1 Introduction

1.1 Introduction

Fluid machinery is classified as those devices that transform fluid energy to shaft work or vice versa. The history of fluid machinery is long and the design technology of fluid machinery has developed with the development of fluid mechanics. Although the exact governing equations for single phase Newtonian viscous fluid, that is, the Navier–Stokes equations, were derived in middle of the nineteenth century, various approximate analysis methods, such as those with inviscid assumptions, were still used in the analysis of fluid flow before the Navier–Stokes equations were practically solved by numerical analysis using electronic computers more than a hundred years later. Thereafter, owing to the rapid development of computers, computational fluid dynamics (CFD), which solves the governing differential equations, becomes practical in the analysis of fluid flow.

Due to the complexity of the flow path in fluid machinery, application of three-dimensional (3D) CFD to the aerodynamic or hydrodynamic analysis of fluid machinery was somewhat delayed, but recently, CFD has been widely used in the analysis and design of fluid machinery. In the early stages, CFD was only used in the analysis of flow fields in fluid machinery due to the long computing time. But, continuous enhancement in computing power made the design optimization of fluid machinery using CFD practical. Thus, now CFD is utilized not only in the analysis of the flow in fluid machinery, but also in design through systematic optimization algorithms. However, instead of replacing the conventional design methods of fluid machinery, design optimization using CFD is being used as a supplementary design due to excessive computing times when it is used for the entire design of a fluid machine.

A typical design procedure recommended for the design of fluid machinery using CFD is as follows; a preliminary design using an approximate analysis method to determine a basic model of the fluid machine considered, a parametric study using 3D CFD to find the sensitivities of performance parameters on some selected geometric/operational parameters, and single- or multi-objective design optimization of the fluid machine using the design variables selected through parametric study. The design optimization requires repeated evaluations of the objective function(s), which is selected among the performance parameters of the fluid machine, and the number of objective function evaluations depends on the number of design variables and the optimization algorithm employed. An increase in the number of design variables in an optimization is generally expected to improve the results of the optimization, but the number of design variables for optimization is restricted mainly by the computational time. Therefore, design

Design Optimization of Fluid Machinery: Applying Computational Fluid Dynamics and Numerical Optimization,
First Edition. Kwang-Yong Kim, Abdus Samad, and Ernesto Benini.

optimization could become more popular in fluid machinery design if computing power is further enhanced.

1.2 Fluid Machinery: Classification and Characteristics

The fluid machines that transform fluid energy to shaft work are called turbines; more specifically, gas, steam, wind, and hydraulic turbines, depending on the working fluid. The other group of fluid machines that transform shaft work to fluid energy includes pumps, fans, blowers, and compressors. All the machines in this group using liquids are called pumps. But, if gases are used for the work, machines in this group are divided into fans, blowers, and compressors, depending on the magnitude of pressure rise.

Fluid machinery is also divided into two categories; turbomachinery and positive displacement fluid machinery. In turbomachinery, rotating blades (rotors) perform continuous energy transfer from or to the fluid flow passing through the blade passages. However, in positive displacement fluid machinery, there is a displacement of a certain amount of working fluid without relative motion between the fluid and moving part of the machine in rotating or reciprocating motion. In other words, the working fluid does not flow in certain parts of these machines. The following sections in this chapter are mostly concerned with turbomachinery.

Turbomachinery can be also categorized according to the change in the flow direction through the impeller as shown in Figure 1.1. If the flow direction does not change through the impeller, those machines are called axial flow turbomachines. Machines where the flow direction changes perpendicularly through the impeller are called radial flow (or centrifugal) turbomachines. If the change in flow direction is neither axial nor radial, the machines are called mixed flow turbomachines. Also, the rotors of

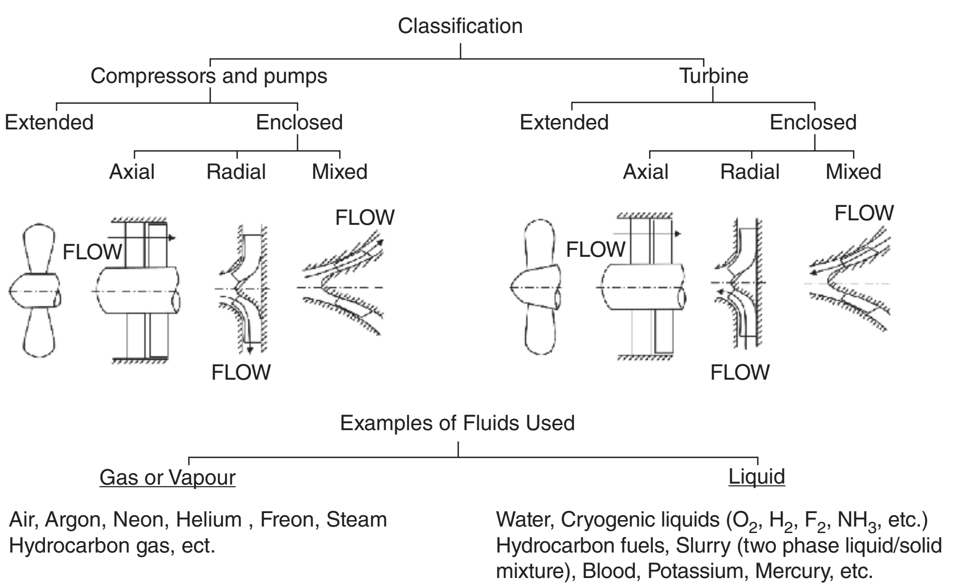


Figure 1.1 Classification of turbomachinery types. Source: Reprinted from Lakshminarayana 1996 (Figure 1.1 from original source), © 1996, with the permission of John Wiley & Sons, Inc.

turbomachinery may be enclosed in a casing or exposed to the environment without. Most turbomachines belong to the former group of enclosed turbomachines, but some, such as the wind turbine, prop fan, and ship propeller, belong to the latter group of extended turbomachines.

An important flow phenomenon found only in fluid machinery employing liquid as the working fluid is cavitation, which indicates generation of gas bubbles at normal temperature of operation due to a decrease in the local static pressure. In pumps or hydraulic turbines, cavitation occurs by the rotating blades that cause low local pressure. Repeated breaking down of bubbles near the solid wall induces erosion damage and also noise. Thus, cavitation is an important factor to be considered in the design of hydraulic machinery. On the other hand, in fluid machines that use gas as a working fluid and operate at high speed, the compressibility of gas causes unique flow phenomena such as shock waves that are not found in hydraulic machinery.

A typical parameter, which is used to classify various types of turbomachinery, is specific speed. The specific speed is defined as a non-dimensional parameter combining operating parameters of turbomachinery as follows;

$$N_s = NQ^{1/2}/(g\Delta H)^{3/4} \tag{1.1}$$

Constant specific speed indicates the flow conditions that are similar in geometrically similar turbomachinery. However, if the gravitational acceleration, g, is assumed constant, the parameter becomes a dimensional parameter, $NQ^{1/2}/(\Delta H)^{3/4}$. The specific speed, N_s is the most important parameter in turbomachinery that can be used in the selection of turbomachinery type as shown in Figure 1.1. The range of specific speeds for a specified type of turbomachinery shown in Figure 1.2 indicates the range where the turbomachine type shows maximum efficiency.

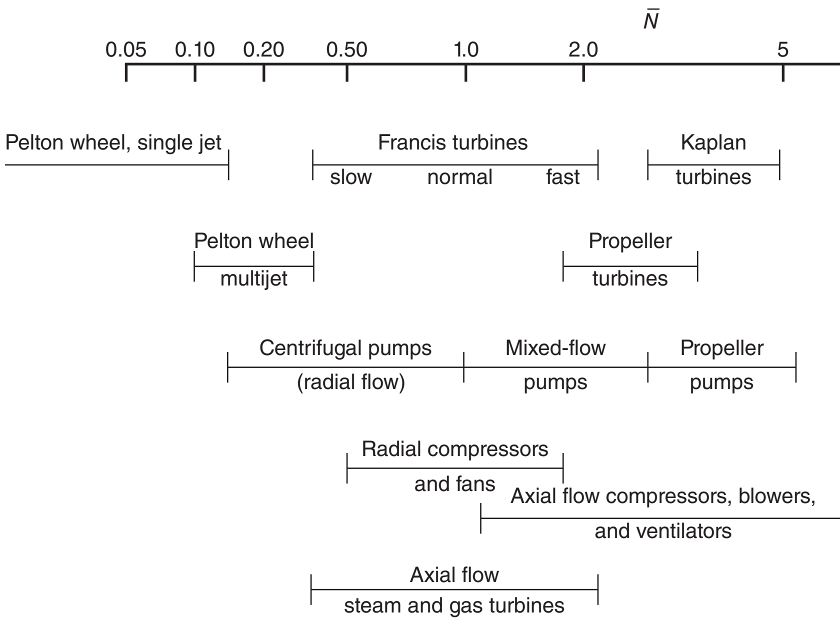


Figure 1.2 Specific speed suitability ranges of various designs. Source: Csanady 1964.

1.3 Analysis of Fluid Machinery

Analysis of turbomachinery should involve the analyses in a variety of fields; fluid mechanics, thermodynamics, solid mechanics, rotor dynamics, acoustics, material science, mechanical control, manufacturing, and so on. However, aerodynamic/hydrodynamic performance is essential in the evaluation of the basic performance of turbomachinery. Since it is difficult to include all the analyses here, only aerodynamic/hydrodynamic analysis and design methods are introduced in this chapter.

The history of turbomachinery is quite long. For example, waterwheels have been utilized by human beings for several thousands of years. The design of such ancient fluid machines was required even before the basic *theory* of fluid dynamics was set up. Therefore, the analysis method of fluid machinery was developed with the development of fluid mechanics. Until the numerical calculation of 3D Navier–Stokes equations became possible by using electronic computers in the middle of the twentieth century, analysis of turbomachinery was based on various approximate fluid mechanical theories as shown in Table 1.1. Analysis using inviscid equations and one-dimensional analysis using empirical formulas for energy losses are typical examples of such approximate analysis. Thus, many simple design methods based on these approximate analyses have developed over a long time, but the rapid development of electronic computers since the late twentieth century makes the numerical calculation of full Navier–Stokes equations practical. And, recently, 3D CFD has even become popular in the analysis of turbomachinery.

Direct numerical simulation (DNS) of Navier–Stokes equations for the wall-bounded turbulent flow was first realized by Kim et al. (1987). However, DNS cannot be used in analyzing practical flows due to excessive computational expenses. Since the numbers of spatial meshes and time steps required for DNS increase rapidly as the Reynolds

Table 1.1 Various approximations for flow analysis.

Governing equations	Assumptions
Stream function equation	Two-dimensional (2D) potential flow
Laplace equation (stream function or velocity potential)	Irrotational inviscid flow
Euler equations	Inviscid flows
Boundary layer equations	Boundary layer approximations
Stream function and vorticity equations	2D viscous flows
Parabolized Navier–Stokes (PNS) equations	If the streamwise pressure gradient can be prescribed in thin-layer Navier–Stokes (TLNS) equations, the numerical solution is independent of downstream boundary conditions
TLNS equations	If thickness of boundary layer is smaller than the body length, the streamwise diffusion terms can be neglected in Navier–Stokes equations
Full Navier–Stokes equations	

number increases, DNS analysis of turbomachinery flows is still impractical. Large eddy simulation (LES), which solves equations only for large eddies of turbulence by modeling small eddy motion, is an approximation of DNS, but LES still needs a huge amount of computing time and storage for turbomachinery analysis, as in the recent work of Pacot et al. (2016). Therefore, analysis using Reynolds-averaged Navier–Stokes (RANS) equations is the only practical method to solve full Navier–Stokes equations for turbulent flows in turbomachinery, and is thus implemented in most commercial CFD software. Because RANS equations are obtained by using Reynolds decomposition of instantaneous quantities, a turbulence closure model must be used for the Reynolds stress components to close the problem. However, no single turbulence closure model (Wilcox 1993) developed so far guarantees sufficiently accurate solutions for all types of turbulent flows. As turbulence models, the two-equation models, k-ε (Launder and Sharma 1974), k-ω (Wilcox 1988), and shear stress transport (SST) (Menter 1994) models have been most widely used for practical calculations. The SST model combines k-ε and k-ω models by implementing the k-ω model in the near-wall region and k-ε model in the region far from the wall.

The flow analysis methods for turbomachinery were classified by Lakshminarayana (1996) as shown in Figure 1.4. The flow in turbomachinery is complicated and 3D as shown in Figure 1.3. And, thus, full Navier–Stokes equations are required to be solved to resolve the complex viscous flow structures including flow separation. However, the zonal method is used when the solution of full Navier–Stokes equations in the computational domain is expensive. In this method, multiple zones are defined in the computational domain, different approximations are applied to different zones, and the solutions are integrated into the whole domain to get a complete solution. This method

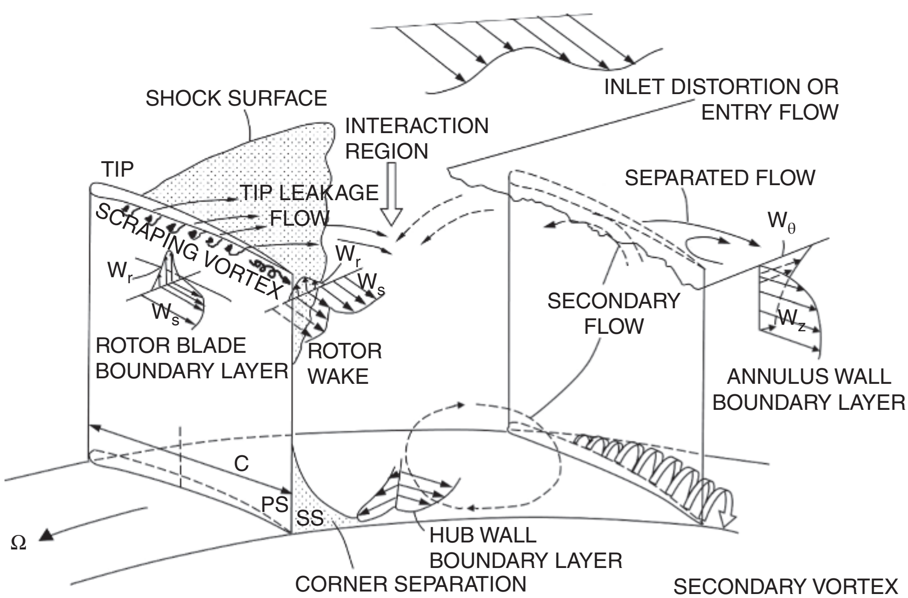
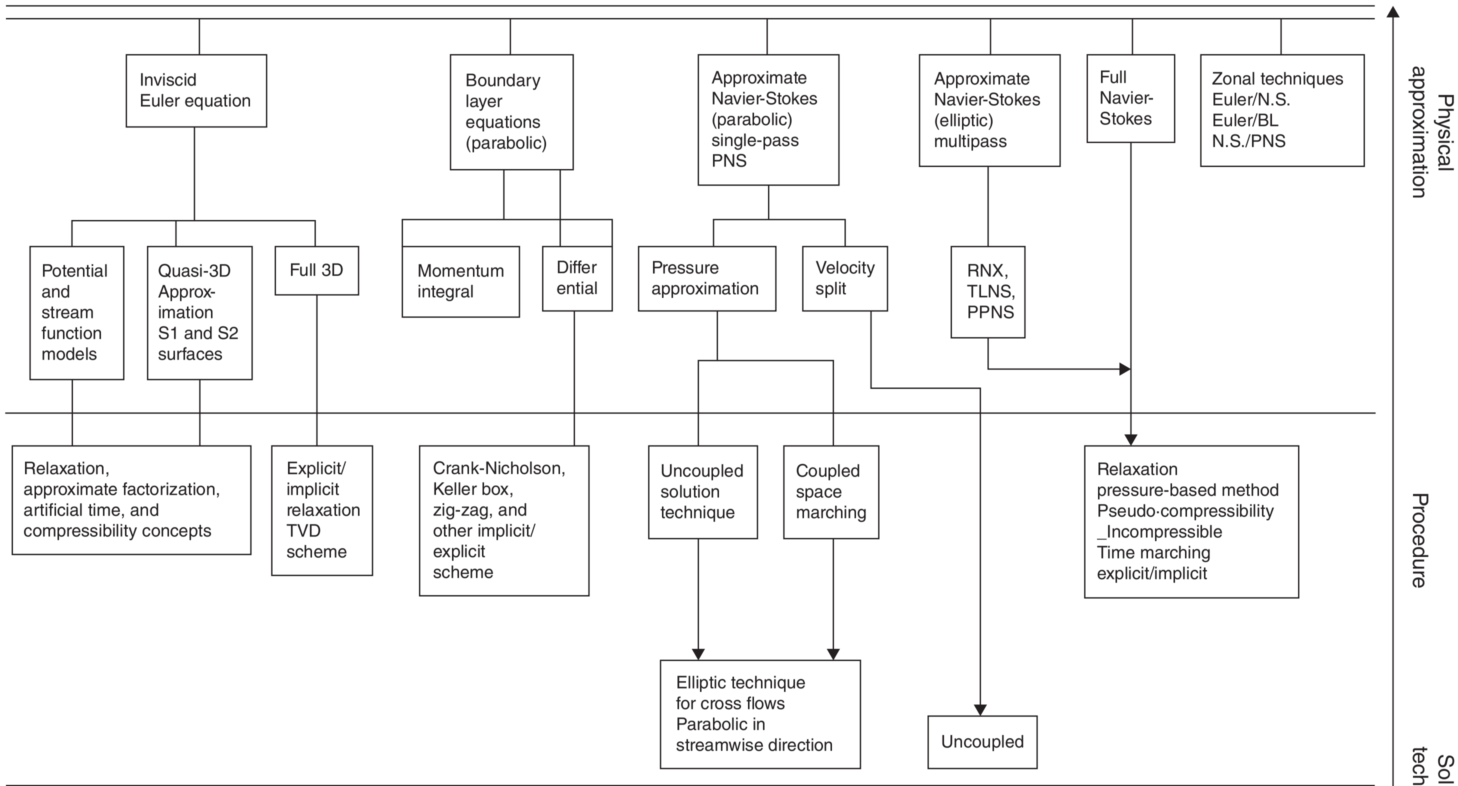


Figure 1.3 Flow structure in a rotor passage of an axial flow compressor. Source: Reprinted from Lakshminarayana 1996 (Figure 1.15 from original source), © 1996, with the permission of John Wiley & Sons, Inc.



Discretization: Finite volume/finite difference/Finite element
Algebraic equation: SOR, SLOR, ADI, preconditioning, conjugate gradient, multigrid, direct method

Figure 1.4 Flow analysis methods for turbomachinery. Source: Reprinted from Lakshminarayana 1996 (Table 5.2 from original source), © 1996, with the permission of John Wiley & Sons, Inc.

is complicated but less expensive without a great loss of accuracy. The computational errors involved in the analysis arise from different sources: incomplete physical models such as turbulence closure, discretization of the governing differential equations, and the solution procedure of algebraic equations.

Although the 3D analysis of turbomachinery flow using Navier–Stokes equations has become practical, it is still impractical to perform a whole design process using design optimization based on this analysis method due to excessive computing time. Thus, for a new design of a turbomachine, a preliminary design using approximate analysis methods is still needed to determine the values of the numerous (geometrical and operational) design parameters of the machine. As a second step, through a parametric study using 3D CFD with selected design parameters, some design variables that sensitively affect the performance of the turbomachine can be determined among the tested parameters. Then, a design optimization using these design variables would further improve the performance of the turbomachine. This is a most effective way to design turbomachinery with limited computational resources because design optimization using systematic optimization algorithms requires repeated analyses of turbomachinery flow and the number of repeated analyses is roughly proportional to the third power of a number of design variables.

1.4 Design of Fluid Machinery

Complex turbomachines, such as a gas turbine engine that consists of a multistage compressor, combustor, and multistage turbines, require many engineering considerations in their design, including thermodynamic, aerodynamic, and thermal analyses. However, for most other simple turbomachines, such as fans, compressors, pumps, and turbines, a relatively simple design process has been applied. Designs of turbomachinery have been developed over a long time, along with the development of aerodynamic/hydrodynamic analysis technology. Therefore, the procedure of turbomachinery design generally consists of several steps that perform different levels of performance analysis and design.

A typical method for aerodynamic design of turbomachinery blades follows the following steps.

1.4.1 Design Requirements

Design requirements and operating conditions of a turbomachine need to be specified in terms of flow capacity, RPM (revolutions per minute), pressure rise, efficiency, noise level, and inlet flow conditions.

1.4.2 Determination of Meanline Parameters

Design parameters such as hub-to-tip ratio, tip diameter, pitch, and chord length, are determined at the meanline as representative dimensions and on the basis of specific speed charts.

1.4.3 Meanline Analysis

Meanline performance analysis is performed using thermodynamic equations, flow deviation, and pressure loss models to estimate roughly the effects of design variables on aerodynamic performance. From this parametric study, proper fan design variables with feasible ranges for high-performance design can be determined.

1.4.4 3D Blade Design

The 3D shape of the turbomachinery blades is defined using the methods proved for camber line, blade thickness distribution, and stacking line. With specified design requirements, the baseline design of blade cross-section is obtained by mean camber line. Also, the thickness distribution of the blade cross-section is built using a distribution of points defined as a fraction of camber line length. The design of the 3D blade is completed by determining the stacking line of blade elements along the blade span considering sweep and lean and by performing a conformal mapping of the planar surfaces of the blade sections to the cylindrical surfaces.

1.4.5 Quasi 3D Through-Flow Analysis

Based on the 3D blade design, a quasi 3D through-flow method analyzes the aerodynamic performance of the turbomachine using Euler's equation, pressure loss models, and the equation of motion for radial equilibrium. This analysis predicts blade-to-blade and spanwise flow distributions and provides the aerodynamic performance through the mass-averaging of predicted flow field data. However, this analysis method has a problem in predicting 3D flow structures including leakage flow, secondary vortex, and endwall boundary layer.

1.4.6 Full 3D Flow Analysis

To precisely analyze the 3D flow field and aerodynamic performance of turbomachinery, full 3D flow analysis using Navier–Stokes equations can be used. This analysis method requires complicated grid generation and a reliable turbulence closure model for the complex 3D flow field in the turbomachine, and thus much more computing time and effort than the approximated through-flow analysis.

1.4.7 Design Optimization

Owing to the recent development of numerical methods and computers, full 3D CFD analysis can be directly used for single- or multi-objective design optimization of turbomachinery. However, a design method using an optimization algorithm requires repeated evaluations of the objective function(s) using 3D CFD, which generally takes a lot of computing time and thus there is a limitation in the number of design variables for optimization. Therefore, in the initial stage of the optimization, a parametric study using a number of geometric and/or operating parameters is usually performed in order to select the design variables and their design ranges for optimization.

1.5 Design Optimization of Turbomachinery

Although analyses of complex turbulent flows in turbomachinery take a long computational time, the recent development of high-speed computers has made it practical to optimize the aerodynamic or hydrodynamic design of turbomachinery using governing equations for 3D viscous flows, such as RANS equations. Systematic optimization using high-fidelity analysis produces high-performance and reduces computational and experimental expenses in turbomachinery design.

General objectives of turbomachinery design are efficiency, pressure ratio, weight, and so on, and geometrical/operational parameters are generally used as design variables for optimization. In a design called inverse design, the optimum turbomachinery geometry is deduced from prescribed ideal flow conditions (and thus from prescribed objectives). This inverse design only requires low computational cost, but there is a difficulty in specifying the target flow field where the designer's insight and experience are required. If optimum objectives are found by changing the design variables, the design is called direct design or design optimization. The present book is mostly concerned with this design method. The design optimization methods can be classified into two categories: gradient-based and statistical methods.

The gradient-based methods are categorized into finite difference, linearized, and adjoint methods depending on how the gradients of the objective function are calculated. Because the computing time for the finite difference and the linearized methods depends on the number of design variables, these methods are not suitable for design problems with a large number of design variables. The adjoint method has an advantage in computing time because its computing time does not depend on the number of design variables; however, this method is not being widely used because of its complexity and counter-intuitive natures (Wang and He 2008).

As a statistical approach, surrogate-based optimization methods are widely used in the design optimization of turbomachinery due to their easy implementation and affordable computing time. By employing surrogate model(s) of the objective function(s), it is possible to largely reduce the number of objective function calculations required for the optimization. The modeling fidelity is important in surrogate modeling. Various surrogate models have been developed so far (Queipo et al. 2005), and weighted average models have also been suggested based on global error measures (Goel et al. 2007). The simulated annealing and the genetic algorithm are also available for optimization but are known to have relatively large computing times.

Parametric geometric modeling is an essential element in design optimization of turbomachinery. To optimize the shape of a turbomachine, the geometry must be modeled. Lieber (2003) suggested that the techniques used to describe the geometries of components and flow paths are required to have sufficient generality for the accommodation of complex configurations. Transfer of information for flow-path geometry to other functional groups must also be considered in the design process. In many turbomachinery optimizations, the Bézier curves are used to parameterize the geometry, and related control points can be used as the design variables. The B-spline curve, which is a piecewise collection of Bézier curves, is used when a single Bézier curve cannot be used for the shape due to complexity. The parameterization of turbomachinery blades by Bézier curves have two advantages: the curves can be controlled by a small number of points to produce a smooth profile and, thus, require a small number of design variables.

References

Csanady, G.T. (1964). *Theory of Turbomachines*. New York: McGraw-Hill.

Goel, T., Haftka, R., Shyy, W., and Queipo, N. (2007). Ensemble of surrogates. *Structural and Multidisciplinary Optimization* 33 (3): 199–216.

Kim, J., Moin, P., and Moser, R.D. (1987). Turbulence statistics in fully-developed channel flow at low Reynolds number. *Journal of Fluid Mechanics* 177: 133–166.

Lakshminarayana, B. (1996). *Fluid Dynamics and Heat Transfer of Turbomachinery*. New York: Wiley.

Launder, B.E. and Sharma, B., I. (1974). Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. *Letters in Heat and Mass Transfer* 1 (2): 131–137.

Lieber, L. (2003). Fluid dynamics of turbomachines. In: *Handbook of Turbomachinery*, 2e (ed. E. Logan and R. Roy). CRC Press.

Menter, F.R. (1994). Two-equation eddy viscosity turbulence models for engineering applications. *AIAA Journal* 32: 1598–1605.

Pacot, O., Kato, C., Guo, Y. et al. (2016). Large eddy simulation of the rotating stall in a pump-turbine operated in pumping mode at a part-load condition. *ASME Journal of Fluids Engineering* 138 (11): 111102.

Queipo, N.V., Haftka, R.T., Shyy, W. et al. (2005). Surrogate-based analysis and optimization. *Progress in Aerospace Sciences* 41: 1–28.

Wang, D. X., and He, L. (2008). Adjoint Aerodynamic Design Optimization for Blades in Multistage Turbomachines – part I: Methodology and Verification, *ASME Turbo Expo 2008* GT2008-50208.

Wilcox, D.C. (1988). Reassessment of the scale-determining equation for advance turbulence models. *AIAA Journal* 26 (11): 1299–1310.

Wilcox, D. C. (1993). Turbulence modeling for CFD. 5354 Palm Drive, CA: DCW Industries, Inc.