.92 Pa which does not reach the design data 200 Pa according to numerical simulation results. The reason is considered that the equivalent diffusion ratio is more than 2.00. Equivalent diffusion ratio was defined by
Lieblein as the ratio between the maximum velocity on the blade and outlet velocity. It has a great effect on the total pressure loss coefficient which can be obtained in cascade test. The loss of total pressure is significant if the equivalent diffusion is more than 2.00. There are two methods to decrease the equivalent diffusion ratio. One is the blade number increasing to share the blade loading for each blade and the other is enlarging the blade area to decrease the blade loading in each blade. In Chapter 7, the latter improvement will be conducted to make it lower so that the static pressure rise could approach to the design value, and then the designed propeller fan will be made and tested in the experimental apparatus as shown in Chapter 4.

![Presure](image)

(a) Half-ducted fan

(b) Ducted fan

Fig.6-7 Static pressure distributions on SS
Fig. 6-8 Static pressure distributions on PS

Fig. 6-9 Total pressure at outlet
Fig. 6-10: Diagram of circular arc carpet for designed camber angle ($\tau_m \leq 1.2\%$)

Fig. 6-11: Diagram of circular arc carpet for designed attack angle
Chapter 7
Verification of Designed Propeller Fan

7.1 Improvement of half-ducted design

As mentioned above, the static pressure rise of the half-ducted designed propeller fan in Chapter 6 has not reach the design value. The reason is considered that the equivalent diffusion ratio is more than ideal value. In order to make it reduce or close to the ideal value, the blade area will be increased by extending the chord length from 40mm to 50mm in axial direction in design. As shown in Figure 7-1, the streamlines distribute on the meridional plane and the blade shape is prescribed as a quadrilateral of $50 \times 40$mm but not a square of $40 \times 40$mm presented in Figure 6-1 (b). All the other design parameters are given the same values as in Chapter 6. After the calculation of meridional flow and blade to blade flow as described in Chapter 2, the blade shape is obtained shown on right of the figure. However, the equivalent diffusion ratio is improved but still more than ideal value.

The numerical simulation has been conducted to the designed propeller fan using the turbulence model explained in Chapter 3. According to the calculation results, the efficiency is 49.07% in the condition of design flow rate and the static pressure rise can reach to 85% of design value which increases about 7% than the design in Chapter 6. The designed propeller fan rotor is processed by Three Dimensional Printer Type No.OBJET30PRO made by Stratasys, as shown in Figure 7-2. The performance test and outlet flow field measurement described in Chapter 4 are carried out to this propeller fan. The test results and numerical simulation analysis will be discussed in the following sections.
7.2 Performance curves

Figure 7-3 shows the performance curves of designed propeller fan. The curves of $\eta$, $\psi$ and $\tau$ are efficiency, pressure and torque against flow rate $\phi$, respectively. All these parameters are in non-dimensional form normalized by Equations (4-1) - (4-4) in Chapter 4. The red curves are obtained from the numerical simulation and the black ones are propeller fan test results in two ways of which flow up is the flow rate increase and flow down is the flow rate decrease in the measuring process. The vertical dash lines are the design flow rate coefficient 0.264 and the flow rate coefficient 0.304. The red solid point is the design point in which the pressure
coefficient is 0.336. The design point is near the stall point according to the performance test. The performance curves of experiment and numerical simulation are different especially in unsteady conditions and have the same tendency in steady conditions.

### 7.3 Internal flow of designed propeller fan

#### 7.3.1 Velocity distributions

The measurement at outlet of designed propeller fan has been conducted using the slanted hot wire measuring system as explained in Chapter 4. The measuring locations are the same radial positions as show in Figure 5-9 but locate at 7 mm downstream of the blade trailing edge. **Figure 7-4** shows the axial and tangential velocity distributions in terms of the data from design, test and numerical simulation along the radial direction at outlet of designed propeller fan at design flow rate 0.264. The scales of 0 and 1 in abscissa are at the positions of hub and casing, respectively. The axial velocity $V_a$ and tangential velocity $V_t$ are non-dimensionalized by rotating speed at blade tip. The red lines are specified values in design process, and the black lines are obtained from hot wire test and three dimensional calculation results respectively corresponding to solid circle and hollow circle.

In **Figure 7-4 (a)**, the tested and calculated axial velocity lines almost approach to design values except lower values near hub and casing regions. The lower axial velocity is considered to be the cause of hub separation flow and tip leakage vortex. Even though the lower axial velocity in the test is a little more severely reducing from the blade mid-span, the test result still reaches agreement very well with the calculated result. In **Figure 7-4 (b)**, the increasing tangential velocity can be seen from mid-span to blade tip in the test and calculation lines in which they have the same tendency. It might be the tip leakage vortex and separate flow region on suction surface moving with blade make circumferential velocity increase. However, they are different near hub that the tangential velocity enlarges in numerical simulation and minimizes in the test.
According to the performance curves in Figure 7-3, the design point is in unsteady condition. Therefore, two relatively higher flow rate are taken to investigate the outlet flow field. One is in stall condition $\phi=0.28$ and the other is in steady condition $\phi=0.304$. Figure 7-5 shows the distributions of circumferentially averaged meridional velocity $V_m$, axial velocity $V_a$, tangential velocity $V_t$ and radial velocity $V_r$ at outlet of designed propeller fan. The tendency of these velocity distributions in three conditions are almost the same. But generally speaking, in steady condition $\phi=0.304$ it has larger the axial velocity, smaller tangential velocity and radial velocity than the other unsteady conditions. The radial velocity distributions in three conditions are all below the design data that is better for the flow of the propeller fan. Therefore, the distributions of axial velocity and tangential velocity in steady condition $\phi=0.304$ are presented in Figure 7-6. According to the design values, axial velocity and tangential velocity in steady condition $\phi=0.304$ have almost the same distributions as in Figure 7-4. However, the axial velocity is a little bigger and the tangential velocity is more uniform at outlet in steady condition $\phi=0.304$. 

**Fig.7-3** Performance curves
Fig. 7-4 Distributions of velocity on design flow rate

Fig. 7-5 Distributions of velocity at $\phi = 0.264, 0.28, 0.304$
From the above analysis, the designed propeller fan is near stall condition in design flow rate, the overall velocity distributions are below the design values especially near hub and casing regions but follow the designed tendency along radial direction. It is considered to be the blockage effect of the hub-corner stall and vortex occurring near blade tip. They will be discussed in the next section.

### 7.3.2 Blockage effect of hub stall

**Figure 7-7** show the contours of static pressure on hub and suction surface. Firstly, the distributions of static pressure on suction surface is improved near leading edge comparing to previous half-ducted design referring to **Figure 6-7**. The low pressure center still exists on suction surface at about one third length of chord. On hub, the static pressure is increasing from inlet to outlet of the propeller fan rotor and below zero pa.

**Figure 7-8** show the contours of total pressure on section of z=0, 10, 15, 25 mm. The coordinate of z is defined in **Figure 7-1**, and z=0 and z=25 are at the mid chord of blades and blade outlet. The ŝSS and ŝPS marked in **Figure 7-8** denote the suction surface and pressure surface of blades. The low energy flow fluid in flow passage of blades, which develops from the suction surface as in **Figure 7-8 (a)**, moves away...
from suction surface, and distributes near mid pitch in \textbf{Figure 7-8 (c)}. Its lower center focuses on the top area and hub regions near suction surface. Observing its moving trajectory along axial direction as seen from \textbf{Figure 7-8 (c)-(d)}, the low energy flow fluid on the top moves in adverse to suction surface meanwhile the higher pressure occurs before it near suction surface, and the one on the hub near suction surface expands to the whole hub area at outlet. The low energy fluid near hub makes the flow velocity decrease and blocks the flow in the flow passage at outlet. \textbf{Figure 7-9} shows the contour of velocity in axial direction at propeller rotor outlet. The velocity of the low energy fluid in flow direction is below zero near hub which means hub stall occurring at outlet. And high energy flow fluid near casing and suction surface as in \textbf{Figure 7-8 (d)} has extremely low axial velocity which indicates it must have high circumferential velocity or radial velocity components.

\textbf{Figure 7-10} shows the distributions of streamlines at rotor outlet which are the same with pathlines in steady calculation. The streamlines of tip vortex are marked by blue color. The tip vortex is another reason for the velocity decrease especially near casing at outlet in \textbf{Figure 7-4, Figure 7-5} and \textbf{Figure 7-6} which are explained in previous section.

\textbf{Fig.7-7} Distributions of static pressure on hub and suction surface
Fig. 7-8 Distributions of total pressure on four sections

Fig. 7-9 Contour of velocity in axial direction at blade outlet
Fig.7-10 Streamlines at rotor outlet

7.3.3 Aerodynamic noise of designed propeller fan

Large Eddy Simulation and the Fowcs Williams and Hawkings (FW-H) equation are used to compute the unsteady flow field and obtain the acoustic signal, respectively, as addressed in Chapter 3. And the Fast Fourier Transform (FFT) has been applied to process the acoustic signal. The time step size and number of time steps for LES are 0.002 and 1150, respectively. And the time step size and number of time steps for acoustic calculation are $4 \times 10^{-5}$ and 1500, respectively. The designed propeller fan has five blades and rotating speed of 3000 rpm, therefore, the blade passing frequency (BPF) is 250 Hz. The maximum frequency in the sampling time is 12500 Hz decided by the time step size and the minimum frequency is 25 Hz decided by the total time in the sampling process. The rotor blade and hub are considered to be the sound source.

**Figure 7-11** shows five points of receivers of which the coordinates are (-0.5, 0, 0), (0.5, 0, 0), (1, 0, 0), (0.707, 0.707, 0), (0.025, 0, 0), respectively. Receiver 5 is placed at outlet of blade rotor on axis and receiver 4 is located 1 m away from the center of rotor on 45° axial line. **Figure 7-12 - Figure 7-15** present the distributions of sound pressure level on receiver 1-5 at the design flow rate $\phi=0.264$ when the acoustic source data are specified as PSSSHub, PSHub, PSSS and PS, respectively (here, PS, SS and Hub are pressure surface, suction surface and hub of blade, respectively). The
\( \overline{\text{ASD}} \) is the abbreviation of acoustic source data. The abscissa is the frequency in the range of \((0, 12500)\) Hz, the ordinate is the sound pressure level which can be calculated by the following equation:

\[
SPL = 20 \log_{10} \frac{p'}{p_0}
\]  

(7-1)

Where, \( p_0 \) is a reference pressure chosen as \(2 \times 10^{-5} \) Pa, and \( p' \) is the pressure fluctuation relative to the background pressure. The unit of sound pressure level is Decibel (dB) and 50-60 dB is responding to the sound of people talk.

The SPL distributions at design flow rate on receiver 1-4 are presented in Figure 7-12 (a-d). Receiver 1 and receiver 2 almost have the same SPL distributions due to their same distance from rotor center, and the high SPL is almost kept in the same discrete frequency on receiver 5 in Figure 7-12 (e). The SPL relatively decreases on receiver 3 and 4 which are 1 m away from rotor center. The total sound pressure level is 47.7dB and 44.6dB on receiver 3 and receiver 4. However, the SPL on receiver 3 which is in the flow direction is relatively larger than that on receiver 4, and so as receiver 2 is compared with receiver 1 in counter flow direction. The total sound pressure level is 94 dB on receiver 5 and the SPL distributions are shown in Figure 7-12 (e). The SPL at five points on the figure are belong to relatively high discrete frequency of 5693Hz, 7905Hz, 9619Hz, 10134Hz and 11403Hz from left to right which is different with the discrete frequency of the blade passing frequency (BPF). However, the discrete frequency in the case of this dissertation is not BPF which might be due to the application of one fifth of rotor by considering periodic flow and might be caused by the hub stall. In Figure 7-12 (f) for the red line, the sound source does not include the hub, the total sound pressure level decrease to 78 dB, and the SPL distribution is below the black line which is the same with the figure in Figure 7-12 (e). Therefore, the hub stall have a great effect on the generation of noise. In order to investigate this reason, the distributions of sound pressure level on five receivers are presented in Figure 7-13 - Figure 7-15 when acoustic sources are specified as PSHub, PSSS and PS.

In Figure 7-13 (a-d), the discrete frequency is disappear on receivers 1-4 when
acoustic sources are pressure surface and hub. The values of SPL at the discrete frequency in Figure 7-13 (e) changed a little on receiver 5 (same with the red line in Figure 7-13 (f)) compared with that when acoustic sources are pressure surface, suction surface and hub in Figure 7-13 (f) black line, that is, the flow near suction surface generates discrete noise and the discrete noise transmit to downstream.

In Figure 7-14 (a-d), the discrete noise increases and the broadband noise decreases in comparison with Figure 7-13 (a-d). In Figure 7-14 (e) (same with the black line in Figure 7-14 (f), the SPL in discrete frequency is lower than that in Figure 7-13 (e). In Figure 7-14 (f), the sound produced by sound source of blade surfaces is lower than the sound produced by sound source of pressure surface and hub, that is, the flow near suction surface a little effect on broadband noise, and the flow near hub greatly contributes to the broadband noise generation.

In Figure 7-15 (a-d), the discrete frequency is disappear on receivers 1-4 when acoustic source is pressure surface and the broadband noise keeps almost the same in comparison with Figure 7-13 (a-d), that is, the flow near suction surface have a great effect on discrete noise and a little effect on broadband noise. In Figure 7-15 (e) (same with the red line in Figure 7-15 (f)), the SPL in discrete frequency is much lower than that in Figure 7-12 (e) (same with the black line in Figure 7-15 (f)). In Figure 7-15 (f), the broadband noise produced by sound source of pressure surface is lower than that produced by sound source of blade surfaces and hub, that is, the flow near pressure surface hardly contributes to the generation of discrete noise.

Compared Figure 7-13 and Figure 7-15, the flow near hub greatly contributes to the generation of discrete noise and broadband noise but the discrete noise dissipates quickly and does not transmit to downstream. Compared Figure 7-13 - Figure 7-15, the flow near suction surface have a great effect on discrete noise and a little effect on broadband noise, and the discrete noise is a little lower than that produced by sound source of hub and transmits to downstream of rotor. As for non-stall condition of flow rate $\phi=0.304$, the same conclusions can be obtained.

Figure 7-16 presents the distributions of sound pressure level on receiver 1-5 at
non-stall condition of flow rate $\phi=0.304$ when the acoustic source data are specified as PSSSHub. In Figure 7-16 (a-d) the broadband noise is a little higher and the discrete noise is a little lower than that in Figure 7-12 (a-d). In Figure 7-16 (f), the same results can be obtained, that is, the flow near hub still greatly effects the noise generation and the effect of the flow near suction surface on the noise generation has been relatively improved at non-stall condition.

**Fig.7-11** Schematic of receiver positions
(a) Receiver 1

(b) Receiver 2

(c) Receiver 3

(d) Receiver 4

(e) Receiver 5

(f) Receiver 5

Fig. 7-12 Sound pressure level on five receivers
($\phi=0.264$, ASD=PSSSHub)
Fig. 7-13 Sound pressure level on five receivers
\(f=0.264, \text{ASD} = \text{PSH}_{ub}\)
Fig. 7-14 Sound pressure level on five receivers

(\( f = 0.264 \), ASD = PSSS)
Fig. 7-15 Sound pressure level on five receivers
($\phi=0.264$, ASD=PS)
Fig. 7-16 Sound pressure level on five receivers
\( (\phi=0.304, \text{ASD}=\text{PSSSHub}) \)
Chapter 8
Conclusions

8.1 Dissertation conclusions

In order to study half-ducted propeller fan on its internal flow and aerodynamic performance, firstly the effects of the inlet geometry on propeller fan performance and three dimensional internal flow fields were analyzed by experimental method, secondly the half-ducted propeller fans were designed by the comparison of half-ducted and ducted design method, then the designed propeller fan was verified by performance test and outlet flow field measurement in a wind tunnel, finally the internal flow and aerodynamic performance were analyzed by numerical simulation.

Firstly, in the investigation on the inlet geometry of the propeller fan, the half-ducted and semi-open structures were investigated, and internal flow fields of a propeller fan with six types of bell-mouth size were surveyed by experimental measurement of hot wire anemometer in order to improve the radial inlet flow. Some conclusions can be summarized as follows:

1. The performance of the propeller fan cannot be improved by opening the casing area of inlet geometry due to the ducted design method of it.
2. The inlet flow with radial component is an outstanding characteristic in a non-ducted propeller fan. This characteristic makes the blade loading increase, the tip vortex strengthen and the flow fields become contraction at outlet.
3. The larger bell-mouth size can obtain relatively better performance characteristics and more improved internal flow for a propeller fan, which is because the bell-mouth of bigger radius leads the fluid entering the rotor in radial direction smoothly and its larger size controls the casing covered length which would affect the formation and development of tip leakage vortex.

Then, based on the conclusions of inlet geometry study, the half-ducted design
method for a propeller fan was proposed by specifying the flow angles according to experiment data. In comparison of half-ducted design with ducted design, the summaries are as follow:

1. Highly twisted blade can be avoided by the half ducted design method of the propeller fan.
2. The half-ducted designed propeller fan has better performance characteristic, improved velocity flow field and blade loading distributions.

Finally, the half-ducted designed propeller fan designed by the second time was made by three dimensional printer and tested in the wind tunnel, and the internal flow and aerodynamic performance were analyzed. The results are as follows:

1. The pressure performance was improved comparing to the first time. The designed propeller fan is near stall condition at the design flow rate according to test results. However, the half-ducted design is still feasible and practicable comparing to the traditional ducted design for a propeller fan.
2. According to numerical simulation, the low energy flow fluid concentrated near hub region blocks flow in the flow passage and makes energy loss. And the high energy fluid with very low axial velocity focused near casing and suction surface are considered to be vortex.
3. The total sound pressure level is 93 dB and 47.7dB at outlet of fan rotor and 1 m away from rotor center at design flow rate. The discrete frequency is not blade passing frequency (BPF) and high in this study which might be due to considering one fifth of rotor flow passage for simulation and caused by large fluctuation flow especially the hub stall. The dissipation of noise is lower in flow direction than that in other directions.
4. The flow near hub greatly contributes to the generation of discrete noise and broadband noise but the discrete noise dissipates quickly and does not transmit to downstream. The flow near suction surface have a great effect on discrete noise and a little effect on broadband noise, and the discrete noise is a little lower than that produced by sound source of hub and transmits to downstream of rotor. The flow
near pressure surface hardly contributes to the generation of discrete noise.

8.2 Research prospects

Make effort to improve the equivalent diffusion ratio below 2 in design process and design a half-ducted propeller fan with high performance and low noise.
Acknowledgment

I would like to express my gratitude to all the people who gave me help during the writing of this dissertation and the period of my studying in Saga University.

My deepest gratitude goes first to Prof. Yoichi Kinoue, Prof. Toshiaki Setoguchi, Prof. Shigeru Matsuo and Associate Prof. Norimasa Shiomi for their constant encouragement and guidance. They not only walked me through all the stages of writing this dissertation but also took good care of me in my life in Japan. Without their consistent and illuminating instruction, this dissertation could not be completed in present form. Meanwhile, I should express my great gratitude to Prof. YingZi Jin from ZheJiang Science and Technology University, who gave me her earnest teaching and this chance to study in Japan.

Second, I would like to give my great thankfulness to the people for helping me to finish this dissertation. Doc. Miah Md. Ashraful Alam who works in Institute of Ocean Energy, Saga University, helped me revising the manuscript and gave me a lot of useful advice. Satoshi Hiwatashi and Hajime Enohata, Takuya Yanagawa and Shunichi Isozaki gave me a great help in experiment and design process, respectively. The designed propeller fan was made by three dimensional printer in Department of Mechanical Engineering, Matsue College of Technology. I hereby express my great gratitude to Associate Prof. Manabu Takao, and Sasagawa foundation for supporting my research.

Then, a special gratitude should give to all the students in the laboratories of Fluid Mechanics. Especially Ryutaro Ohhira, Junji Nagao, Satoshi Yoshijima, Takashi Nakabaru, Yusuke Oka, Shogo Nakamichi, Kazuyuki Yokoo, Yushiroh Nishiyama, et al., thank you very much for your kind help and I really enjoyed my time with each one of you. It will be a great memory in my life during the time in Saga with you.

Last but not least, I own much to my beloved parents for their continuous support, understand and encouragement.
References


129


