The Aerodynamics of the Inlet of Centrifugal Compressor

J. Mulugeta
York International Corporation

Follow this and additional works at: http://docs.lib.purdue.edu/icec

http://docs.lib.purdue.edu/icec/1341

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.
Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at https://engineering.purdue.edu/Herrick/Events/orderlit.html
THE AERODYNAMICS OF THE INLET OF A CENTRIFUGAL COMPRESSOR

Jarso Mulugeta
York International Corporation
York, PA U.S.A.

ABSTRACT

In their development phase, centrifugal compressors are tested for aerodynamic performance using large test beds and according to the ASME test code PTC 10. The code specifies testing arrangements and the rating methods that are consistent with the quoted levels of uncertainties. In particular, the code specifies for an inlet device, a ten diameter-long straight section of pipe with specific type of flow straighteners. Similar specifications for the discharge pipe and the geometry before and after the orifice flow meter, are established to assure for reliable performance data. Real life application, however, requires inlets with treacherous bends ahead of the pre-rotation vanes that are quite different from the test setup. As a result, the performance rating usually needs adjustment to match chiller performance when the compressor is applied in a system.

The purpose of this experimental and numerical study is to examine the flow field in an inlet device and to quantify the effect of the inlet shape as well as the angular position of the pre-rotation vanes.

INTRODUCTION

For a fixed rotational speed, the characteristic curve of a particular centrifugal compressor—also known as the Mach line—describes the unique relationship between its capacity and imposed head. This relationship, in the form of a map with associated compressor efficiency islands is usually used by the manufacturer in a computer program in order to rate the machine for a specific application.

The range of application of the compressor on a water chiller is usually extremely large [1]. On a compressor performance map, instead of just the characteristic curve, operation in the entire region below it is required to cover the large variation in weather and cooling demand conditions. Two devices that are complementary for achieving the broad range of application are the variable speed drive and the pre-rotation vanes. Because the latter is less expensive than the former, it enjoys wide usage in the industry.

For such an important device, the pre-rotation vanes usually receive very little attention in the literature of the design process of compressors. Concern is limited to minimizing the total pressure loss across it. Typically, the vane cross-section view has one of the NACA profiles; sometimes with a modest camber in it. In one particular recent design, two rows of vanes were used with the first set rotatable over only half the range of the second row in order to reduce the wake for a particular amount of flow rotation. Often, the impeller design process ignores the presence of the vanes altogether, since at full load design condition they are in an axial position and have little influence on the ensuing flow.

In 1967 and 1968, York International aerodynamicists had concerns whether the popular arrangement was effective in cost and aerodynamic performance. In addition, they were interested in quantifying the effects of the complex mitered elbows upstream of the pre-rotation vane arrangement. In order to achieve this, a program was initiated and a consulting firm, Dynatech,[2] was employed to complete a very comprehensive study of five different kinds of devices that can produce the two outcomes of pre-rotation vanes: throttling and swirl.

The object of the study reported here is to reproduce numerically the results of one of these tests exclusive of the upstream mitered elbow.
FLUIDS IN SOLID-BODY ROTATION

Rarely do the Navier-Stokes equations allow easy analytical solutions for practical problems. The case of flow in a "solid body rotation" with uniform axial velocity contained in a cylindrical pipe and going through a contraction—for example a reducer—is one of the exceptions if we can ignore the effects of viscosity. This problem is of interest to us because, as in our inlet, the flow is primarily a swirl flow, the shape of the pre-rotation vane housing is basically that of a reducer; and in each case we are interested in the flow in the cylindrical exit of the cone. In this simplified problem, the mechanism that produced the swirl is of no interest to us; rather, the dynamics caused by the contraction of the pipe diameter is. Hopefully, using the pattern we observed in this simple case, we could characterize the flows exiting our pre-rotation vane housing.

The equation of motion, when viscosity is ignored can be written as:

$$\frac{\partial \mathbf{u}}{\partial t} - \mathbf{u} \times \omega = -\nabla \left( \frac{1}{2} q^2 + \frac{p}{\rho} - \mathbf{g} \cdot \mathbf{r} \right),$$

where $q^2 = \mathbf{u} \cdot \mathbf{u}$, $\omega = \nabla \times \mathbf{u}$ is the vorticity; and the tensor identity $\frac{1}{2} \nabla \left( \mathbf{u} \cdot \mathbf{u} \right) = \mathbf{u} \cdot \nabla \mathbf{u} + \mathbf{u} \times \omega$ is used.

When the inlet radius of the reducer is $a$ and the discharge radius is $b$ and at the large entrance the fluid has an angular velocity $\Omega$ and a uniform axial velocity of $U$, and with all the conditions above satisfied, the solution at the discharge diameter becomes:

$$\frac{u_z}{U} = 1 + \left( \frac{a^2}{b^2} - 1 \right) \frac{\frac{1}{2} kb J_0(kr)}{J_1(kb)},$$

$$\frac{u_\theta}{\Omega r} = 1 + \left( \frac{a^2}{b^2} - 1 \right) \frac{b J_1(kr)}{r J_1(kb)}$$

$k = 2\Omega/U$ is a measure of the vortex strength at the inlet; and the result is valid only in the cylindrical section at the exit where the flow becomes independent of $z$. The solution thus depends on the parameter $k$, radius $r$ and the ratio of the radii at the inlet and discharge because the dynamics involves vortex stretching.

The expressions involving the Bessel functions can be calculated from the series representations of them:

$$J_n(z) = \sum_{m=0}^{\infty} \frac{(-1)^m \left( \frac{1}{2} z \right)^{n+2m}}{m! (n+m)!}$$

Use of just the first four terms give results accurate to the 5th decimal place. Therefore, we can set the values of $a$, $b$, and $U$ to the same value as in our experimental setup and continue varying the vortex strength $k$ to see its effect on the flow field. As it turns out the "telltale" sign of the solid body rotation lies in the distribution of $u_\theta$ along the radius. It will always show the $u_\theta/r = \text{constant}$ kind of distribution, while the axial velocity distribution changes dramatically to support the correct vortex strength. This distribution includes the possibility of local flow reversal. For low values of $k$ the axial velocity distribution along the radius is almost constant, while the tangential velocity varies with $r$. As the value of $k$ is increased, first the axial velocity varies gently like the inverse of radius proceeding with a sharper slope as $k$ is further increased until the flow begins to reverse on the outer diameter. The solution can not be valid beyond the point of flow reversal, as its formulation involved the definition of a conservative field of stream functions from which the velocities were determined by taking its gradient. It will be difficult to construct such a field if flow reversal is allowed. This is not to conclude that the solid-body rotation type of flow cannot support reverse flow at its center. Rather, that this solution can not be expected to show it. On the other hand, the solution does show that if flow reversal is to occur, it will first take place at the outer diameter rather than the center as the vortex strength is increased.
EXPERIMENTAL STUDY

Figure 1 is a schematic of the test setup. A straight inlet section with a bell-mouth was placed upstream of the pre-rotation vane assembly. Velocity traverse measurements were taken at the discharge of the vanes, where the impeller eye would have been located in a compressor configuration. The pressure was measured further downstream in the plenum. This measurement is what is used to calculate the total pressure loss in the assembly. The temperature and the flow rate were measured on the straight 6 in section. The 600 cfm blower down stream of the entire assembly can produce enough flow to achieve a velocity of up to 244 ft/s in air at the traverse section. The inlet was kept at atmospheric pressure.

A cobra probe and a prism probe is used to measure the flow angle and the dynamic head at the traverse section. Since this traverse mechanism can also be moved around circumferentially by 90°, it allowed measurement over half the entire cross section. The probe (the cobra probe for example) traversed from within 0.83 in of the insert wall—which is ten times the probe diameter—to .03 in of the far wall per the specification of the manufacturer and good practice.

The particular traverse data of interest was taken with the pre-rotation vanes rotated by 38° from axial position. At this position the flow rate was 350 ft³/min; and the average axial velocity was 149 ft/s at the cross section where the traverse data was taken. The corresponding average gas flow angle was 35°; with a total pressure loss of 1.83 dynamic head. The total pressure loss when the pre rotation vane was in the axial position is only 0.63 dynamic head.

Because of the odd number of vanes and since the wakes and the jets down stream of the vanes were also rotating, the traverse data was dependent on the traverse angular position. As the probe went in and out of the wake and jet region, the measurement showed fluctuations in the various variables that were measured [4]. In other words the problem was not an axisymmetric problem.

The measured values of axial velocity, tangential velocity and gas angle can be seen on figures 2, 3 and 4.

NUMERICAL ANALYSIS

As the application of numerical methods grows in the turbomachinery industry, designers are looking for a quick way to marry old methods with the new-found tools. One method is to recalculate old experiments and in the process calibrate the program. This study is one such effort. We have already resolved all of the engineering issues through an extensive test program as declared above. The question here is, could we have also done it using a 3-D CFD code if it had been available? The answer, as it will be evident, is clearly yes.

Although the pre-rotation vane assembly flow is a periodic problem, with a period of $\frac{2\pi}{p}$, the entire assembly is modeled using 99, 505 grid nodes. The reason for modeling the whole assembly is, the goal was to include a mitered elbow inlet calculation; in which case the solution would have lost its periodicity and would have required the full model. The other issue that needs mentioning is the issue of turbulent quantities. This work should really not be judged for the accuracy of these quantities because, the test program did not include these measurements, and any value we used as inlet condition was a total guess. Most of all, our aim is the verification and calibration of the tool against measurements of the mean flow quantities.

In order to generate the grid, a template, Turbogrid, was used just for one period of the vanes. Then by transforming the grid coordinates eight times the vane section was completely mapped. The sections upstream, downstream, and center were done separately using CFX-build, and then all sections were assembled and connected together inside TASCflow.

The actual testing had vanes that were rotatable; and as such had about .020 in radial clearance with the housing when the vanes are in a completely closed position. Since the vane housing is basically conical in shape with an included angle $2\alpha = 30^\circ$, for any other position other than completely closed the clearance becomes bigger than the nominal mentioned above. The total leakage area is actually greater through this.
clearance than through the center hole at the inner tips of the vanes. However, the numerical model did not include this clearance although it can definitely be handled.

Of the various discretization schemes available, the “Linear Profile Skew” with advection correction was used in this analysis, which is second order accurate. Although, less robustness was expected in favor of accuracy, this calculation proceeded without any problem. The inlet total pressure and temperature was fixed at atmospheric conditions. The discharge average static pressure was also fixed and the calculation was carried on. The solution process was repeated by changing the exit boundary condition each time until the resulting flow rate matched that of the measurement. Actually, the exit was extended beyond the measurement plane and the boundary condition there was set such that it allowed flow both in and out of the domain. This was done to minimize the local influence of the boundary condition on the calculated result.

RESULTS

Figures 2, 3 and 4 are plots of both calculated and measured data of flow angle, axial velocity and tangential velocity at the traverse plane and along a traverse which is at an arbitrary angle. Particularly, Figure 2 is a plot of the gas angle distribution. This angle determines the incidence angle in a compressor. The solid line represents the calculated gas angle and the stars are the measured angles. The agreement is truly very good except for one measurement point near the far wall. I believe this difference has to do with the leakage through the clearance that was not modeled. The straight line is the predicted mass averaged gas angle which is different from its measured counterpart by only 2°. The calculated angle plot shows some jaggedness which corresponds to the wakes and jets behind the pre-rotation vanes which have not mixed out yet.

Figure 3 is a plot of both calculated and measured axial velocity. Once again the agreement is excellent. The jet at the center is not observed in the measurement data. In fact, when similar data is analyzed in a plane only 0.2in down stream from this plane, the jet has totally disappeared and the trend that looks like a “volcano’s mouth” in this plot becomes more dominant which is consistent with the measurement.

Figure 4 is a plot of tangential velocity. The stars once again represent the measured data. What is interesting about this plot is that both experiment and calculation conclude that the flow is in solid body rotation at the center and constant angle rotation at the outer radius.

In general, we found the results of this set of calculations extremely encouraging, showing the merits of CFD in complex flow analysis.

REFERENCES

Low Velocity Air Test Facility

Figure 1.

GAS ANGLE ALONG TRAVERSE

Figure 2.
AXIAL VELOCITY ALONG TRAVERSE

Figure 3

CALCULATED
* MEASURED

Figure 4

CALCULATED
* MEASURED