2000 유체기계 연구개발 발표회 논문집, pp. 3 ~ 8
부경대학교 용당캠퍼스 종합복지관, 2000. 12. 1 ~2.

New Development of Turbo-compressor and Fan on Design, Re-rating and Numerical Simulation in China

Chuangang Gu*

Institute of Fluid Machinery, Energy & Power School, Shanghai JiaoTong University
No. 1954, Huashan Road, Shanghai, 200030, P. R. China
Tel/Fax: +86-21-64470256
Email: cggu@mail.sjtu.edu.cn

ABSTRACT

The new development of turbo-compressor and fan in China is introduced here. For improving the performance of compressor and fan, we have studied the design method, numerical simulation, unsteady flow and the strategy for anti-surge control, got some achievement.

INTRODUCTION

Compressors and fans are key equipment in the petrochemical industry, iron and steel factories, air-condition companies, etc. Their efficiency and reliability directly relate to the economic benefit of these enterprises. In the recent one or two decades, turbomachinery is being developed very quickly. I would like here to introduce the development and trend in aerodynamic design, numerical simulation, unsteady flow and the strategy for anti-surge control of compressors and fans in China.

The large-scale compressor group requires several technologies: group performance, aerodynamic design technology, manufacturing engineering, measurement and control, seal technology, group rebuilding, energy-saving, etc. In this article the large-scale group includes centrifugal and axial type turbomachinery.

1. Advanced Design and Manufacturing Technology

The requirements for advanced machines include, high efficiency, wide operation range high speed, miniaturization, easy assembly (low price), superior manufacturing technology, advanced materials, guarantees for safety and reliability, etc.

1) High efficiency

The efficiency of large axial compressor is generally between 0.88 and 0.92, for example, GHH, Sulzer, Elliott and Shaanxi Blower Works in China. They use adjustable stator vanes. The efficiency of axial compressors manufactured by the JiLiFu factory in Russia can reach 0.92, and in the Yanwa factory it is around 0.90. The efficiency of high-flow rate centrifugal compressors could be above 0.85. Many companies in the USA, Europe, Russia, Japan, such as Demag, Delaval, Elliott, Sulzer, Nuovo Pignoni, GHH and ShenYang Blower Works, Shaanxi Blower Works, JinXi Chemical Machinery Factory in China could reach such high level.

Now, the question is how to increase the efficiency of centrifugal compressors as well as fans at a condition of high-pressure ratio and low flow rate. It is very difficult to increase the efficiency because the flow passage of the low flowrate compressor is very narrow and the frictional losses take up a great amount of total losses. Moreover, the efficiency of impeller has a complex relationship with its geometric size, such as D_h/D_2, the axial width of the impeller, Z, the shape of shroud, hub and blade, and so on. In the case of a fixed impeller diameter, D_2, a larger D_h, will shorten the length of the blade and increase the load along the blade. It would generally reduce the efficiency of the impeller. When Z is too small, the change in flow direction becomes too sharp and the efficiency would also decrease. However, for multistage compressor, the slim shaft or the large distance between two bearings would result in a rotor dynamics problem if D_h is too small or Z is too large. So the efficiency cannot be high for last stages of such a compressor.

For the low flow rate centrifugal compressor, when the flowrate coefficient [1] is $0.012< \phi<0.02$ and $b_2/D_2<0.02$, the efficiency between 0.68 0.75 is considered to be acceptable. For $\phi<0.012$, for example, the model stage at Shenyang Blower Works and some companies in Russia ($\phi=0.0105$, and $b_2/D_2<0.01$), the efficiency is 0.6; Some model stages of Shenyang Blower Works are, generally, $\phi=0.006$, $\eta=0.55$.

For increasing the efficiency and saving energy, the key lies in how to guide the design by three dimensional flow theory and CFD. The main principle is to make the fluid flow smoothly and the pressure diffusion reasonable, without serious separated flow inside.

Currently some good CFD software is used to predict and analyze the flow field. The magnitude and position of separating regions can be calculated qualitatively. It is very useful for design. For all the high flowrate stage, we adopt the three dimensional impeller (the blade is twisted). But when $b_2/D_2<0.02$, the performance difference between the three dimensional and two dimensional impeller is not so obvious. Therefore the two dimensional impeller is often used instead of a three dimensional one due to its lower cost and shorter axial length. In order to improve the efficiency of the low flowrate stage a new type of impeller and diffuser as well as return channel and/or scroll should be used. As is well known, the surface roughness of the parts influences greatly the performance, sometimes, it could decrease the efficiency by *5-6%*. Similar to those in Europe and American, the factories in China would like to use the vaneless diffuser because the stage with vaneless diffuser has the advantage of a wide operation range although its efficiency and pressure ratio is lower.

However, many companies in Russia often use two-dimension impellers with vaned diffusers. They have made extensive studies of his topic and developed many different types of two-dimension impellers with vaned diffuser in the middle flowrate range. The efficiency and operation range could be compared with the three-dimension impeller with vaneless diffuser. We have even helped some factories to develop new types of two-dimension impeller with vaned diffusers and obtained good performance. The two dimensional vane has certain competitive advantage due to its short manufacturing cycle and low cost.

2) Wide Operation Range

Adjusting the rotation speeds is an old method to enlarge the operation range, and it is economical. The method for adjusting all or part of the stator blades is commonly used to enlarge the operation range for axial compressors, but it will lose efficiency to some extent at the design point. Sometimes the partial-height vaned diffuser, partial-length vaned diffuser and some special shapes of vaned diffusers, which seems wedge-shaped are adopted for centrifugal compressors. The adjustable inlet-guide-vane was already adopted in every stage to enlarge the operation range for some DH type of compressor. Certainly, a good proportional linkage between all stages should be adopted to ensure the matching of all stages.

There are several methods to enlarge the operation range: From the aspect of blade design, for example, making a better load distribution along the blade, designing carefully the shape of the blade leading edge which reduces the sensitivity to the incidence angle. And we should pay attention to the performance curves with negative slope when developing new stage. These yield a more stable system's characteristic.

When designing the diffuser, the width is important. Some people use a special anti-separation board in the vaneless diffuser and obtain a wider operation range. The method of carving notches on the casing of axial-flow fans or compressors has also shown to benefit the centrifugal stage.

The method of active control of anti-surge has been researched in laboratories. An inverse phase wave is generated, which can delay stall. But the method can not be used in practical designs now. In addition, the method of carving small notches on vane surfaces in different ways might enlarge the operation range of compressors and fans.

3) High Rotation Speed and Miniaturization

It is well known that increasing the impeller tip speed U_2 is one of the most effective methods for increasing the pressure ratio. For the same pressure ratio, increasing the rotation speed could make the radius of the impeller decrease. Certainly, increasing the rotation speed also result in a series of problem, such as, strength vibration, the gearbox technology with high speed-ratios decrease of efficiency caused by increasing Mach numbers in the flow passage, etc. It involves aerodynamics, material mechanics, rotor dynamics and other subject. But this is the direction of today's development. It has been reported that, in the next century, the Mach number of centrifugal compressor will be in the range of 1.1 1.5 and the impeller tip speed would be up to 600 m/s. Then, the pressure ratio of air in single stage compressor could reach to 6 7.

Now, more and more compressors are running above the second critical speed. Some companies use a high speed-ratio gearbox to make the rotational speed increase to 30000 50000 RPM and the pressure ratio

---

[1] $\phi = Q\Big/\left(\frac{\pi}{4}D_2^2U_2\right)$

of air in two-stage compressors could reach 8 12, which is a good solution for the design and manufacture of compressors in the range of small flow rate and high pressure ratio.

4) Anti-surge System

The anti-surge system (Compressors Control Company) includes an anti-surge controller for every stage of the compressor. The control system has an inlet pressure sensor, inlet flow-rate sensor, outlet pressure sensor, outlet flow-rate sensor, speed sensor and anti-surge valve. The controller will immediately open the anti-surge valve when surge occurs, comparing the measured running condition with the data stored in the computer database.

The position of the anti-surge valve is ensured by the controller via the signal transferred from pressure sensor. It ensures that the compressor operates above a minimum safe flow-rate. The warning limits can be adjusted by the non-stable flow impulse signal.

As is well known, the stall is from the impeller or diffuser, but surge is related with the compression system. We established a test bed of a system, which includes a small compressor, pipe, valve and some tanks with different volumes. When surge occurred, we measured the surge frequency and the forward-backward flow by using a hot-wire. We found that there are different surge frequencies in the system for different pipe lengths and tank sizes. A model for predicting surge of a system consisting of compressors, pipes, valves and tanks has been made. The model results compare well with the experimental data. It was found that the unsteady outlet condition had great influence on the performance of the compressor. The performance curve tested from steady condition could not be used to express the unsteady characteristic of the compressor.

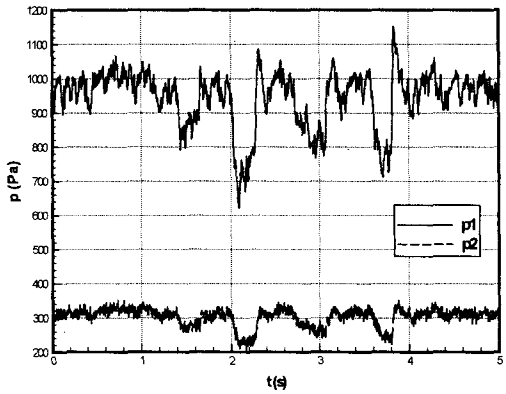
We want to develop a new method to predict the exact surge point of a compressor system before the machine is manufactured. Then we will know the period of time before the first surge and the second surge wave according to the surge frequency of the system before assembly. We can then select the sensitivity of the surge-valve and sensors to avoid the second surge wave and ensure that the system runs safely.

2. Re-Rating Existing Machines

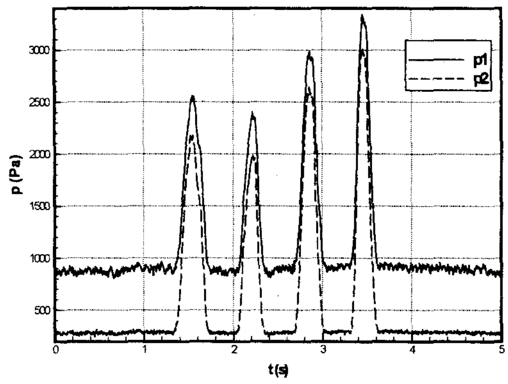
At present economic conditions, every country has a great interest to rerate existing machines and expand their production capability. Avoiding building and installing a new machine with a new compressor base and casing, and pipe lines, etc., the rerating and modifying of older machine saves considerable investment and time.

Recently, a copy Demag-Delaval Corporation's report about rebuilding and increasing the efficiency of turbo-compressor in ethene production indicates that the efficiency of compressors manufactured during the seventies can generally be improved by 20% 40% (shaft speed is at 100% 105%).

Now, there are many re-rating opportunities in China. Some of these are contracted to foreign companies, some to Chinese companies. We have also modified some combined compressors and steam turbines without changing the casing and base structure of the compressor. Generally the flow-rate can be increased by 15 25 %, the pressure ratio by 10 % (according to the user's request). And the efficiency, using modern design methods can be increased by over 3 5%.

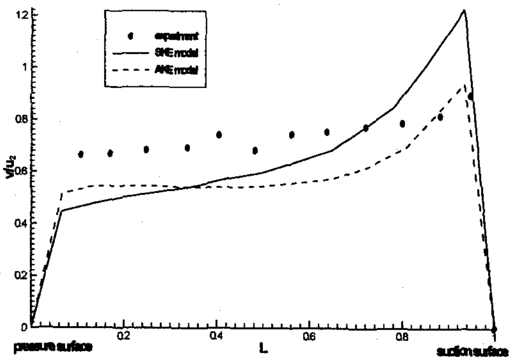


(a) Adjust Inlet Valve

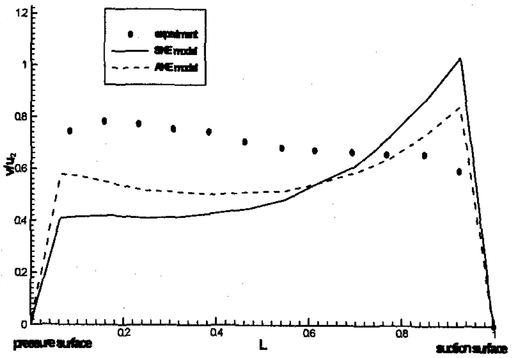


(b) Adjust Outlet Valve

Fig. 1 Instantaneous Pressure With The Valve Opened and Closed



(a) in the middle of the passage



(b) at the outlet of the impeller

Fig. 2 Relative Velocity Distribution Along The Width of Impeller Passage

The main objectives for re-rating are as follow:

a. Increasing the inlet pressure at the first stage;
b. Increasing the effectiveness of the heat exchangers to improve the cooling ability and pressure loss, thus reducing power consumption;
c. Improving the shape of the flow passages and the manufacturing quality (including impeller, diffuser, return channel, scroll etc and using 3-D impellers) to reduce power consumption;
d. Modifying the quality of seals, bearings and couplings to save power and avoid frequent failures;
e. Increasing the rotation speed by 5 percent;
f. For some machines, install water or air spraying equipment to scour the sediments;
g. Increasing the flow capacity to increase the plant's production;
h. Improving the operating range of the existing machine to better accommodate fluctuating demands.

3. Numerical Simulation for Design

There are so many kinds CFD software, for example, FLUENT, STAR-CD, PHOENICS, etc. I will not introduce the applicability of these codes here. At our laboratory, the quasi-three dimensional codes were developed by ourselves to calculate the internal flow field in compressors and fans. We know not only the fundamentals of fluid mechanics, but also the structures and the characters of the codes. Then we could analyze the load distribution of the blades according to calculated results by the code. As for the effects of fluid viscosity or the flow loss, we usually considered them in advance by experience.

In order to improve the design method for turbo-compressor and fan, the viscosity of fluid should be fully taken into account. Now the k-ε turbulent model and the other similar models are adopted to calculate the flow fields in the centrifugal compressors and fans. There are several experimental coefficients in these turbulent models, which were derived from the experiments of flow around the flat plate and the uniform turbulent flow. Once are there high rotating speed, high adverse pressure gradient, great curvature in the field, application of the turbulent models was limited. Therefore, the application of k-ε model in turbomachinery was studied.

The eddy-viscosity representation for Reynolds stress of anisotropic k-ε (AKE) model considering the effects of curvatures and rotation has been derived by statistics. The numerical coefficients in AKE model are determined and revised in terms of experimental data. The flow field in a centrifugal impeller was simulated by using AKE model and standard k–ε model (SKE) respectively, as shown in fig. 2. It is the relative velocity distribution along the width of the impeller passage. The solid dot in Fig. 2 is the experimental data we measured by Laser Doppler Velocimeter (LDV). Compared with the experimental data, the computational analysis by AKE model is closer to the experimental data than SKE model.

Sometimes, because the trace particles can not enter in some corners of the impeller, we can not get the interested information from such regions by LDV. Now we use Particle Image Velocimeter (PIV) to measure the unsteady flow field between impeller and vaned diffuser.

REDUCING NOISE OF AXIAL FAN

We ever calculated the flow field in axial fan by using our code. We found that there was a large size vortex behind the blade near the hub. The vortex had great influence on the performance of the blade and the following stages. We tried to weaken the intensity and

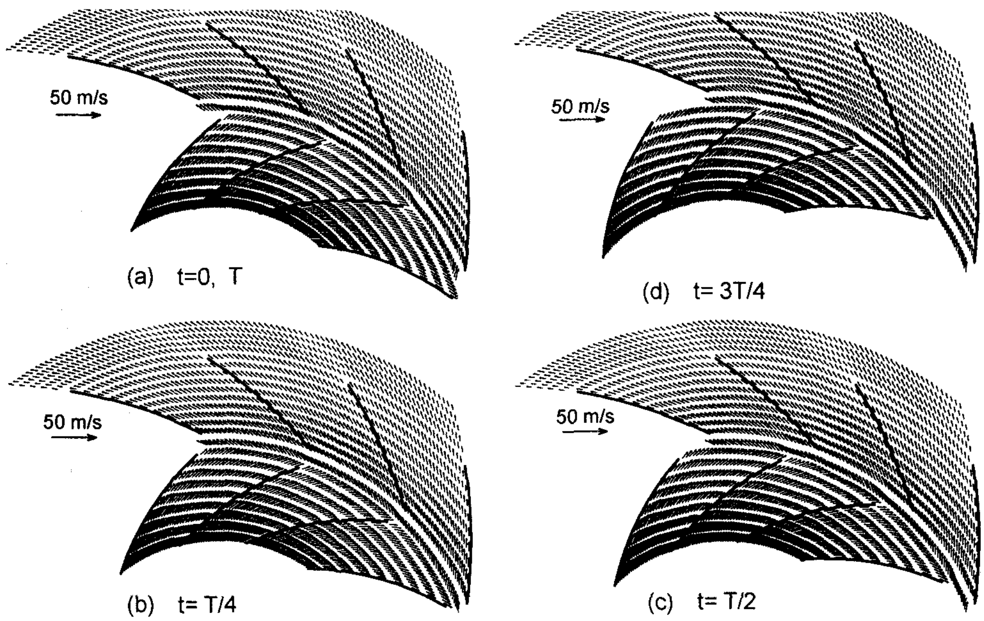


Fig. 3 Vector In Impeller And Vaned Diffuser At Different Time In One Cycle

size of the vortex by modifying the shape of the blade. The flow field with the new shape of the blade was calculated, the result was better than that one before modified. By manufacture and experiment, it was found that the performance is greatly increased and the noise is low. Regrettably, we didn't test the fan with old shape. Now, we have designed many fans by the method for architecture, air conditioner.

It is verified to be an effective design method. We can design turbomachinery with the high performance and low noise by controlling the vortex.

NUMERICAL SIMULATION FOR UNSTEADY FLOW

Recently, numerical simulation of unsteady turbomachinery flow has become a very popular area, which appear to be a natural trend as computers are getting more and more powerful.

Because of the rotation of impeller, the flow field in turbomachinery is inherently unsteady, wherever in stator or in rotor. It cannot get a satisfied answer to the whole flow field only by means of studying the flow field including only single relatively moving element. In order to understand the whole flow field with stator and rotor it needs to transfer the flow modes in different regions into a unified mode and to educe a series of universal control equations that are suitable to different regions. Then the interactions between the different regions can be considered simultaneously.

We derived the integral equations with absolute and relative velocity to simulate the whole flow field at the same time. The interface between the different regions is sliding surface because there are different velocity systems for rotor and stator regions. We have used sliding mesh and unstructured mesh to simulate unsteady flow in centrifugal fan with vaned diffuser. The simulation result is shown in Fig. 3.

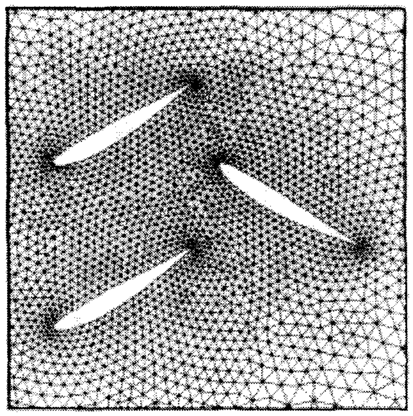


Fig. 4 Unstructured Mesh

The unstructured mesh can be used to create complicated geometric regions conveniently. The scheme is similar to that in finite element method. We have a lot of works to do at this aspect. Fig. 4 is the unstructured mesh with three airfoils.

As is well known, there is inherently difference between unsteady and steady flow. Almost every large research institutes such as MIT, NREC, P & W have been studying the unsteady flow, have the unsteady CFD software. The application of unsteady CFD is to guide the turbomachinery design, unsteady design method. Now there isn't a clear definition of the unsteady design method.

Currently, the design process is

1) Designing by one or two dimensional method to get preliminary profile of the flow field;
2) Analyzing the flow field by quasi-three dimensional method;
3) Modifying the profile by the distribution of velocity and pressure.

Then analyzing, modifying, until the satisfied profile is found. Strictly, it is the direct problem, not the inverse problem.

We wish the unsteady design method could be used to evaluate the performance of the new machine. That is our effort in the future. We will set up a criterion about unsteady design method. We aim at directly designing turbomachinery with good performance and good unsteady performance by the criterion. At least now, the unsteady design method should be used to estimate the performance by simulating the unsteady flow.

REFERECES

Chuangang Gu, Yongmiao Miao, Blade Design of Axial-Flow Compressors by the Method of Optimal Control Thoery-Physical Model and Mathematical Expression, ASME J. of Turbomachinery, Vol. 109, pp99, 1987

Chuangang Gu, Yongmiao Miao, Blade Design of Axial-Flow Compressors by the Method of Optimal Control Thoery-Application and Results, ASME J. of Turbomachinery, Vol. 109, pp102, 1987

Chuangang Gu, Laiqin Luo, Yongmiao Miao,Experimental Investigations of Flows Through a Plane Cascade at Large Angles of Attack with Separation, ASME J. of Turbomachinery, Vol. 110, pp323, 1988

Ji Dai, Chuangang Gu, Yongmiao Miao Temperature Rise During Deep-Surge in Centrifugal Compressor System , ASME paper 95-GT-438 (Washington D.C.) 1995.8

Chuangang Gu, Ji Dai, Suping Wen, Moujin Zhang A Study of Caculational Model For Predicting Surge Line in A Centrifugal Compression Sustem with Cooler and Condenser, ASME Conference, Asia , 1997.9

Zhengxian Liu, Exprimental and Computational Investigation of a Rotating Centrifugal Impeller Flow Field, Ph. D. Thesis, Xi'an Jiaotong University, 1997.10

Chuhua Zhang, Unstructured Grid Numerical Simulation of 3D turbulent flow and Application Investigation in Centrifugal Fan, Ph. D. Thesis, Xi'an Jiaotong University, 1999.2

Tong Wang, Numerical Study of Unsteady Flow Field Due to Rotor-Stator Interaction in Turbomachinery and Experimental Investigation of a Pipe System, Ph. D. Thesis, Xi'an Jiaotong University, 1999.3