

**OPERATION OF CENTRIFUGAL COMPRESSORS IN CHOKE CONDITIONS**

Rainer Kurz  
Manager, Systems Analysis  
Solar Turbines Incorporated  
San Diego, CA , USA

Russell K. Marechale  
Aerodynamic Specialist  
Solar Turbines Incorporated  
San Diego, CA , USA

Edward J. Fowler  
Consulting Engineer  
Solar Turbines Incorporated  
San Diego, CA , USA

Min Ji  
Principal Engineer  
Solar Turbines Incorporated  
San Diego, CA , USA

Michael J. Cave  
Group Manager, Gas Compressor Aero Performance  
Solar Turbines Incorporated  
San Diego, CA, USA

**ABSTRACT**

Centrifugal compressors are at times required to operate in or near the choke region. Various limits of the degree of allowable operation in choke have been established. Based on test data and numerical data, the behavior of centrifugal compressors in the choke region is studied. Changes in aerodynamic performance, thrust load, volute behavior and radial loading are considered. The issue of excitation of impeller vanes is addressed. Particular consideration is given to multistage machines, as well as dual compartment machines, in particular regarding the effects of impeller mismatch during operating conditions at flows significantly higher than the design flow.

Limitations in the overload operating range of a compressor not only impact the operational flexibility , but also can require more complicated control systems. The paper addresses aerodynamic, structural as well as rotordynamic issues related to the operation in choke.

**INTRODUCTION**

Centrifugal compressors are frequently used in the oil and gas industry in higher pressure applications (Rasmussen and Kurz, 2009). One of the characteristics of applications in the oil and gas industry is the requirement to use the compressors over a wide range of operating conditions (Figure 1). The machines, usually speed controlled, are expected to operate from low flows near surge to very high flows near or in choke. While the operation near surge, as well as issues to prevent compressors from surging have drawn significant amounts of attention (Moore et al, 2009). The potentially damaging effects of surging compressors are widely acknowledged.

Operating compressors in the high flow region , i.e. at higher flows than at the best efficiency point, often referred to as overload , choke or stonewall, has been identified by many manufacturers as a region of operation that needs to avoided, also (Brun and Kurz, 2007) .Many compressor manufacturers will limit the operation of the compressors at high flows) by

defining a ‘choke limit’ or ‘sonic limit’ on a compressor map, prohibiting the operation of the compressor at low pressure ratios. This can require installation of additional throttle valves, or more complicated recycle and anti-surge valve selections, and thus impacts the cost of operation.

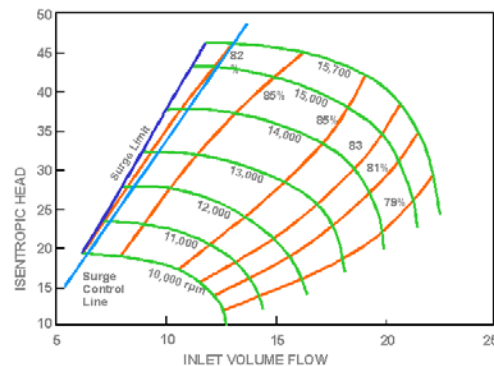


Figure 1: Typical Map of a Speed Controlled Centrifugal Compressor

In the literature, we find only a few papers (Sorokes, Miller and Koch, 2006, and Borer, Sorokes, McMahon, and Abraham, 1997, Brun and Kurz, 2007) discussing the issue of compressor operation in choke.

Borer et al (1997) discuss the impact of aerodynamic forces on impeller vanes. They point out that downstream flow non-uniformities, as they can be created by vaned diffusers or discharge volutes, can create fluctuating stresses in blade leading edges. Especially in high pressure compressors, the forces acting on blade leading edges can be quite large. Sorokes et al (2006) also focus on the risk of damaging the impeller leading edge or inlet region if flow fluctuations in combination with off-optimum incidence create dynamic forces that exceed the allowable stresses. In particular they point at the interaction between non uniformities created by compressor

components such as diffuser vanes, volutes, guide vanes or return vanes, and the flow field at the impeller inlet.

The present paper will focus in particular on the impact of axial thrust loading of the compressor.

**OPERATION IN CHOKE**

Unfortunately, the setting of choke limits is somewhat arbitrary in many instances, and may be subject to questions, in particular if it limits operational flexibility. To be very clear: Prolonged operation in the choke region should be avoided, because if nothing else, the efficiency of the compressor is very low. Whether the compressor operates in choke or not is not as clearly defined, and not as obvious from an operational standpoint as the operation of a compressor in surge. A compressor map showing a single speed line will usually show a more or less steep drop in head and efficiency when the compressor is operated at flows higher than BEP. This behavior is more pronounced at Mach numbers approaching 1, and high mole weight gases (Figure 2).

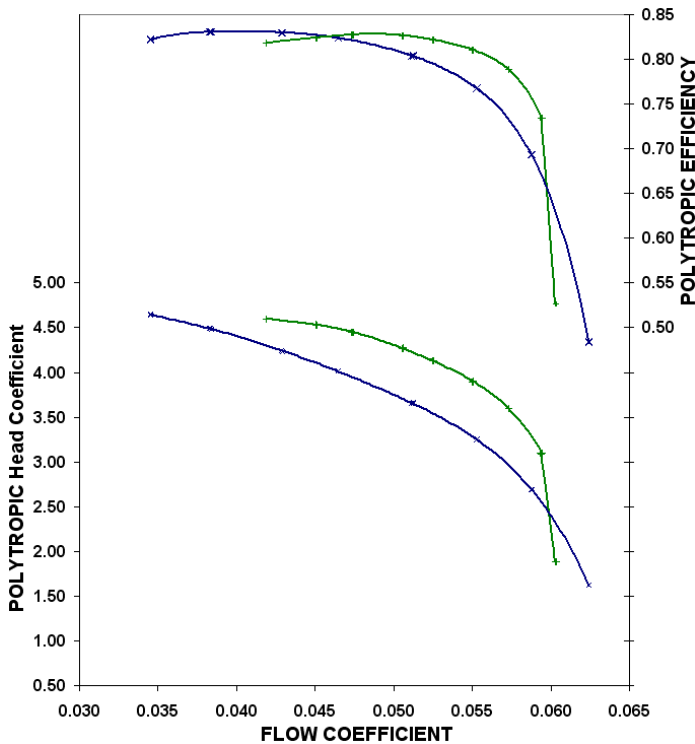


Figure 2: Stage map for  $M_n=0.56$  and  $M_n=0.76$

From an aerodynamic standpoint, choke refers to a situation where flow passages become blocked either due to the occurrence of compression shocks or due to massive flow separation. In centrifugal compressors, these flow passages can either be the impeller inlet, or the inlet to diffuser vanes. Both operation near the surge limit and in choke leads to flow conditions that are severely different from the flow conditions at the compressors design point. If one uses the airfoil of an aircraft as an analogy, surge and choke would mark the stall points of the airfoil at very high positive flow incidence (i.e. high angles of attack) and at a very high negative flow incidence.

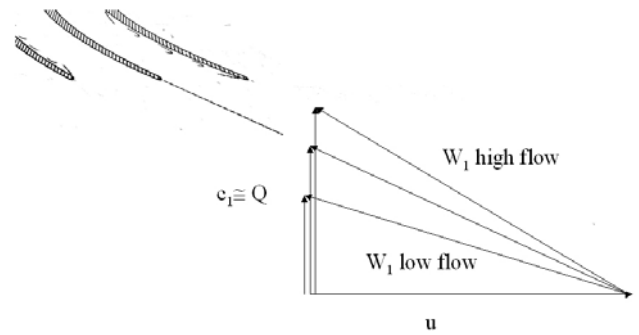


Figure 3: Inlet velocity triangles at different incidence

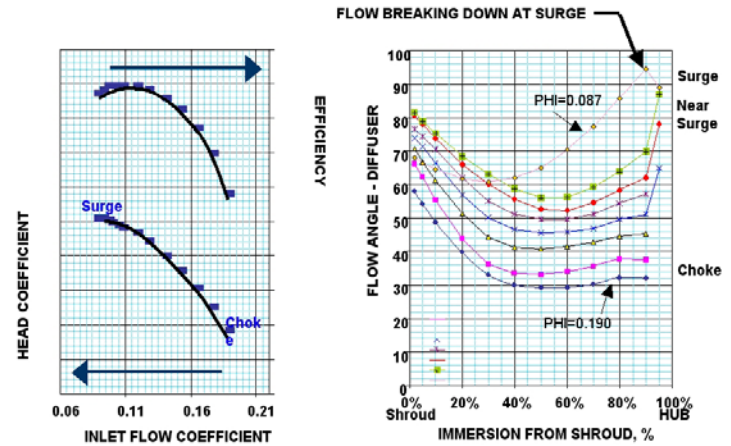


Figure 4: Flow in a vaneless diffuser from surge to choke

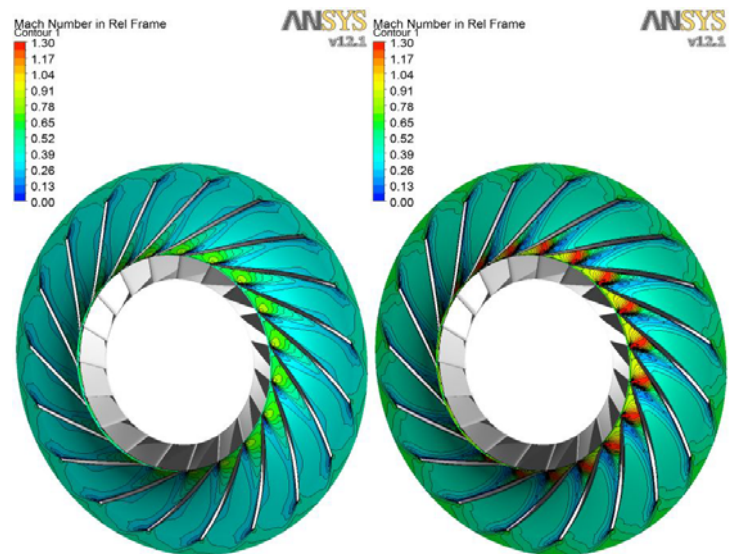


Figure 5: Mach number contours (relative frame) for impeller operating at  $M_n=0.56$  (left) and  $M_n=0.76$  and slightly lower flow coefficient (right). At the lower Mach number, stall due to negative incidence has developed. At the higher Mach number, shock has formed at the pressure surface. Refer to Fig. 2.

In the compressor, choke is related to a flow regime at very high flows which means that the flow channels between blade rows may experience blockage effects, either from sonic flow shocks, wake areas, strong secondary flows, or simply by the fact that the disturbed flow uses the through flow area less

efficiently. In reality, one often sees a combination of all of these.

If the compressor impeller is operated at its design point, the flow will enter the impeller in the optimum direction. ( Figure 3). If the flow is reduced, the direction of the flow into the impeller changes, increasing the losses, and ultimately leading to flow separations on the suction side of the impeller vanes. This is often referred to as stall. If the flow is increased starting at the design point, the losses are also increased, but so are the velocity levels entering the impeller. Eventually, the combination of flow separation (this time on the pressure side of the impeller), and the occurrence of compression shock waves will essentially limit the flow that can pass through the impeller (Figures 2 and 5). The shock waves themselves tend to be dynamic and fluctuating in nature. The reality is a little more complicated, since the flow field is three dimensional. Therefore, compression shocks may only block part of the flow path (typically closer to the shroud side), and separated flow may form rather complex flow structures.

Vaneless diffusers offer no particular challenge for operations in choke (Figure 4). In particular, no issues with compression shocks, or a particular danger of flow separation has to be considered.

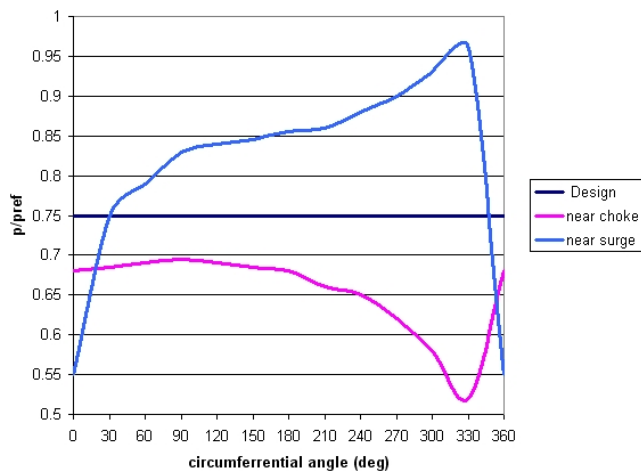


Figure 6: Circumferential Pressure Profile of a Volute (Fiedler, 1989)

Volutes, on the other hand show a distinct change in the circumferential pressure distribution when operated away from the design point (Figure 6), which has to be considered when the radial forces on the rotor are to be assessed.

While operation at high flow is unattractive to the user due to the associated drop in efficiency, there are operational situations where it can be encountered:

- During start-up when the recycle valve is opened too much.
- Process upsets, for example if two compressors operate in parallel, and one of them has to be shut down (Ohanian et al, 2002)

-Compressor undersized for the desired operating conditions.

-Performance degradation due to fouling

## OFF –DESIGN AERODYNAMICS

In multistage compressors impellers are in general selected such, that at the design point, all impellers operate at or near their best efficiency point (BEP) (or at about the same individual impeller surge margin. Surge margin is defined for a compressor operating at some flow  $Q$ , relative to the flow at surge  $Q_s$ , for a constant speed:

$$SM = \frac{Q - Q_s}{Q} \Big|_{N=const} \quad (1)$$

This approach tends to yield a good efficiency at the design point as well as a wide operating range. When the compressor is operated away from the design point, for example at higher flows than BEP, the first impeller will create less pressure ratio, and therefore achieve less volume reduction than before (this can be seen, for example in Figure 5a). Therefore, the second and subsequent impellers will increase their respective surge margin at a faster rate than the first impeller. In other words, the rear impellers will in general get to choke earlier than the impellers further in front (Figure 7). This means in particular, that in situations where the compressor appears to be near choke, one or some of its stages may already be in deep choke.

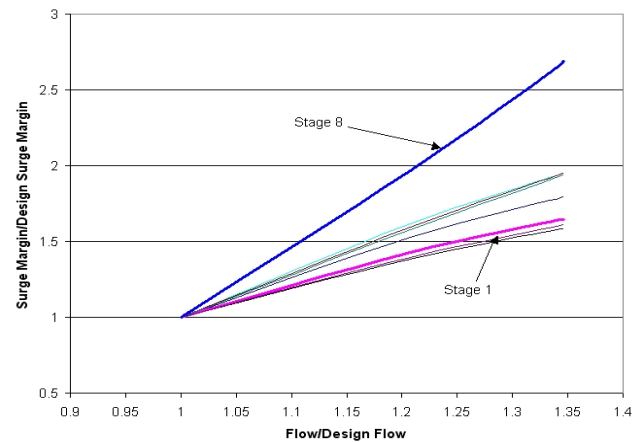


Figure 7: Shift in relative impeller operating points at increased flow for an 8 stage centrifugal compressor.

## THRUST LOADS

Thrust loads of centrifugal impellers are a result of a pressure imbalance between the front face and the rear face of the impeller. The sum of these forces over all impellers and the forces created by the balance piston are the resulting load on the compressor thrust bearing.

From the axial momentum equation, which takes into account the change of the axial momentum of the gas, and the forces due to the static gas pressure in axial direction:

$$\oiint \rho \vec{C} (\vec{C} \cdot d\vec{A}) = \oint p \cdot d\vec{A} + \vec{F} \quad (2)$$

we get the resulting forces on the impeller as (Figure 8):

$$\vec{F}_{impeller} = \vec{F}_{momentum}(c_{exit}, c_{inlet}) - \vec{F}_{pressure}(p_{cavity,front}, p_{cavity,rear}, p_{inlet}, p_{exit}) \quad (3)$$

The front and rear cavities are formed between the impeller tip and the labyrinth seals at the impeller inlet, and the impeller hub seals, respectively.

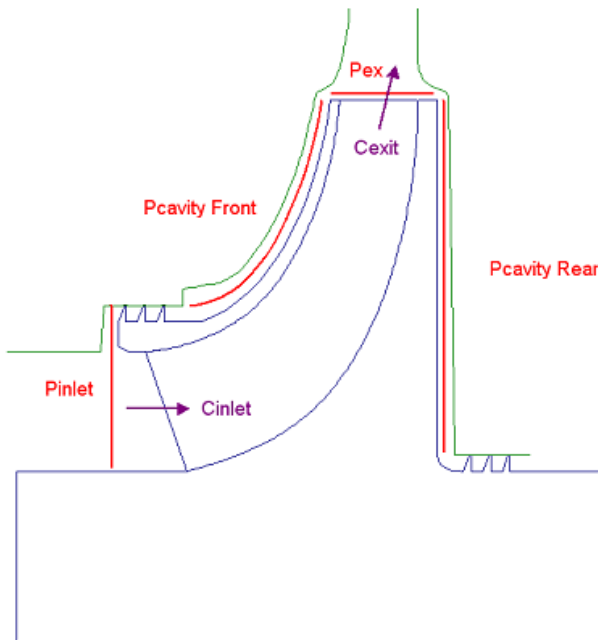


Figure 8: Forces on the impeller

The force on the thrust bearing is thus (Figure 9):

$$\vec{F}_{thrustbearing} = \vec{F}_{impeller} - \vec{F}_{balancepiston}(p_{discharge}, p_{suction}) \quad (4)$$

In the simplest approach to calculate the forces on the impeller, one would assume the pressure in the front and rear cavities to be equal to the pressure at the impeller tip. In a shrouded impeller however, the gas in the cavity is subject to swirl, and as a result, the static pressure at lower radii is lower than at the tip. The amount of swirl is a function of the cavity geometry, and the leakage flows through the labyrinths.

The cavity static pressure distribution can be calculated by:

$$p(r) = p_{tip} - \frac{1}{2} \rho (q \omega)^2 (r_{tip}^2 - r^2) \quad (5)$$

accounting for the cavity characteristics by introducing a cavity swirl coefficient  $q$ .

A simple approach would assume constant swirl coefficients for front and rear cavities. This approach is frequently used in the industry, but high pressure compressors require more accurate estimates. Correlations and CFD analysis (Figure 10 a) are being used for these, along with subscale test measurements for validation.

Of particular importance for the topic of off design operation is the fact that the swirl coefficient changes when the impeller is operated away from its design point (Figure 10 b). Also, the magnitude of the swirl coefficient on the impeller backside changes in the opposite direction from the swirl coefficient on the front side of the impeller. This means that the thrust imbalance (for a given pressure level and a given speed) changes not just due to the pressure difference between the impeller eye and the corresponding backside, but also due to different swirl factors in the cavities in the front and back of the impeller. This imbalance, in particular, changes when the compressor moves from the design point to choke. In general, the shroud side swirl is higher than the backside swirl, a result also reported by Koenig et al, 2009.

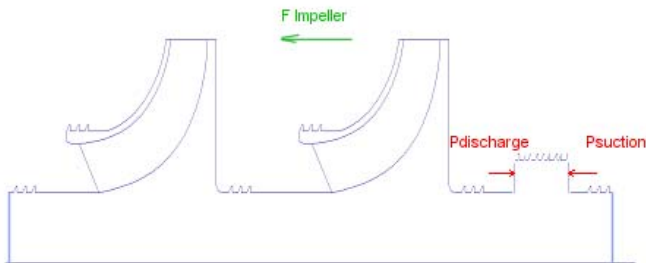


Figure 9: Balance Piston

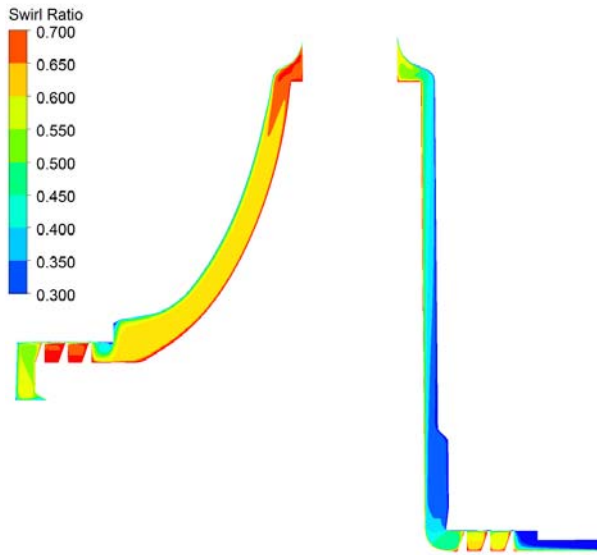


Figure 10 a: Swirl ratio in the shroud and the backside cavity

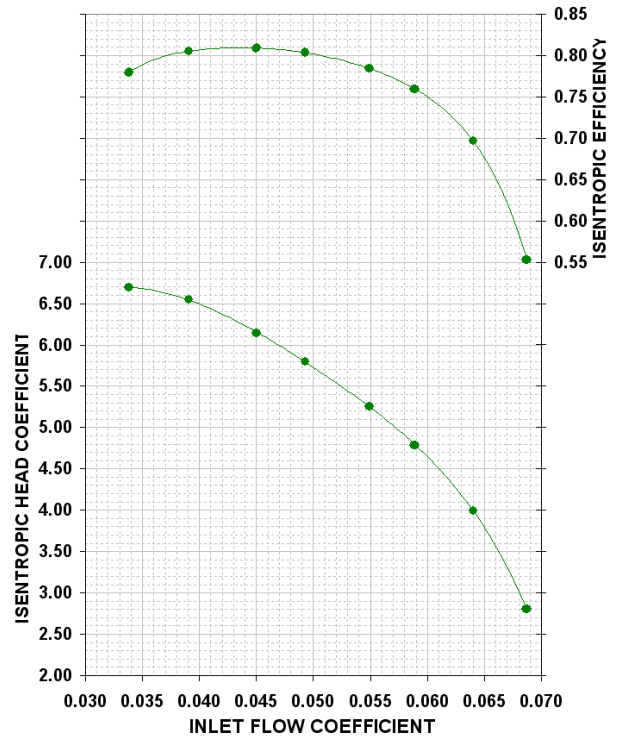


Figure 11 a: Non dimensional map for multistage compressor

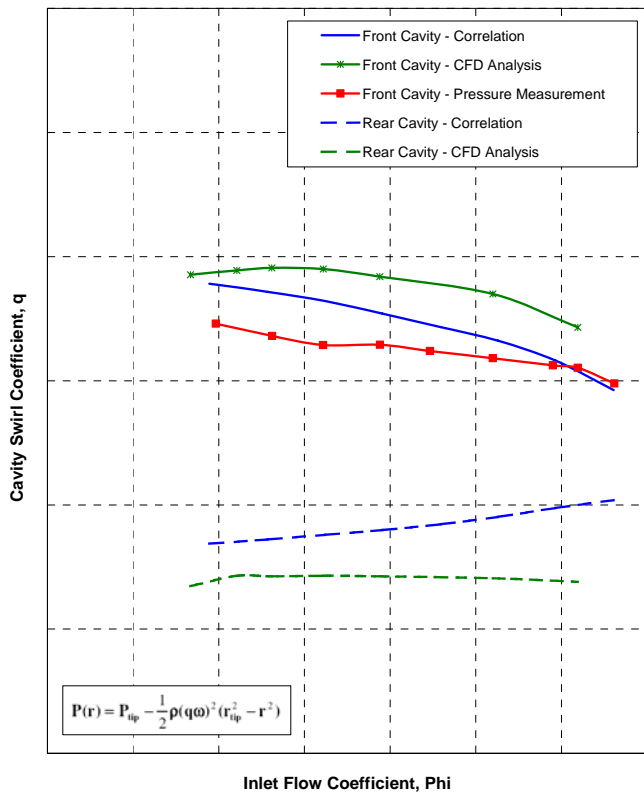


Figure 10b: Cavity Swirl Coefficient for a Medium Flow Stage for different operating points

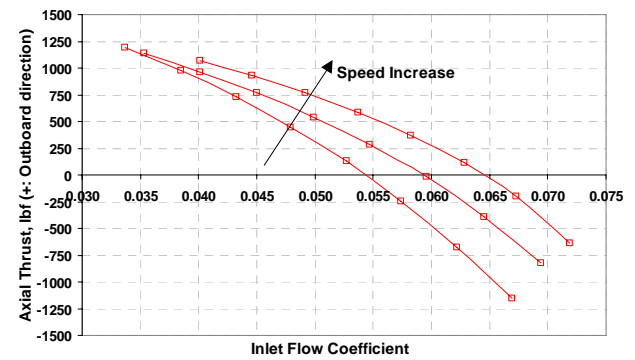


Figure 11 b: Change of axial thrust with operating point

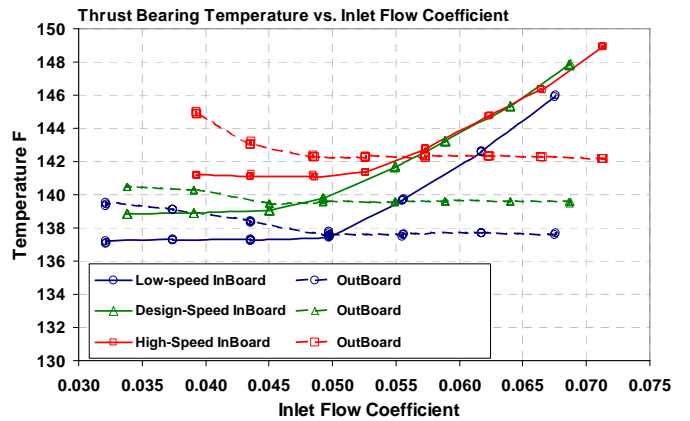


Figure 11 c: Thrust bearing temperature as a function of operating point



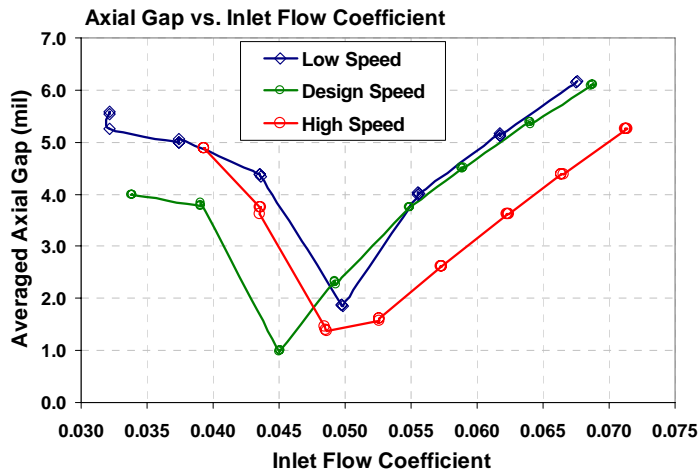


Figure 11 d: Axial position of the rotor as a function of the operating point

Because the thrust load has a direct impact of the thrust bearing temperature, which can be measured conveniently, Figures 11 a –d establish the correlation between non dimensional operating point (Figure 11a), thrust load at different speeds (Figure 11 b), the resulting bearing temperature of the loaded and unloaded pads of the thrust bearing (Figure 11 c), as well as the axial position of the rotor as a result (Figure 11 d). The inboard bearing shows a significant increase in temperature (albeit not to a level that would cause concern) when the compressor enters the choke region. The outboard bearing shows only a much lower increase in temperature when the operating point moves towards surge. For this particular application, with the particular selection of the balance piston size, the thrust load reverses direction, which explains the behavior of the in board and out board bearing temperature. Of course, the bearing temperature increases also with speed. As a result of the thrust load changes and the changing load capacity of the thrust bearing with speed, the axial gaps for all speeds are fairly close together, but change significantly when the compressor is operated from design point to surge or into choke.

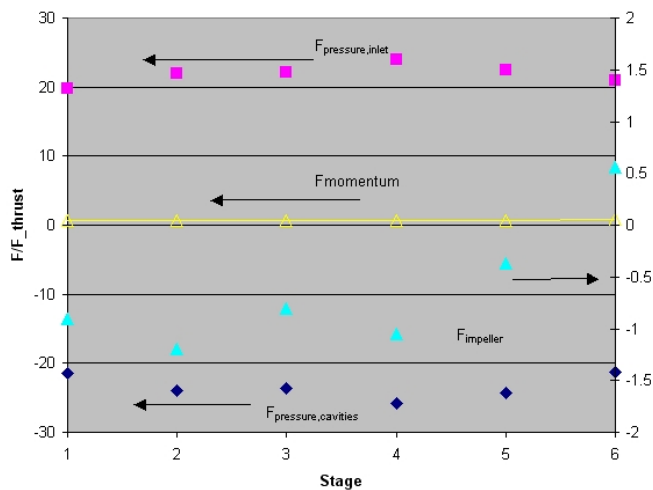


Figure 12: Contributing Forces to the Impeller Thrust in a 6 Stage Compressor.

If we compare the magnitude of the forces acting on the impeller (Figure 12), the pressure from the inlet eye and the pressures in the cavities are usually dominant, but act in opposite direction. In general, they generate a resulting force, much smaller than the pressure forces, in direction of the compressor inlet, but as can be seen in Figure 12, this is not always the case. The momentum force, generated by deflecting the gas form more or less axial to more or less radial direction, is usually much smaller than the pressure forces. At very high discharge pressures near choke, when the pressure differential over the impeller is rather small, the momentum force can become dominant, and create a net force towards the discharge end of the compressor.

The descriptions in this section are based on shrouded impellers. As opposed to open faced impellers, that have free standing blades, the blades in shrouded impellers are covered. Therefore, the pressure distribution on the front face of the impeller is governed by the impeller discharge pressure and the impact of swirl flow. In an open faced impeller, the pressure distribution would be governed by the pressure build up in the impeller flow passages.

### MULTISTAGE MACHINES AND MACHINES WITH MULTIPLE SECTIONS

Another issue that has to be considered with multistage compressors operating near choke: The overall compressor may still produce head, when some individual stages are already reducing head, thus, acting as throttles. These impellers will consequently see a lower pressure on their discharge side than on their suction side, which can in some instances alter the axial thrust balance in the compressor, leading to increased load on the thrust bearings.

The mechanical design of compressors with multiple sections regarding the arrangement of impellers can either be a straight trough design, or a back to back design. In a back to back design, with the impellers in the first section facing in opposite direction of impellers in the second section, most of the axial thrust balance is accomplished by the impellers themselves. Usually, a relatively small balance piston can handle the axial thrust. For all applications, the axial thrust has to be determined for all operating conditions. Two section machines are particularly critical in this respect if they have to handle multiple streams that change flow independently of each other. Thrust limitations can create limits for the unbalance between the sections.

In highly transient conditions, such as emergency shutdowns (Moore et al,2009), this has to be considered. In some instances, hot gas recycle valves have to be employed to equalize the pressure between the sections before the thrust imbalance can cause damage.

In the previous considerations, the impact of seal leakage flows on impeller and balance piston thrust forces has been mentioned. Impeller seals, as well as balance piston seal clearances can increase over time, although designs where the labyrinth teeth are facing abrasable material are less prone to this problem. Nevertheless the fact remains that increased seal clearances can change the trust balance. However, monitoring

bearing temperatures seems a good way of protecting the compressor from thrust load issues, especially when operating at extreme points of the map.

## IMPELLER VANES

Borer et al (1997) and Sorokes et al(2006) have specifically discussed in more detail the issue of damage to the impeller vanes due to high cycle fatigue. With respect to the operation in choke, the frequency of the exiting forces on the impeller vanes can be captured in a Campbell diagram (an example is given in Figure 13). Comparing the exiting frequencies (in this case the vane passing frequency from the inlet vane) with natural frequencies of the impeller and its vanes reveals the separation margin between excitation and response. If the separation margin is deemed too small, impeller modifications such as increases in vane thickness can reduce stress levels and modify impeller natural frequencies. Figure 14 shows the result of such a redesign, where it was possible to achieve the stress reduction and a larger separation margin, while retaining the impeller performance.

It should be noted that in some axial compressors, but also in centrifugal compressors with free standing blades, a phenomenon called choke flutter can be observed when they are operated in the choke region (Fottner, 1989). The mechanism is different from the mechanism mentioned above, as it is the flow around the blade that causes the excitation. However, the frequencies that are excited are, just as in the case of disturbances in the inlet or exit flow, the blade natural frequencies. The aerodynamic excitation of the blade is usually caused by boundary layer separations, transonic shock patterns, and the related vortex frequencies. This problem has, to our knowledge, not been encountered in centrifugal compressors with shrouded impellers.

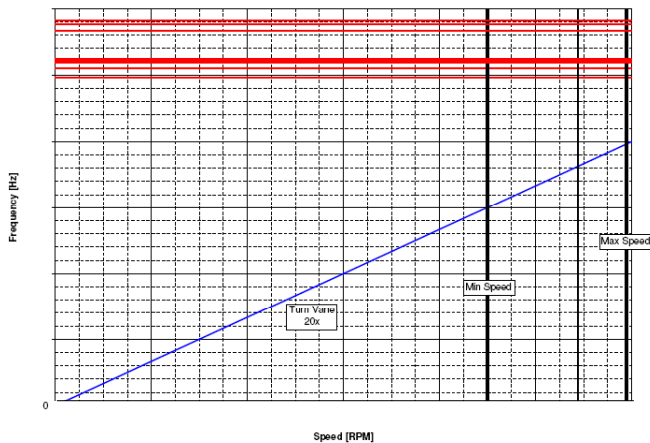


Figure 13: Campbell Diagram for an Impeller.

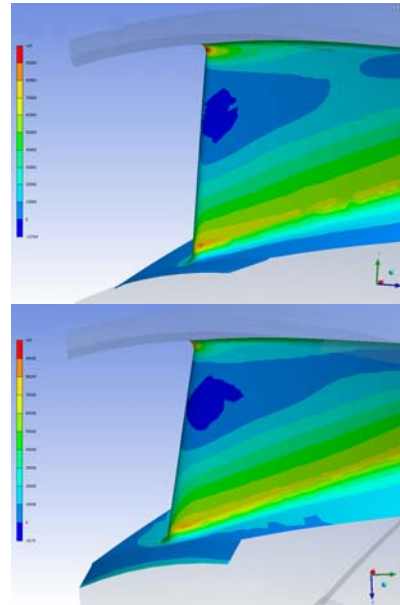


Figure 14: Principal stress distribution near the leading edge of an impeller vane. The right picture shows the improved version of the same impeller with reduced stress in the hub and shroud region.

## CONCLUSIONS

Besides the performance penalties, the study indicates that operating in choke is often not a problem for the compressor provided:

- The balance piston is sized correctly to provide adequate thrust load balance over the entire operating range
- Issues like blade strength to deal with alternating stresses are considered, or the occurrence or strength of alternating stresses is reduced.

## NOMENCLATURE

$\rho$  = Density

$p$  = Pressure

$\vec{C}$  = Velocity

$\bar{A}$  = Area

$\vec{F}$  = Force

$Q$  = Flow

$M_n$  = Machine Mach Number

$SM$  = Surge Margin

$r = \text{radius}$

$p = \text{pressure}$

$tip = \text{impeller tip}$

$q = \text{cavity swirl coefficient}$

$\rho = \text{density}$

$\omega = \text{impeller rotational speed}$

## REFERENCES

Borer, C., Sorokes, J., McMahon, T., and Abraham, E., 1997, 'An Assessment of the Forces Acting Upon a Centrifugal Impeller using Full Load, Full Pressure Hydrocarbon Testing, 26<sup>th</sup> Turbomachinery Symposium, Houston, Tx.

Brun, K., Kurz, R., 2007, 'My Compressor can (cannot) run in Choke or Stonewall', Turbomachinery International, Nov/Dec 2007.

Fiedler, K., Turbinen und Turboverdichter, Universitaet der Bundeswehr Hamburg, 1989.

Fottner, L., 1989, 'Review of Turbomachinery Blading Design Problems, AGARD LS 167, Toronto, Canada.

Koenig, S., Petry, N., Wagner, N.G., 2009, 'Aeroacoustic Phenomena in High pressure Centrifugal Compressors – A possible Root Cause for Impeller Failures' 38<sup>th</sup> Turbomachinery Symposium, Houston, Tx.

Moore, J.J., Garcia-Hernandez, A., Blieske, M., Kurz, R., Brun, K., 2009, 'Transient Surge Measurements of a Centrifugal Compressor Station during Emergency Shutdowns, 38<sup>th</sup> Turbomachinery Symposium, Houston, Tx.

Ohanian, S., Kurz, R., 2002, 'Series or parallel Arrangement in an Two-Unit Compressor Station', ASME J Eng for GT and Power Vol. 124, pp936-941

Rasmussen, P.C., Kurz, R., 2009, 'Centrifugal Compressor Applications-Upstream and Midstream', 38<sup>th</sup> Turbomachinery Symposium, Houston, Tx.

Sorokes, J., Miller, H., Koch, J., 2006, 'The Consequences of Compressor Operation in Overload', 35<sup>th</sup> Turbomachinery Symposium, Houston, Tx.



**MODELING AND PREDICTION OF SIDESTREAM INLET PRESSURE FOR MULTISTAGE CENTRIFUGAL COMPRESSORS****Jay Koch**

Principal Engineer  
Dresser-Rand  
Olean, NY, USA

**Moulay Belhassan**

Principal Engineer  
Dresser-Rand  
Olean, NY, USA

**Jim Sorokes**

Principal Engineer  
Dresser-Rand  
Olean, NY, USA

**ABSTRACT**

It is common for some compressors in certain applications to have one or more incoming sidestreams that introduce flow other than at the main inlet to mix with the core flow. In most cases, the pressure levels at these sidestreams must be accurately predicted to meet contractual performance guarantees. The focus of this paper is the prediction of sidestream flange pressure when the return channel outlet conditions are provided. A model to predict the impact of local curvature in the mixing section is presented and compared with both Computational Fluid Dynamics work and measured test data.

**INTRODUCTION**

Despite recent advances in analytical tools, developers and users of state-of-the-art centrifugal compressor equipment continue to rely heavily on testing to ultimately confirm the performance of new components or stages. This is especially true in heavy hydrocarbon applications such as compressors used in liquefied natural gas (LNG), ethylene, or gas-to-liquid facilities. Heavy hydrocarbon gases have very low gas sonic velocities that produce high Mach numbers in the aerodynamic flowpath. By their nature, such high Mach number, high flow coefficient stages have very narrow flow maps characterized by limited choke and surge margin. In addition, many of these applications require the machines with sidestreams (or sideloads) to accommodate the flows entering and/or exiting the compressor as determined by the particular process for which they are intended. The sidestreams and associated mixing further complicate the performance prediction process because the pressure, temperature and flow conditions at each one of these sidestreams as well as at the exit of the machine must be met within stringent tolerances to optimize the amount of end product delivered by the process. Therefore, it is imperative that the compressor OEM and end user have a firm

grasp of the performance characteristics of any impellers, diffusers, return channels, and/or other flow path components used in such equipment.

The high cost associated with delays in the project schedule increase the attractiveness of design and testing methods that ensure these machines meet the contractual operating requirements the first time; without the need for repeated modifications and test iterations to correct performance shortfalls. Given the size, construction style, and in-house test piping arrangements for many large centrifugals, one modification / re-test iteration could take several weeks. Repeated iterations on the OEM's test facility could delay the start-up of a new plant by months, leading to lost production / revenue for the end user as well as lost revenue and reputation for the compressor OEM. Consequently, OEMs spend continually work to enhance their performance prediction methods. This paper details work done toward that end.

This paper is the latest in a continuing series of publications describing work completed in association with a novel test vehicle that was designed to replicate the performance of a full-scale, large frame-size, multi-stage centrifugal compressor for high flow coefficient, high Mach number stages. The test vehicle, described by Sorokes et al (2009), is equipped with a vast array of internal instrumentation that make it possible to evaluate the aero/thermodynamic and mechanical behavior of a compressor. Sufficient instrumentation is installed to allow assessment of the entire compressor as well as individual components or combinations of components. Of particular interest to this study, the test rig included instrumentation at critical locations within the sidestream components to permit an assessment of the losses through each sidestream element.

**BACKGROUND**

As part of the compressor selection and design process, the OEM must be able to predict the performance of the overall