CHAPTER 9

Design and optimization of Turbo compressors

C. Xu & R.S. Amano

Department of Mechanical Engineering,
University of Wisconsin-Milwaukee, USA.

Abstract

An airfoil was referred to raise static enthalpy and pressure. A successful compressor design greatly benefits the performance of the whole power system. Lean design methodologies have been used for industrial power system design. The compressor designs require benefit to both OEM and customers, i.e. lowest cost for both OEM and end users and high efficiency in all operating range of the compressor. The compressor design and optimization are critical for the new compressor development and compressor upgrade. The design experience and design considerations are also critical for a successful compressor design. The design experience can accelerate compressor lean design process. An optimization process is discussed to design compressor blades in turbo machinery. The compressor design process is not only an aerodynamic optimization, but structure analyses also need to be combined in the optimization. This chapter discusses an aerodynamic and structure integration optimization process. The design method consists of an airfoil shape optimization and a three-dimensional gradient-based optimization coupled with Navier–Stokes solvers. A model airfoil of a transonic compressor is designed by using this approach, with an efficiency improvement. Airfoil sections were stacked up to a three-dimensional rotor blade of a compressor. The efficiency is improved over a wide range of mass flow. The results indicate that the optimization process can provide improved design and can be integrated into a compressor design procedure. Initial design considerations of turbo compressor are commonly performed with experience base, although computer technology and numerical methods have made a significant progress. Three-dimensional computational fluid dynamics codes are still not major design tools for compressors. Major design systems for industrial compressor design are largely based on one-dimensional mean line and two-dimensional through flow tools. The experience of designers is one of the key factors to drive design success. The discussions and information
on compressor design considerations in open literature are also reviewed. This chapter consists of three parts: Part I provides an overview of the turbo compressors and basic design procedures. Part II discusses a blade optimization process. An axial compressor blade design was used as an example to illustrate the optimal design. Part III provides some design experience of centrifugal compressors.

1 Part I: Compressors and their design

1.1 Introduction

Compressors are devices to pressurize working gases. One of the basic applications for the compressors is to increase the gas pressure and deliver gas to the downstream of the compressor. For example, compressors can provide high-pressure gas for combustion, high-pressure gas to drive gas tools, transport process gas through pipelines, or compressor gas for heat recovery or gas conditioning circles.

Compressors can be separated into two basic types: The positive displacement and continuous flow compressors. The reciprocating and rotary compressors are two major positive displacement compressors. Ejectors and turbo compressors are the two main continuous compressors. Axial and centrifugal compressors are the two basic types of turbo compressors. The main purpose of this part is to describe an industrial turbo compressor design process.

Gas turbine and compressor manufactures are competing to produce more efficient and wide operating range machines. Correspondingly, many research works and developments were carried out to meet the manufacturers’ needs [1–8]. Compressor blade design is one of the critical processes to ensure the overall compressor have good performance. For all the turbo compressors, there are two major parts: rotating blade and stationary blade. The basic functions of compressor moving blades are added the work to gas and turn the gas to the required angles to have diffusion in the flow path to increase the static pressure and discharge to next component. For the stationary blades, the basic function is to turn the gas to the required angles to have a right diffusion level for the gas. The design goal for modern compressor blades are to achieve the desired flow turning with minimum losses and can tolerance high incident within the constraint of geometric orientation of the blade row requirements defined by overall compressor design.

Both rotating and stationary blades involve the airfoil and blade designs. The basic compressor blade performance was mainly determined by five key factors: shape of the blades, airfoil or blade section stagger angle, solidity, inlet and discharge flow angle, airfoil or blade section loading. Unlike turbine blades through which flows are accelerated, the adverse pressure gradient due to the flow diffusion in compressor blades. The adverse pressure is an unfavorable force on the boundary layer development. It is very difficult to achieve a thin boundary layer. The boundary layer separation often happens inside the compressors. Therefore, the main challenge for compressor blade design is to generate a blade shape to meet the flow angle requirements and reasonable loading distributions.
Efficiency had been a major criterion in the design of compressor and turbine components for many years. The search for maximum efficiency is traditionally made by successive modifications of the geometries and verification by wind tunnel tests and flow calculations. Such a process can be time consuming and outcome strongly depended on the designs’ experiences. This approach is difficult to have advanced design such as shock-free transonic blades and blades with optimized dihedral.

Many design methods were proposed in both research stage and industrial applications [1–9]. The blade design methods can be categorized into two basic methods. One is direct design and another is inverse design method. For direct design method, blade was design first based on the designer experience and then a Navier–Stokes equation code or a fast inviscid code was used to evaluate the airfoil performance. The process of design and performance calculations are completed until a satisfy airfoil was built. The designed airfoils are then stack-upped to produce a three-dimensional blade. Normally, some three dimensional optimization processes are used to continue optimizing the design [1–9]. In inverse design method [5], normally Euler equations with modified boundary conditions were used for an iterative procedure to search the desired design.

With the rapid progress in computational fluid dynamics (CFD) and computer technologies, compressor design has been developed from purely empirical methods to apply more CFD simulation into design process. The CFD method allows the aerodynamic designer to have detailed flow information and to optimize the blades. However, at the beginning stage of the design, the experience data are very helpful to accelerate the design and provide the basic information, to engineers in other discipline. In this chapter, we will start with basic introduction of the compressor, and then talk about design process and optimization, and finally the design experience the author will be discussed.

1.2 Types of turbo compressors

Turbo compressors can be categorized into two types: compressors and turbines. Compressors absorb power to increase the fluid pressure or head, and turbines produce power by expanding fluid to a lower pressure or head. Compressors have a wide application in order to provide the high-pressure gas for combustion in jet engines, to transfer process fluid in pipelines, to provide high-pressure gas to driving tools, etc. The compressor system is one of the important parts of a power system. Both centrifugal compressors and axial compressors are continuous flow compressors.

Compressors are categorized into three types according to the nature of the flow path through the passages of the rotors. When the fluid through flow path is wholly or mainly parallel to the axis of rotation, these types of compressors called axial compressors as shown in Fig. 1a. Centrifugal compressors having through flow paths are wholly or mainly in a plane perpendicular to the rotational axial as shown in Fig. 1b. When the through flow in the compressors has a significant amount of velocities in both radial and axial directions, we call this type of compressors as
mixed flow compressors as shown in Fig. 1.c. Sometimes, mixed flow compressors are also reference as centrifugal compressors.

The axial compressor was first patented by Sir Charles Parsons in 1884 [10]. His compressor concept just simply reversing an axial turbine for use as a compressor; however, the efficiency of compressors from reversing turbines was less than 40%. The difficulties associated with the development of axial compressors stemmed from the fundamentally different nature of the flow process compared with that in axial-flow turbine. It was not until 1926 that any further improvement on axial compressor efficiency was undertake when A.A. Griffith [11] outlined the basic principles of airfoil theory of compressor and turbine design.

Axial flow compressors are the most common types of compressor for aircraft engines and big industrial gas turbine. The gas in an axial compressor flows in an axial direction through a series of rotating rotor blades and stationary stator vanes. The flow path of an axial compressor decreases in cross-section area in the direction of flow, reducing the volume of the gas as compression progresses from stage to stage of compressor blades.

In axial compressors as shown in Fig. 2, the gas being delivered to the face of compressor by the gas inlet duct, the incoming gas passes through the inlet guide vanes (IGVs). Gas entering the first set of rotating blades and flowing in axial direction, is deflected in the direction of rotation. The gas speed is slowed down and turns as it is passed onto a set of stator vanes, following which it is again picked up by another set of rotating blades, and so on, through the compressor. The pressure of the gas increases each time it passes through a set of rotors and stators. The increase in the stage pressure almost inevitably leads to some aerodynamic constraints. The main constraint is increased Mach number, possibly giving rise to shock induced boundary layer separation or increased losses due to poor diffusion of the flow. The maximum stage pressure achieved was about 2.1 based on the recent development.

The aerodynamic principles are applied to the compressor design to increase axial compressor efficiency. The axial compressor blades are treated as lifting surfaces like aircraft wings or propeller blades. The cascade effect is a primary consideration.

Figure 1: Types of compressors: (a) axial, (b) centrifugal, and (c) mixed flow compressors.
in determining the airfoil section, angle of attack, and the spacing between blades to be used for compressor blade design. The blade must be designed to withstand the high centrifugal forces as well as the aerodynamic loads to which they are subjected. The clearance between the rotating blades and their outer case is also very important. The rotor assembly turns at a high speed and must be rigid, well...
Axial compressors have more benefits than centrifugal compressors at high flow and low-pressure ratio compressions. The flow range for most axial compressors is from 10,000 CFM (4.7 m$^3$/s) to 2,000,000 CFM (943 m$^3$/s) and the pressure ratio range is from 1.05 to 15. Some high-pressure ratio multistage compressors can produce pressure ratio over 30 with more than 20 stages.

Centrifugal compressor development was stimulated by aircraft propulsion. Frank Whittle and Hans Joachim Pabst von Ohain built their turbojet engines independently [1]. On August 27, 1939, Hans von Ohain’s engine first time powered an aircraft flight. Shortly later, the Whittle turbojet engine was first flown on May 15, 1941. Both engine designs incorporated a centrifugal compressor to increase the gas pressure. After systematic studies [1], when researchers noticed that for the increasingly larger powered engines required for aircraft propulsion, the axial compressors were preferred. Axial compressor offered a smaller frontal area and relative better efficiency for large gas flow compression. However, centrifugal compressors still offer the advantages for low gas flow rate compressors.

The fluid comes into the centrifugal compressor through an inlet duct and can be given a prewhirl by the IGVs. The IGVs give circumferential velocity to the fluid before it flows inside of the compressor inlet. A positive vane angle produces prewhirl in the direction of the impeller rotation, and a negative vane angle produces prewhirl in the opposite direction. The positive prewhirl decreases the relative inlet Mach number. One purpose of installing the IGVs is to decrease the relative Mach number at the induce tip inlet because the highest relative velocity at induce inlet is
at shroud section. The disadvantage of this application is that a decrease in the relative Mach number also reduces the energy transfer. Another purpose of installing IGV is to increase the compressor operating range. When the compressor flow reduces, the positive prewhirl can reduce the inlet incidence to delay the flow separation to enlarge the operating range.

Centrifugal compressors operate by taking in outside gas and rotating it by means of an impeller as shown in Fig. 1. The impeller, which is usually an aluminum alloy, or steel or even titanium, guides the gas toward the outer circumference of the compressor, building up the velocity of the gas by means of high rotational speed of the impeller. The impeller consists of two basic parts: an inducer such as an axial-flow rotor part, and the blades in the radial direction where energy is imparted by centrifugal force. Impellers have two types: shrouded and unshrouded. The shrouded impeller normally used for a multistage compressor. The unshrouded impeller allows a much higher impeller rotational speed application: for a steel impeller, allowable tip speed would be as high as 450 m/s for an unshrouded impeller but only below 350 m/s for a shrouded impeller. The shrouded impellers are mainly used in multistage compressor for which it would be difficult to maintain an acceptable small clearance between the impellers and stationary shroud.

A significant impact to turbo machinery design and analysis is the introduction of the quasi-three-dimensional calculation methods by Hamrick et al. [12] and Wu [13]. Hamrick et al. proposed a streamline curvature method by handle momentum and continuity equations separately in a stream function; however, Wu developed his matrix method by coupled both equations via a stream function. Both methods introduced stream surfaces of first (S1) and second (S2). Three-dimensional flows were solved two-dimensional way in mathematics. The methods were called Quasi-Three-Dimension method. Wu [13] defined that S1 surface was in the surface from pressure side to suction side and S2 surface is from hub to shroud in impeller. It was assumed that flow only follows the stream surface and there was no flow crossing perpendicular from one surface to another. The losses were taken into account by making use of a prescribed entropy increase, which was obtained from empirical equations.

With the development of the computer technology in mid-1970s, Quasi-three-dimensional calculations and three-dimension viscous solution were developed. Bosman and El-Shaarawi [14] used a computer program to calculate the flow inside of centrifugal compressor rotors. The calculations started with a middle S2 surface at the mid pitch position. The flow quantities were calculated in this stream surface. After calculating the flow characteristics in S1 surface, the streamlines obtained from S2 were rotated until intersecting with the pressure and suction surfaces. Losses inside centrifugal compressors were considered by using a prescribed entropy increase taken from empirical equations. The three-dimensional viscous code was first developed for subsonic flows in a simple geometry [15]. The three-dimensional codes developed at this time were difficult to quantitatively predict the flow inside centrifugal compressor blade.
In 1980s, with the introduction of three-dimensional numerical method to solve the complex geometries with viscous solvers [16–18], the flows inside of the centrifugal compressors were better understood. The turbulence models used very often for centrifugal flow simulations were one-equation or two-equation turbulence models. The secondary flow patterns were simulated and reported. The flow simulation results were agreed with experimental studies in overall flow patterns.

After 1990, considerable progress has been made in understanding the complicated flows inside centrifugal compressors. The detailed flow measurements were available by using Hot-wire, [19, 20] fast response high-resolution pressure transducers [21–25], and LDV and PIV [26]. Strong passage vortices due to secondary flows were found. The impeller and diffuser flow pattern were reported. The detail measurements of the flow inside of the centrifugal compressors were used to guide compressor design.

Recently, with the developments of new method for CFD solution [27–29], many optimization methods started to combine into compressor impeller blade and diffuser airfoil designs. The advance of the modern design will continue to push compressor design toward the wide operating range high stage pressure ratio centrifugal compressor. The CFD simulation had pushed the compressor stage efficiency to a very high level. The future potential of the efficiency improvement is about 2% for state-of-art designs with current manufacturing capabilities. The potential is about 10% if we can have a perfect design. The better three-dimensional design may improve the operating range. However, the compressor efficiency may drop when we improve the operating range.

1.3 Aerodynamic design

The design of the turbo compressor begins with a specification and proceeds through preliminary design to blade design and analysis using design tools. The design process begins with mainline calculation, which is based on the required design flow, and stage pressure ratio or head rise to select the compressor type, number of stages and find the velocity triangles at inlet and outlet of each components of the compressor. Then the performance and basic flow parameters of each component of the compressor are analyzed based on mean line information, then the two-dimensional airfoil sections are designed and stacked up, and finally three-dimensional flow analyses are performed to do further optimization. The objective of aerodynamic design of the turbo machines is to obtain both maximum overall machine efficiency and wide operating range within the limitations imposed by stress, cost, manufacturing and other considerations. Before the 1970s, compressors were mainly designed by empirical methods, based on the accumulated experience of designers and their company. The compressors some were successful design and some time, the designed compressor could not meet efficiency and operating range target. After 1970s, CFD had played an important rule in designing the turbo machinery. Significant improvements in efficiency and operating range have been achieved. Improved experimental measurements have become available over the last three decades [30]. The combinations of the numerical predications.
and detail flow measurements had led to a great improvement in flow physics in the centrifugal compressors.

Recently, a competitive product demands a high level of design optimization of compressor and involves several different disciplines considerable historical design experience and a variety of design tools. Although most of the compressor design systems allow the efficient transfer of design information, how to obtain the design information especially three-dimensional flow field information is still a big challenge. Most of the compressor design involves considerable iterations between aerodynamics, structure and manufacturing cost. During the compressor design, the knowledge of the designers and experience still play an important role. Although many integrated design system and optimization codes permit some design optimizations, many design correlations still need designers’ input based on designers’ experience. Part II introduces an aerodynamic design and optimization procedure. Part III summarizes the authors’ centrifugal compressor design experience to help compressor designers who are in the initial stages of their careers to perform aerodynamic design.

2 Part II: Blade design and optimization

2.1 Introduction

With the development of the CFD, it has been implemented in the turbo machine design process. Great efforts are devoted to improving the efficiency of the gas turbine components. The airfoil designs for turbine and compressor airfoils plays an important role in increasing the turbine efficiency. In the airfoil design, there are two types of implements the aerodynamic design engineer often considers. One is to design and employ custom-design blade profiles with minimum losses and controlled blade boundary layers. The second and even more complex part is to minimize losses resulting from secondary flows near hub and casing. Recently, three-dimensional blade design concepts were proposed to help control secondary flows [31]; however, the complex flow is difficult; to simulate even by using fully three-dimensional Navier–Stokes flow solvers. And the validation of the Navier–Stokes solvers needs lot of experimental data and it is time consuming and expensive. Therefore, almost all the aerodynamic designs are based on the two-dimensional design. The inviscid analyses of the two dimensional airfoil sections still play an important role in the design process. Authors recently developed an airfoil design process based on the Bezer curves to produce the custom airfoil sections based on the flow field requirements of the airfoil [2]. In this study, the airfoil design process was implemented with optimization based on the two-dimension viscous turbulent code.

It was known that for a blade row in an annulus, the stream surface between two annular walls is twisted for most cases. These twists are induced by either shed vorticity or by secondary from arising from inlet vorticity. Stream surface twist can arise in an irrotational flow owing to either span wise components of velocity or span wise blade forces. Many efforts had been adopted to reduce the stream
surface twist and reduce the secondary flow losses, such as sweep [32], lean [33], bow [34] and twist [35, 36] the blades or design a non-axis symmetric end wall. However, there is little information in the available literature for using three-dimensional design and almost no information is available to show how to integrate the three-dimensional features into design process. Most of the study was still on the academic research and was based on the particular machines or blades. Moreover, most of the studies were based on the particular blade and flow situations. For example, Singh et al. [35] argued that closing the blade throat near the end walls could obtain significant efficiency improvements and Wallis and Denton [36] also obtained an efficiency increase from almost the opposite type of blade twist near the end wall. For different machines and different designs, may different techniques should be used according to the flow field nature of the designs. It is very important to design a blade design procedure and optimize the design.

The increased use of CFD tools has been driven mainly by two factors. First, from performance standard point of view, efficiency has steady increased. Second, the turbo machinery industry as a whole has been pushed toward reduced cost designs. The cost reduction is in terms of development, modification, production and operating costs. The cost reduction drives a turbo machine toward high loading in order to reduce stage count, while maintaining or exceeding past performance goals. The current design of the new stage is already outside of the standard airfoil database. Most of the airfoil needs to be designed and the development of the design tools to meet this requirement becomes critical. The application of CFD methodology to improve the turbo machinery design is becoming established within the turbo machinery community. However, only a limited number of publications suggest how to use CFD to help design and modify processes especially during the blade design. This paper serves to present a design process, which contains a novel two- and three-dimensional viscous turbulent code and optimization process.

2.2 Design system

Expensive manpower is invested to find configurations that are stable and efficient in the work range in the turbine and compressor design. One of the most important methods is so called stream-surface balding where two-dimensional blade profiles are to be found that insures the desired working range stability and efficiency. During the design, the constraints arising from aerodynamics, aeromechanical, mechanical, heat transfer, and manufacturing considerations have to be satisfied.

The design of turbine and compressor blade had made a great progress [31–36]. Many advanced design method and CFD tools had been incorporated into design procedure. However, most of the design procedures only focus on the flow prediction and there are few papers that describe the design’s overall processes and design implements. For example, Wellborn and Delaney [34] described a compressor design system used by Rolls-Royce plc, which comprise three tools, through-flow, 3D isolated blade row calculation and 3D multistage predication. The turbo machinery design is an integrated process, which contains a process
Design and Optimization of Turbo Compressors

from mean line, through-flow, airfoil design and analysis. This study developed a design process, which can be easily adapted by industry.

The aerodynamic design procedures for turbo machinery airfoil used in this study are shown in Fig. 3. The design system consists of the mean line analysis, through flow analysis, airfoil section design, airfoil stackup, 3D blade row and multistage flow analysis. For obtaining the highest design efficiency, the optimizer was used to do the section optimization. The three-dimensional CFD code was used for blade stack up optimization. The optimizer may be used for three-dimensional balding although authors do not encourage to use optimizer for 3D optimization. The discussion about issues regarding to 3D optimization will be given in section of optimization.

Mean line analysis determines the loading of the stage and annular area. It plays an important role in the turbo machinery design. The mean line design for the first-stage compressor and last stage turbine is critical. The enthalpy rise for compressor and drop for turbine are fixed by the mean line analysis. The overall machine character is determined by meanline analysis. The compressor and turbine efficiency is strongly influenced by mean line design.

Through flow analysis is one of the preliminary design modules. The streamline curvature calculations can be used to optimize the overall parameters of a multi-stage turbo machine. This module establishes the definitions of the flow path and work distributions in radio direction. The velocity diagrams at design point for different blade row and different streamlines are determined. The optimization code can be used to do the optimization for selecting the best design parameters, for example, stage loading, and stage enthalpy change.

Figure 3: Blade design and optimization procedure.
The airfoil section design is a very important step for the aerodynamic design. All the airfoils are designed by the section design and reasonable stack up. In this design system, authors [2] developed a blade profile design system in which airfoil consist of four segments where tangent slopes as well as curvature are supposed to be continuous at the segment joints.

A three-dimensional blade method was developed in this study as shown in Fig. 3. It can be seen that CFD code for both section analysis and three-dimensional blade row analysis is very important. Development of the efficient and wide range application CFD codes is very important. In this study, an effective numerical method was developed for both two-dimensional and three-dimensional code. In this study, a brief introduction for the numerical method developed for two-dimensional incompressible and compressible flows was made. Details for the three-dimensional method and applications are given in reference [37].

2.3 Flow solver for section analysis

A time-marching algorithm [27, 36–41] was used in the present cascade flow computations. The computation starts with a rough perturbation, which develops under certain boundary conditions. In this approach, the governing equations are replaced by a time-difference approximation with which steady or time-dependent flows of interest can be solved at each time-level.

Assume that passage-averaging plane, where the through-flow solution is calculated, is located at the radius \( r \) with the constant angular velocity \( \Omega \). The quasi-three-dimensional time-dependent Navier–Stokes equations on the passage-averaging plan can be written as:

\[
Q_t + E_x + F_y = R_x + S_y + H
\]

where \( E_x, F_y, R_x, \) and \( S_y \) are the flux vectors.

In the above equation, the effective viscosity is the sum of the molecular viscosity and turbulent viscosity, i.e.

\[
\mu_{\text{eff}} = \mu + \mu_t
\]

Incorporating artificial dissipation terms into the time-derivatives [27, 41, 42], eqn (1) is modified as:

\[
\Delta t \left( \frac{\partial \Lambda_E}{\partial x} + \frac{\partial \Lambda_E}{\partial y} + \frac{\partial \Lambda_F}{\partial x} + \frac{\partial \Lambda_F}{\partial y} \right) + I \left( Q_t + E_x + F_y + R_x + S_y \right)
\]

where \( \Lambda_E \) and \( \Lambda_F \) are matrix-valued dissipation terms with respect to positive or negative values of \( \lambda_1, \lambda_2, \) and \( \lambda_3 \). Matrix \( \Lambda_E \) can be written as:

\[
\Lambda_E = \lambda I + \left( \frac{\lambda_1 + \lambda_2}{2} - \lambda_3 \right) \left( \frac{v - 1}{a^2} E_1 + E_2 \right) + \frac{\lambda_1 - \lambda_2}{2a} \times \left[ E_3 + (\gamma - 1)E_4 \right]
\]
where

\[
E_1 = [1, u, v, C_p T]^{T} \left[ (u^2 + v^2) / 2, -u, -v, 1 \right]
\]

\[
E_2 = [0, 1, 0, u]^{T} \left[ -u, 1, 0, 0 \right]
\]

\[
E_3 = [1, u, v, C_p T]^{T} \left[ -u, 1, 0, 0 \right]
\]

\[
E_4 = [0, 1, 0, u]^{T} \left[ (u^2 + v^2) / 2, -u, -v, 1 \right]
\]

(5)

and where

\[
\begin{align*}
\lambda_1 &= u + a \\
\lambda_2 &= u - a \\
\lambda_3 &= u
\end{align*}
\] (6)

However, in the calculations, we cannot choose the values of \( \lambda_1, \lambda_2, \lambda_3 \) according to eqn (6) because near the stagnation points \( \lambda_3 \) approaches zero, whereas the flow in sonic condition, \( \lambda_1 \) and \( \lambda_2 \) both approach zero. To solve this problem, we limit these values in the following manner:

\[
\begin{align*}
\lambda_1 &= \max(u + a, u/M_r^2) \\
\lambda_2 &= \max(u - a, u/M_r^2) \\
\lambda_3 &= \max(u, u/M_r^2)
\end{align*}
\] (7)

and

\[
M_t = \begin{cases} 
0.001 & M \leq 0.001 \\
M & 0.001 < M \leq 1 \\
1 & M > 1
\end{cases}
\] (8)

where \( M \) is the calculated Mach number.

By integrating eqn (1) over space to form the hybrid scheme, we have

\[
\int \int \int \Delta \left[ \frac{\partial \lambda^E_x}{\partial x} + \frac{\partial \lambda^E_x}{\partial y} - \frac{\partial \lambda^E_y}{\partial x} - \frac{\partial \lambda^E_y}{\partial y} \right] + I \int \int \int \frac{Q^{\gamma+1} - Q^{\gamma}}{\Delta} \ dV
\]

\[
+ \int \int \int \left( \frac{\partial E^+}{\partial x} + \frac{\partial E^-}{\partial y} + \frac{\partial F^+}{\partial y} + \frac{\partial F^-}{\partial x} \right)^{\gamma+1} \ dV = \int \int \int \left( \frac{\partial R}{\partial x} + \frac{\partial S}{\partial y} \right)^{\gamma} \ dV + \int \int H dV
\] (9)

The evaluation of viscous terms \( R_x \) and \( S_y \) of eqn (1) requires first derivatives of the velocities and the energy value at each cell face. These are achieved by evaluating the gradient of every required flow quantity at the cell center from the known primitive variables at each time step. The terms can be written as

\[
\int \int \int \left( \frac{\partial R}{\partial x} + \frac{\partial S}{\partial y} \right) \ dV = \int (R_x n_x + S_y n_y) \ dS
\] (10)
In the present study, the second order approximation form was used as:
\[ \delta Q_{i,j}^{n+1} = Q_{i,j}^{n+1} - Q_{i,j}^n \]  
(11)

Defining \( A = \frac{\partial E}{\partial Q} \) and \( B = \frac{\partial F}{\partial Q} \), we have
\[ (E_{n+1}^{n+1}) = (E_{n}^{n}) + (A_{n}^{n}) (Q_{n+1}^{n+1} - Q_{n}^{n}) \]  
(12)
\[ (F_{n+1}^{n+1}) = (F_{n}^{n}) + (B_{n}^{n}) (Q_{n+1}^{n+1} - Q_{n}^{n}) \]  
(13)
and the implicit Jacobians, \( A^\pm \), can be written as:
\[ A^+ = (A + \rho_A I)/2 \]  
(14)
\[ A^- = (A - \rho_A I)/2 \]  
(15)
where \( \rho_A \) is the spectral radius of the Jacobian matrix \( A \).

Finally, we can obtain,
\[ C_{i-1,j} \delta Q_{i-1,j}^{n+1} + C_{i,j-1} \delta Q_{i,j-1}^{n+1} + C_{i,j} \delta Q_{i,j}^{n+1} + C_{i,j+1} \delta Q_{i,j+1}^{n+1} + C_{i+1,j} \delta Q_{i+1,j}^{n+1} = \Delta Q_{i,j}^{n+1} \]  
(16)
where
\[ C_{i-1,j} = \frac{\Delta t}{2} (A_{E}^{-})_{i-1,j} S_{y,i-1/2,j} - \Delta t (A_{E}^{-})_{i,j} S_{y,i-1/2,j} \]
\[ C_{i,j-1} = \frac{\Delta t}{2} (A_{F}^{+})_{i,j-1} S_{x,i,j-1/2} - \Delta t (B_{F}^{+})_{i-1,j} S_{x,i,j-1/2} \]
\[ C_{i,j} = \Delta V_{i,j} - \frac{\Delta t}{2} (A_{E}^{+})_{i,j} S_{y,i-1/2,j} \]
\[ - \frac{\Delta t}{2} (A_{E}^{-})_{i,j} S_{y,i+1/2,j} \]
\[ - \frac{\Delta t}{2} (A_{F}^{+})_{i,j} S_{x,i,j-1/2} - \Delta t (A_{F}^{+})_{i+1,j} S_{x,i,j+1/2} \]
\[ + \Delta t (A_{y}^{+})_{i-1/2,j} S_{y,i-1/2,j} - \Delta t (A_{y}^{+})_{i+1/2,j} S_{y,i+1/2,j} \]
\[ + \Delta t (B_{x}^{+})_{i,j} S_{x,i,j-1/2} - \Delta t (B_{x}^{+})_{i,j} S_{x,i,j+1/2} \]
\[ C_{i,j+1} = \frac{\Delta t}{2} (A_{E}^{-})_{i,j+1} S_{y,i+1/2,j} + \Delta t (A_{E}^{+})_{i,j+1} S_{y,i+1/2,j} \]
\[ C_{i+1,j} = \frac{\Delta t}{2} (A_{F}^{+})_{i+1,j} S_{x,i,j+1/2} + \Delta t (B_{F}^{+})_{i+1,j} S_{x,i,j+1/2} \]
Since eqn (16) is an implicit scheme, it can improve the numerical stability even with a large time step. In this numerical scheme, $C_{i-1,j}, C_{i+1,j}, C_{i,j-1},$ and $C_{i,j+1}$ are all scalars, which are calculated using integration combined with the flux vectors. In this way, the computational efforts are greatly reduced as compared with other coefficient matrix implicit schemes for all time-steps [27, 41]. The implicit equation (eqn (16)) can be solved by using two sweeps along $i$ and $j$ directions as follows:

$$C_{i,j-1} \delta Q_{i,j-1}^{n+1} + C_{i,j} \delta Q_{i,j}^{n+1} + C_{i,j+1} \delta Q_{i,j+1}^{n+1} = \Delta Q_{i,j}^{n+1} - C_{i-1,j} \delta Q_{i-1,j}^{n+1} - C_{i+1,j} \delta Q_{i+1,j}^{n+1}$$

(17)

Since the present scheme consists of an explicit part to calculate $\Delta Q_{ij}$ and an implicit part to calculate $\delta Q_{ij}$, which possesses the advantage of both schemes.

In this study, the time-step is selected according to the CFL number constraints as:

$$\Delta t = \min(\Delta t_x, \Delta t_y)$$

(18)

where

$$\Delta t_x \leq \frac{CFL \cdot \Delta x}{|u| + a + (2\omega/\Delta x \rho)}$$

$$\Delta t_y \leq \frac{CFL \cdot \Delta y}{|v| + a + (2\omega/\Delta y \rho)}$$

where $\omega = \max(\mu/Re, \mu_t + 2\mu/Re, \mu\gamma P/Re)$.

The choice and development of a turbulence model remain important points to reproduce the flow features. However, there is still no such kind of turbulence model, which can well represent a true flow situation. The Baldwin–Lomax [27, 41] model has several good features, such as its usefulness in separated flows with a small separation region, its relatively smooth and continuous length-scale from transition of the boundary layer into the far wake region, and its ability to accurately predict the wall effect near the trailing edge. For these reasons, the above-mentioned Baldwin–Lomax turbulence model was employed in this study to handle the turbulent flow computing cascade flows around the turbine. The boundary conditions play an important role in determining accurate solutions and rapid numerical convergence. Several types of boundary conditions have been used which are similar to the previous studies [27].

The discussion has been extended for many years for the discussion of best type mesh, which should be used, for turbine or compressor blade flow calculation [43, 44]. The more orthogonal the grid, the smaller will be the numerical errors due to the discretization of governing equations. However, no one type of grid is ideal for blade-to-blade flow calculation. In this study, the H-type mesh was used. Another problem for the blade-to-blade mesh is the trailing edge mesh and trailing edge Kutta condition [45]. The author has noticed that the number of the mesh point near the trailing edge points strongly influences the loss calculation. Here, a realistic method is proposed, i.e. when the mesh on the trailing edge
circle is more than 10 points, an explicit viscous Kutta condition [45] is applied on the trailing edge circle which flow can leave the blade smoothly.

The mesh refinement study was conducted prior to the calculations; as reported by Xu and Amano [27, 37, 41], the mesh size of $110 \times 45$ with 80 nodes on blade is sufficient. The H-mesh was used in all the computations. The computational mesh is shown in Fig. 4. A typical convergence history of the calculation is shown in Fig. 5. It is demonstrated that the present code has a good convergence. In this study, this code was used in the two dimensional airfoil section analysis and optimization.

2.4 Optimization

Turbine and compressor design faced the challenge of designing high efficiency and reliable blade design. Design techniques are typically based on the engineering experience and may involve trials and errors before an accepted design is found. The ability of the three-dimensional codes makes the detailed flow predictions possible; however, it is very time consuming to perform a three-dimensional flow calculation of the design in the design process. The three-dimensional approach is a way to help the designers do some parameters study and compare the numerical study with testing to improve the design. The three-dimensional code developed in this study [37] is used to do parameters studies, like lean, bow, and sweep effects. The two-dimensional analysis developed in the study is used to optimize blade section airfoil.

The blade design often started two-dimensional airfoil section. Some of the design parameters are obtained by through flow optimization and vibration, heat transfer
Analysis. For example, the solidity, the number of the blades, and most of the parameters were optimized through the sectional design. For example, leading- and trailing edge radii, stagger angle, and maximal thickness position. The final three-dimensional analyses are used to make lean, bowed and sweep modification of the blades.

2.5 Method of numerical optimization

The optimization process used in this study was based on the constrained optimization method [46, 47]. If the objective function to be minimized is \( F(\mathbf{X}) \), and constraint functions is \( g_j(\mathbf{X}) \), the optimization problem can be formulated by the objective and constraint functions as

\[
\nabla F(\mathbf{X}) + \sum \lambda_j \nabla g_j(\mathbf{X}) = 0 \tag{19}
\]

where \( \lambda_j \) are the Lagrange multipliers and \( \mathbf{X} \) is the vector of the objective. The finite differencing method can be used to obtain the gradients of objective and constraint functions with respect to design variables and constraints. Many commercial optimization packages are available as an optimizer to design problems [47].

The objective function of this study is adiabatic efficiency. The optimization objective is to obtain the highest efficiency under giving constraints. For more convenient, the total pressure loss was used as an objective variable to judge the design. The definition of the total pressure loss in this study is

\[
\xi = \frac{(p_{in}^* - p_{out}^*)}{\Delta H} \tag{20}
\]

where \( \Delta H \) is the outlet dynamic head of the exit plane.
2.6 Two-dimensional section optimization

In this study, a compressor stator blade original designed by using two NACA65 sections was redesigned and optimized. In this design, the thickness of the airfoil, chord, and stagger angle and inlet and outlet flow parameter were fixed. The designed blade was used to replace the old blade. The section design optimization mainly achieved through adjusts the section maximum thickness location. It was found that the maximum thickness locations influence the losses and performance.

In this study, five sections were selected as design section. All section design used the similar method and procedure. As an example, the result presented here is the design information for 50% span section.

Figures 6–8 show the Mach number distributions of the design blades with different maximum thickness locations. Figure 9 shows the relationship between total pressure losses and location of the maximum thickness. It was shown that the Mach number distributions changed with the maximum thickness. The change of the Mach number distributions will influence the boundary layer development and influence the losses of the section. It was found that there is an optimum location.
of the maximum thickness. Study also showed that this location is changed with
the flow conditions and stagger of the airfoil. After optimization study, the airfoil
section with highest efficiency was selected. The airfoil was stacked up using a
smooth leading edge curve method. Before the airfoil was selected as a final blade,
the three dimensional analysis was performed to investigate the benefits of the
three-dimensional blade features. Three-dimensional analysis showed that after
section optimization the airfoil performance was improved as shown in Fig. 10.
However, the end wall region did not have a significant improvement. Some three-
dimensional features were used to reduce the losses and increase the efficiency.

2.7 Three-dimensional CFD analysis and blading

In the present design system, three-dimensional CFD analysis codes were devel-
oped [37] to guide the three-dimensional final modification of the two-dimensional
section stack up. Some parameters were selected to do the study based on design
experience. In this redesign, the three-dimensional feature-bowed blade was used

Figure 7: Mach numbers distribution at maximum thickness at 0.25c.
Figure 8: Mach numbers distribution at maximum thickness at 0.45c.

Figure 9: Total pressure losses and maximum thickness location.
to reduce the secondary losses. In the three-dimensional blading, the optimizer will not be used in this study because performing the three-dimensional analyses is very time consuming. In the current study, three-dimensional optimization was obtained through parameter study.

In this study, only bow feature was applied. The bow location and degree of the bow were selected as the parameters to study. It was found that the 15° bow located at 30% of the span can eliminate separation and get better performance. Three-dimensional analysis showed that after two-dimensional section optimized design most of the span the flow remained attached and well behaved although it had a small region of end wall separation on the section side as shown in Figs. 11–14. After introducing the bows on the blade, the flow around the compressor blade on both tip and root end wall were attached. The total pressure losses were reduced as shown in Fig. 10. This study showed that total section efficiency was increased about 3% compared to the original blade. Both two-dimensional section design and three-dimension optimization increase the efficiency at similar percentage. It is shown that both two-dimensional optimization and three-dimensional study are important. However, three-dimension blading more depends on the experience of the designer.

It is interesting to point out, based on the design experience, that the three-dimen-sional optimization based on high efficiency normally does not produce final blade shape. The selection of the final configuration was performance through small adjustments based on experience to obtain the blade design with both high performance and good stability. After three-dimensional analysis, some small adjustments were made based on the possible manufacturing uncertainties and applications. For example, the compressor stator design in this study was recambered to increase exit angle about 1.5° to increase the compressor surge margin.

Figure 10: Total pressure losses with span.
Figure 11: Static pressure distribution on the suction surface.

Figure 12: Axial velocity contour near blade suction surface.

Figure 13: Velocity vector near root section near the trailing edge suction surface.
2.8 Discussion

The flow field around turbine and compressor blades exhibits a very complex flow features. The flow field involves various types of loss phenomena and hence high-level flow physics is required to produce reliable flow predictions. The blade design process is a very time consuming process and optimization process is very complicated. The method developed in this study for the airfoils section and blades design and optimization is one the possible processes and very easy for adapting in the aerodynamic design. In this study, a two-dimensional code was used for section design instead of three-dimensional code, which provides an economical way to design, and to optimize. The three-dimensional blades were optimized after stacked up the two-dimensional sections. The lean, bowed and swept features of the three-dimensional blade were created during the three-dimensional optimization. It was shown that both two- and three-dimensional design and analysis were backbones of the blade aerodynamic designs. Results show that the both airfoil shape optimization and three-dimensional optimization can be used to improve the performance of compressor and turbine blade.

3 Part III: Centrifugal compressor design experience

3.1 Compressor design

The new compressor design is focused on the customers’ needs with shortest time to market, low cost and better performance. For pushing the design to the-state-of-art aerodynamic performance, the structural design needs to meet the suitable life of the compressors. The mechanical integrity is one of the important parts of the centrifugal compressor design. The mechanical constraints always are negative factors for aerodynamic design. The purposes of the mechanical analyses are to provide all compressor components at a reasonable time duration to sustain
the aerodynamic and centrifugal force and eigen frequencies do not match critical excitation frequencies [48]. The safety factors of the mechanical design had been reduced dramatically compared with “old fashioned” design. Due to the maturity of the finite element analysis (FEA) tools and material property improvements, the safety factor of a modern industrial compressor design is normally set to 7–12%. The mechanical requirements require structure designers to have a better practice to allow more freedom to aerodynamic designers and to keep all the components at the least weight and at the lowest cost. Long lifetime of a single component of compressors is not a design goal. However, the structure design for defined period of lifetime is trivial. Emphasis on the efficiency will be the foundation of the compressor design, but not a central consideration as in the past. The development cost and development time are also the key factors that need to be considered for the modern compressor design. Industrial compressor design is required to design a state-of-the-art performance compressor without making second build for less cost and development time. This requires compressor design engineers to have extensive knowledge of centrifugal compressor design. The detail design considerations can reduce the time to perform the advance design studies and laboratory investigations. The wide variety of design subjects represents a very complex design world for all designers. The purpose of this paper is to provide information for new engineers and help them to understand the overall design before they start to design a centrifugal compressor.

The compressor market and business model have changed compared with a few decades ago. Industrial compressor design requires design for success in the marketplace, not just for scientific experiments. In the past, the compressor designers developed a new compressor in the development group, and passed the design to manufacturing. The manufacturing group would evaluate how to make it at the lowest cost. Some developments were rejected because they could not meet the market requirements. The new development model requires compressor designer’s design for market, manufacturing and end users. Recently, some new business concept was proposed [49, 50]. New developments will consider an integrated system of manufactures and end users. The new compressor developments become a complex system task. Minimizing manufacturing cost of the compressor design is not enough. The compressor design must consider all aspects of the manufacturing and end users. If surplus is defined as total profit of manufacturing, end users and aftermarket, the new compressor development will focus on the design for maximum surplus. Therefore, in the compressor design stage, many choices of the design options need to be considered before final design and making decisions based on the surplus value. It is essential that design engineers begin to perform compressor design after fully understanding all aspects of the design considerations [51, 52].

3.2 Impeller designs

Impeller is one of the key components to influence centrifugal compressor overall performance [53–56]. Nowadays, the efficiency of centrifugal compressors has
been increased dramatically especially low-pressure ratio centrifugal compressors. The big challenge for centrifugal compressor design is to keep efficiency level high at state-of-the-art and increase the compressor operating range [54, 55]. Increasing the compressor operating range without sacrificing compressor peak efficiency is difficult to achieve. Aerodynamic engineers not only need to understand the surge physics but also need to apply design experience to design. Surge is influenced by all components of the compressor. The physics of surge and stall are still not fully understood. We still cannot find any tool that can capture all features of surge and stall as shown in Fig. 15. Many theoretical studies [57] were focused on the better understanding of surge and stall, but none can be used as a design tool yet. More theoretical work and experimental studies need to be done in order to incorporate stall in the design system. Designs for wide operating range were mainly dependent on the engineers’ experience and their understanding of the stall and surge. Another important objective for impeller design is to reduce the manufacturing cost. Manufacturing cost could be reduced when designs for manufacturability are effectively considered. The designed impellers should meet requirements to be easily withdrawn from a casting mold without destruction and disassembly of the mode. This requires the lean angle of blade change linearly with the impeller radius and axial direction as shown in Fig. 16. All these considerations for design will help the final design to meet the compressor design target with less design iterations.

Different designers may have different design methodologies for impeller designs. What kind of area distributions, curvatures, velocity or pressure profile will lead to a good design is strongly dependent on the designers’ practice and experience. Two totally different design philosophies could produce similar performance. For example, two impellers designed by Garret and Pratt Whitney [49] as shown in Fig. 17 had different shapes with similar performance at design point. The impeller designed by author also presented different features, which also provided good performance. It is shown that if design follows the basic design guideline, a wide range of solutions to the design can be used. The experience data presented in this paper will help designers to set up the design guideline for compressor initial design phase.

Figure 15: (a) Stall and (b) surge pressure variation with time.
Figure 16: Impeller for manufacturing.

Figure 17: Different impeller designs.
Authors’ design shown in Fig. 17 is an example of recent developed single stage centrifugal compressors. At design point, the total to static stage pressure ratio is about 3.7 and the flow coefficient is about 0.12. The running clearance at impeller tip was 4.5% of the impeller exit blade width. Six builds were assembled and tested based on ASME PTC-10 test procedure [58]. The compressor performance obtained from average of six build tests is shown in Fig. 18. The differences of test results for different builds for adiabatic efficiency and head coefficient were within ±0.5% and ±0.75%, respectively. The test uncertainties for total pressure (in psi), static pressure (in psi), and temperature (in Fahrenheit) were ±0.25%, ±0.2% and ±0.5%, respectively, based on uncertainty analysis [59]. Test results showed that the compressor performance was encouraging at both design and off design point. The design met the low cost target and allows large manufacturing tolerances. The insensitivity of the impeller surface finish and large tip clearance make it easy for assembly.

3.3 Impeller geometry

The initial design of a centrifugal compressor is always started with requirements from customers or marketing analysis. Designers select basic configurations and provide basic performance to customers or marketing by using their experience data. Aerodynamic designers also need to provide estimation for the compressor basic geometry to engineers in other disciplines. For example, rotor dynamic engineers and bearing designers rely on the impeller geometry information to perform their work. Although the basic geometry design is not intended to yield optimization of the impeller, it can accelerate overall design process and reduce the development cost.

Figure 18: Single stage compressor performance.
Before aerodynamic designers determine basic impeller geometry, the rotational speed of the impeller needs to be selected. If there are no special requirements for rotational speed, we normally optimize rotational speed based on the Balje’s charts [60] by using optimal specific speed. Although Balje’s charts are not very accurate tools, they are sufficient enough to provide the initial estimate for impeller geometries.

During the initial design, the important information needed for bearing designers and rotor dynamic engineers is impeller weight. Aerodynamic designers can estimate impeller sizes based on the required gas flow, pressure ratio and impeller rotational speed. Our design practices showed that weight of impeller is the function of impeller diameter. Fig. 19 summarizes the relationship between impeller diameter and weight for sixteen ASTM A564 stainless steel unshrouded impellers. The impeller weight mainly determined by impeller disk and blades only contribute a very small portion of the weight. Therefore, we plotted impeller weight and diameter relation in one figure for all designed unshrouded impellers with different blade counts and with or without splitters.

In the initial stage of compressor design, selections of impeller inlet and outlet velocity vectors and choice of blade numbers are key initial design decisions. Velocity vectors may be obtained through a mean line program. The experience data showed that both inlet blade numbers and exit blade numbers were a function of stage pressure ratio. Relationships between numbers of blades and stage pressure ratio are shown in Figs. 20 and 21 for without and with splitter impellers, respectively. In general, high stage pressure causes blade-loading increase and impeller needs more blades to distribute loading. Variations in number of blades at similar pressure ratio were due to the size of the impellers. For small sizes impeller, manufacturing capabilities may limit the number of blades. Impeller sizes plotted in Figs. 20 and 21 were in the range 2–45 inches. The machine performance requirements and manufacturing feasibilities are factors to determine whether splitters are used or not.

Figure 19: Variation of impeller weight with diameter.
The inlet blade height is determined by design inlet flow rate and impeller hub radius. Inlet hub radius is determined by the attachment of impeller. For overhung impellers, inlet hub radius is normally selected in the range between 10% and 20% impeller tip radius. For the shaft and bolt through impeller, selections of inlet hub radius are based on stress requirements.

Figure 20: Number of blades at inlet versus stage pressure ratio.

Figure 21: Number of blades at exit versus stage pressure ratio for split impellers.
Blade thickness at inlet and discharge were determined mainly by tensile and bending root stresses at leading edge and blade exit. FEA calculations and stress tests showed that blade root stresses mainly caused by the centrifugal force. The blade high was a key factor to impact the blade root stresses. The mean line thickness at inlet and exit were determined by the blade heights at inlet and exit as shown in Figs. 22 and 23. Experience showed that the blade thickness changed linearly with blade height.

The three-dimensional features of the impeller blade are dependent on engineers’ experience and stress limitations. The modern impeller is normally a three-dimensional design. The wrap angle, lean angle and back sweep angle are using larger value than those in the past. The large wrap angle can reduce the camber of the blade but increase the frictions of the fluid. Large lean angle permits blade design at all blade sections with desired shape. Leaning the blades creates back sweep and retains purely radial fibers, which is beneficial for bending moments. Experience showed that impellers with back sweep generally have high efficiency.

3.4 Impeller aerodynamic design

One of important guidelines for impeller aerodynamic design is to set a reasonable diffusion ratio of some sort. The diffusion of the impeller can be represented by velocity ratio, diffusion factor and relative Mach number ratio. The ratio of relative Mach number was used in this discussion. Mach number ratio can avoid the
one-dimensional assumption at inlet. MR$_2$ is defined as the ratio of relative Mach number at impeller inlet to the average Mach number at impeller exit. Figure 24 also showed the up and low boundaries for maximum deceleration likely to be achieved for two-dimensional and three-dimensional impellers [54]. It is shown that the experience data fall inside theoretical boundaries. Experience showed that the Mach number ratio MR$_2$ fell between 1.15 and 1.4 gives a good overall performance. The upper boundary of MR$_2$ at 1.4 for industrial compressor and 1.7 for jet engine impellers are reasonable expectations. The experience indicated that large diffusion might cause a huge loss. The ratio of the Mach number can be selected within a large range. An important factor to impact the selection of diffusion level is inlet Mach number. Figure 25 is a relationship between relative velocity ratio, incidence and inlet relative Mach number for a typical industrial impeller. It can be seen that diffusion is not an absolute parameter, which influences the stall of the compressor. It is worth to point out that this test impeller was stall at inducer first. Inducer shroud velocity represented the rotational speed.

Traditionally, impeller inlet incidence is set to zero at design condition [61]. Modern impeller designs need not only to consider maximum efficiency at design point, but also to consider manufacturing cost and off-design performance for whole operating range [55]. Inlet blade angles are not necessarily the same as inlet relative flow angles. Experience data in Fig. 25 showed that changes of inlet flow incidence impact both efficiency and operating range of the impeller. Fig. 26 shows that little negative incidences could raise impeller operating-range. However, when negative incidence increased to a certain level, the operating range did not enlarge and efficiency dropped significantly. The impeller design should avoid this situation.

Estimations of the impeller exit width are critical for both primary performance estimation and basic dimension set up. The major impacts of impeller blade exit

Figure 23: Relationship of impeller exit RMS thickness and tip width.
width are flow capacity and pressure ratio of the stage. It is difficult to calculate the impeller exit width accurately in a simple way. Rodgers diffusion factor equation [62] provided a good estimated value for impeller exit width \( B_2 \). If mean meridional blade length can be estimated as

\[
L = \frac{2\pi(n_2 - n_1)}{4}
\]  

(21)
then impeller exit width can be estimated as

\[
B_2 = \frac{10(D_2 - D_{11})}{1 + (W_2/W_1)} \left[ DF - 1 + \frac{W_2}{W_1} - \frac{\pi D_2 C_{T2}}{2 L_c W_1} \right] \frac{(D_1 - D_{11})}{2}
\] (22)

Secondary flows inside of the impeller caused by imbalance of static pressure and kinetic energy. One of the typical secondary flows, horseshoe vortex, has been well documented. It is shown that the strength of the secondary flows is governed by the vortex starting conditions. The further development of the vortex is determined by the conservation of angular momentum. Impeller meridional blade profiles influence secondary flow loss level. Laminar viscous dissipation function can estimate the secondary flow loss due to the blade profiles [62].

\[
\Delta H = N \mu \left[ 2 \left( \frac{\partial u}{\partial x} \right)^2 + 2 \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 \right] dV
\] (23)

Tip clearance cannot be avoided for unshrouded impellers. Bearing clearances and manufacturing tolerances of impeller and intake ring controlled the minimum impeller tip clearance. The minimum tip clearance was normally defined at maximum rotational speed with hot weather conditions for most motor driven compressors. For compressors installed in the same shaft with gas turbine, minimum compressor tip clearance was estimated when compressor operating at maximum rotational speed with hot weather and machine overall net axial thrust load towards compressor. Tip clearance increases quadratically with impeller rotational speed if
other operating conditions do not change. Tip clearance impacts overall compressor performance because it increases the magnitude of secondary flow inside impeller blades and produces strong tip vortices. Tip clearance flow transports low momentum fluid from suction side to pressure side of blades. The circumferential center of the secondary flow is dependent on the size of tip clearance. Secondary vortices are located near shroud side for small clearance, whereas secondary vortices may spread to the center and even hub of the flow channel for large tip clearance impeller. The clearance distribution affects the wake formation and location at impeller exit. Large clearance at leading edge results in low energy center close to suction side of the blade. Reducing the clearance at leading edge, the wake moves towards pressure side of the blade. The tip clearance setting depends on compressor maximum surplus value. There are several methods to reduce the tip clearance losses [2, 56]. Figure 27 shows that variable clearance could significantly improve the stage efficiency of compressors. The tip clearance changes the compressor stage head and capacity. Test data [55] indicated that head coefficient almost changed linearly with tip clearance as shown in Fig. 28 and flow coefficient followed a secondary order curve with clearance as shown in Fig. 29.

3.5 Reynolds number and surface finish

Reynolds number or Rossby number has big impacts on impeller maximum surplus value. Reynolds number is an indicator of inertial force versus viscous force for a moving fluid. Rossby number is a measurement of inertial force versus Coriolis force. Fundamental fluid dynamics theory [63, 64] showed that the flow inside a pipe for different Reynolds number represent different flow patterns. This is also true for flow inside of impeller blades. The experience showed that if a flat velocity

![Figure 27: Compressor efficiency change versus clearance.](image-url)
profile at inlet between two impeller blades or diffuser vanes, the flow development along the flow channel presents different profiles with different Reynolds numbers. For low Reynolds number flows, the exit velocity profiles are almost parabolic and only with small portion of flat profile. For high Reynolds number flows, the exit velocity profiles have large flat profiles.

The peak meridional velocities for high Reynolds number flows are normally located at the hub pressure sides of the blade due to potential flow effects. Low Reynolds number flow regions are located near suction side of the blades.
The viscous jet and wake interaction causes flow separations. Reynolds number also strongly influences secondary flow patterns. By increasing Reynolds number, the strength of clockwise secondary passage flow circulation increases. Reducing pressure and suction velocity gradient increases flow circulation of count clockwise secondary flow. The optimum design should try to offset each other to minimize the secondary flow losses.

The machining and casting of centrifugal compressor impellers and other components result in an inherent surface roughness. The sizes and forms of roughness depend on manufacturing process. The levels of the surface finish represent the manufacturing cost. It is very important to balance manufacturing cost and performance. Detailed discussions on surface finish and Reynolds number can be found in reference [65, 66]. Loss due to surface finish may be able to represent as wall friction. Wall friction is the function of Reynolds number and can be written as [65]:

\[
\frac{1}{\sqrt{f}} = 1.74 + 2 \log_{10} \left( \frac{k}{B_2} + \frac{18.7}{Re \sqrt{f}} \right) \tag{24}
\]

This equation can be solved by using simple a computer program or spreadsheet.

### 3.6 Diffuser and volute

The fluid leaves impeller with a high velocity and inclines at a large angle to radial direction. It is necessary to decelerate this flow to increase static pressure. Diffusers are used for this purpose. Behind the diffuser a volute or collector is used to decelerate flow further and collect flow.

Designs of diffuser and volute are also critical for compressor. Extensive studies [54, 55, 65] for diffuser and volute designs have been conducted. Many useful suggestions for their design were proposed. Some parameters, for example, solidity of the vaned diffuser, vane deflections, diffuser and volute inlet and exit radii, and volute size were discussed in details in literatures [54, 55, 65]. Good summaries for diffuser designs can be found in references [54, 67].

Figure 30 shows the value of cross-sectional area of the volute channel as a function of the angular location for a modern design [55]. It is noticed that the relationship between cross-sectional area and location angle is non-linear. Experience indicated that this non-linear cross-section volute design benefited both compressor efficiency and operating range.

Recently, compressor designers started to pay attention to volute tongue studies [55]. The volute tongue is located at the intersection of scroll and volute discharge pipe. The volute tongue makes the flow circumferential discontinuity. One of the volute design objectives is to keep small incidences at the tongue area at design condition. However, due to widely operating range of the compressor operation and non-uniformity flow of the upstream, it is difficult to avoid tongue incidence. The non-uniformity of the static pressure produced by a volute tongue creates unbalanced aerodynamic force. This force may cause unacceptable levels of sub
synchronous radial vibrations of rotor and excessive bearing loads. It is important to reduce or eliminate the static pressure distortions at the exit of the impeller.

Studies were conducted to discuss impeller–diffuser interactions [55]. It was found that rounded tongue could enhance off-design performance and eliminate unbalanced aerodynamic force. Figures 31 and 32 showed computed static pressure distributions at vaned diffuser exit and impeller exit at $\phi_0 = 0.7$ and 1.25, respectively. Results showed that only small unsymmetrical static pressure could be seen at vaned diffuser exit for both flow conditions. However, pressure distortions were eliminated for both low flow and high flow cases at impeller exit. Compressor tests [55] confirmed computational results. The pressure variations at impeller exit caused by blockages of diffuser vanes were observed. These pressure variations also could be seen at the down stream of diffuser vanes. Rounded tongue changed head coefficients of compressor stages. Different tongue radius has different impacts for compressor head coefficients as shown in Fig. 33.

3.7 Discussion

Developments of CFD methods and modern measurement technologies have drawn scientific societies to pay attention to CFD applications, CFD calibrations and optimization design by using CFD [2, 53, 54, 56, 68]. Although one-dimensional mean line and two-dimensional streamline calculations have achieved a certain point of maturity, test data and design experience still need to cooperate into design practices. Industrial compressor designs still rely on and will continue to rely on the one- and two-dimensional design procedures. Design experience data are one of important parts for performing advanced compressor development. It is important that the academic field and industrial side continue to study and share experience to improve centrifugal compressor design process.
CFD and modern measurement technologies are very important in compressor designs. This paper presented some design experience to accelerate modern compressor design process. The detailed design data presented here are useful for designers to select design parameters. The experience data presented here also provide opportunities for improving design and manufacturing process to develop new compressors better, faster and cheaper.

Current design data are based on metals, steel or titanium components. The advanced materials will definitely play an important role in the future compressor designs. The static pressure coefficients at diffuser exit on the hub wall (Figure 31) and impeller exit at the hub wall (Figure 32) are shown below.

Figure 31: Static pressure coefficients at diffuser exit on the hub wall.

Figure 32: Static pressure coefficients at impeller exit at the hub wall.
development. The possibilities for carbon composites, ceramic matrix composites and advanced matrix material will provide higher strength at low cost. Some design considerations will change with the progress of material science.

4 Summary

The design of the advanced compressors has been practiced with increasing degrees of sophistication while continuously advancing levels of efficiency and operating regions. It is very important to understand the compressor design process and optimization and learn from basic design experience. This section covered the basic compressor design process, compressor blade optimization and some design experience. The compressor development and design have not reached a level of full maturity. The studies of new design methods and new technologies will continue to draw the attention of scientists and research engineers.

The design optimization process discussed here is one of the many methods. With the progress of the numerical method and computer hardware, the current optimization method may become more mature or may replace by more sophisticated methods. However, the method presented here can serve as reference for blade optimization. The present method was proved to have advantage than traditional method and easy to apply in the design process.

Due to the limitation of the computer CPU speed and memory limitation, full three-dimensional turbulent codes are still time consuming for using in the industrial compressor design and optimization. Major industrial compressor design systems are still reliable to two- and three-dimensional code with correlations from tests and experience. Basic experience and fundamental choice concerning the compressor design are very important for new compressor development. Basic parameter selections based on past design experience can accelerate the new compressor design. The design considerations also addressed the method for improving the manufacturability

Figure 33: Volute tongue radius versus performance.
and reducing the development cost to develop a better performance compressor.

The contents presented in this chapter touched only part of the wealth of issues that compressor development addressed. Authors hope that this chapter will provide the readers with a better grasp of the basic compressor design process and design experience involved during the new compressor development.

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Jacobian matrix</td>
</tr>
<tr>
<td>$B_2, b_2$</td>
<td>impeller tip width</td>
</tr>
<tr>
<td>$C_p$</td>
<td>pressure coefficient = $p/0.5\rho d_2 U_2^2$</td>
</tr>
<tr>
<td>$DF$</td>
<td>diffusion factor</td>
</tr>
<tr>
<td>$D_s$</td>
<td>specific diameter</td>
</tr>
<tr>
<td>$E_x$</td>
<td>flux vector</td>
</tr>
<tr>
<td>$F_y$</td>
<td>flux vector</td>
</tr>
<tr>
<td>$F_r$</td>
<td>function</td>
</tr>
<tr>
<td>$f_r$</td>
<td>friction factor</td>
</tr>
<tr>
<td>$H$</td>
<td>head</td>
</tr>
<tr>
<td>$K$</td>
<td>sand grain roughness</td>
</tr>
<tr>
<td>$L$</td>
<td>blade meridional length</td>
</tr>
<tr>
<td>$r$</td>
<td>RMS radius</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$ICFM$</td>
<td>compressor inlet flow in ft$^3$/min</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number</td>
</tr>
<tr>
<td>$MR$</td>
<td>Mach number ratio</td>
</tr>
<tr>
<td>$Q$</td>
<td>volume metric flow rate</td>
</tr>
<tr>
<td>$RG$</td>
<td>operating range = $(Q_e - Q_s)/Q_e$</td>
</tr>
<tr>
<td>$RMS$</td>
<td>root mean square</td>
</tr>
<tr>
<td>$R_{e_x}$</td>
<td>flux vector</td>
</tr>
<tr>
<td>$S_{y_x}$</td>
<td>flux vector</td>
</tr>
<tr>
<td>$\Delta t$</td>
<td>Time step</td>
</tr>
<tr>
<td>$u, v$</td>
<td>velocities</td>
</tr>
<tr>
<td>$V$</td>
<td>velocity vector</td>
</tr>
<tr>
<td>$W$</td>
<td>RMS relative velocity</td>
</tr>
</tbody>
</table>

Greek symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Lambda_F, \Lambda_f$</td>
<td>matrix-valued dissipation terms</td>
</tr>
<tr>
<td>$\Omega$</td>
<td>angular velocity</td>
</tr>
<tr>
<td>$\mu$</td>
<td>viscosity</td>
</tr>
<tr>
<td>$\rho_A$</td>
<td>spectral radius of the Jacobian matrix $A$</td>
</tr>
<tr>
<td>$\varphi$</td>
<td>flow coefficient</td>
</tr>
<tr>
<td>$\eta$</td>
<td>adiabatic efficiency (total to static)</td>
</tr>
<tr>
<td>$\psi$</td>
<td>head coefficient</td>
</tr>
</tbody>
</table>
Subscripts

1, 2         impeller inlet and exit
C            chock or clearance
I            loss
O            zero clearance
ref          reference
S            surge

References


Schoonmaker, P.M., Preliminary experience with an expert system providing initial centrifugal compressor sizing for performance prediction and analysis, 91-GT-28, 1991.


Wright, T., Comments on compressor efficiency scaling with Reynolds number and relative roughness, 89-GT-31, 1989.

