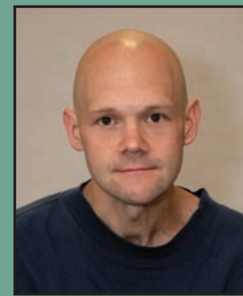
ENGINEER'S notebook

James M. Sorokes



Jason A. Kopko

Paul R. Geise

Angelina L. Hinklein

THE INFLUENCE OF SHROUD CURVATURE AND OTHER RELATED FACTORS ON IMPELLER PERFORMANCE CHARACTERISTICS

Published by ASME and presented at the ASME Turbo Expo, Orlando, Florida, 2009.

James M. Sorokes
Principal Engineer
Dresser-Rand
Olean, New York, USA

Jason A. Kopko
Aero/Thermo Design Engineer
Dresser-Rand
Olean, New York, USA

Paul R. Geise
Solid Mechanics Engineer
Dresser-Rand
Olean, New York, USA

Angelina L. Hinklein
Aero/Thermo Design Engineer
Dresser-Rand
Olean, New York, USA

INTRODUCTION

Given the increasing costs for energy, process compressor users have demanded higher performance from centrifugal compressor OEMs. Nowhere is this truer than in the large-scale equipment used for liquefied natural gas, gas to liquid, ethylene or the like. These large trains often press the power limits of very high horsepower turbines. Therefore, it is imperative that the OEMs provide the maximum achievable efficiency in their aerodynamic designs.

OEMs have expended significant effort to improve the aerodynamic design of the stages required. Many of these efforts focused on centrifugal impellers as they represent the most critical element of a centrifugal stage. There is much in the open literature regarding the improvements derived via arbitrary blade shapes, splitter-bladed impellers and the improvements obtained via advanced manufacturing techniques such as 5-axis point milling, *etc.* Others researchers have focused on the stationary components such as diffusers (vaned or vaneless), return channel, inlets, volutes, and sidestreams. A tremendous amount of work has been done on alternate vaned diffuser concepts, most notably low solidity vaned diffusers. Still, it is universally recognized that superior stage efficiency must begin with the impeller. No amount of "tweaking" of stationary components can overcome a poorly performing impeller.

Adding to the complexity, the heavy mole weight applications also require high flow coefficient impellers that operate at high inlet relative Mach numbers. Further, the compressors are typically multi-stage arrangements, having as many as 6 or 7 stages per casing. Therefore, rotordynamics can become an issue if the staging is excessively long. Even further, high flow coefficient impellers normally utilize highly three-dimensional blading that can result in high stress levels that limit the allowable operating speeds.

The design process for high inlet relative Mach number impellers is well documented; *i.e.*, Sorokes *et al* [2007]; and it is understood that shroud curvature has a significant impact on the inlet relative Mach number.

The purpose of the study described in this paper was to investigate the impact of shroud curvature on the performance of a centrifugal impeller or stage. While there might seem to be an easy solution – simply increasing the curvature radius or radii along the cover of the impeller – this poses a serious problem to designers in the process centrifugal compressor industry. Increasing the cover radii yields a low inlet relative Mach number and improves performance. However, it also results in increased impeller length and increased bearing span for multi-stage compressors. If the bearing span becomes excessive, rotordynamic issues could result. Therefore, it is crucial to find or to understand the compromise between aerodynamic and rotordynamic performance.

The paper investigates the aerodynamic, rotordynamic, and mechanical compromises

ABSTRACT

The paper discusses a computational fluid dynamics (CFD) study done to assess the influence of cover or shroud curvature on impeller performance. The paper describes the various designs and the CFD and finite element analyses (FEA) methods used. Aerodynamic and mechanical analysis results are presented for four impellers of varying cover curvature and axial length. Comments are offered regarding the mechanical issues that must be considered when increasing the length of impellers.

associated with varying the cover curvature and axial length of a centrifugal impeller. Computational fluid dynamics (CFD) has been shown to be very reliable in the design and analysis of new or existing aerodynamic component designs. There are a vast number of references available on the use of CFD for stage and/or component design (too numerous to list herein) including Cumpsty [1989], Raw *et al* [1989], Sorokes [1993], Casey [1994], and Japikse [1996].

CFD results are presented showing the achievable performance benefits. Stress analyses are also used to show the potential mechanical considerations. Finally, comments are offered on potential manufacturing concerns associated with the longer impellers.

DISCUSSION

This study was conducted as part of an overall initiative to develop new high performance staging for the OEM's products. The primary intent was to increase the stage efficiency and flow range for high flow coefficient impellers ($\Phi > 0.100$). The target flow coefficient for the subject impeller was 0.12. The impeller had to fit on a shaft that was 25% or more of the impeller diameter; *i.e.*, $D_{shaft}/D_2 \geq 0.25$. Obviously, the requirement to install the impeller on a shaft impacts the inlet sizing for the impeller. That is, the shroud diameter must be adjusted outward until the necessary or optimal inlet area is obtained.

The shaft diameter for the line of impellers was established to ensure good rotordynamic characteristics. Said rotordynamic characteristics, as noted earlier, are also dependent on the overall shaft length. The aerodynamic designer must be cognizant of limitations imposed by the shaft dynamics.

An existing impeller design was used as the baseline for this analytical effort. The existing impeller had an Ax/D_2 of 0.21. That is, the axial length of the bladed portion was 21% of the diameter of the impeller. This is a relatively short impeller by some standards but this impeller had provided very good aerodynamic and mechanical performance over a wide range of applications. However, the existing impeller was found to be unsuitable for the more demanding high mole weight applications under consideration. When the existing impeller was analyzed at the required machine Mach number (U2/A0) using computational fluid dynamics (CFD), some untoward flow anomalies were revealed as can be seen in

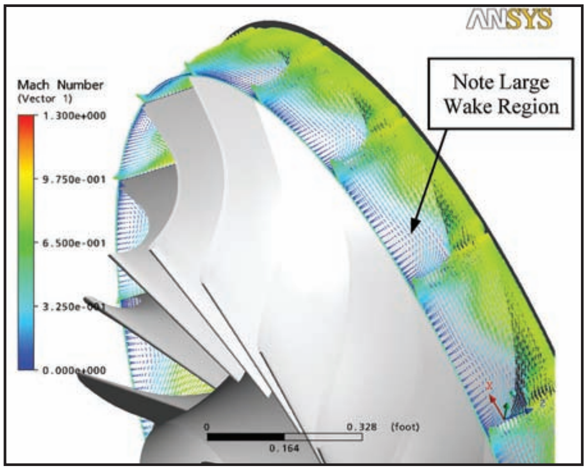


Figure 1 – Exit Flow Profile – Original Impeller.

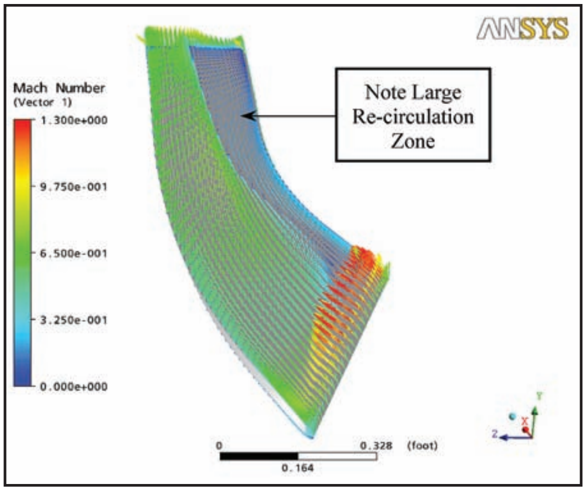


Figure 2 – Suction Surface Velocities – Original Impeller.

Continued on page 16

ENGINEER'S notebook

Figures 1 and 2. Note the large low-momentum and/or reversed flow region along the blade suction surface and along the shroud as well as the very high Mach number in Figure 2. The flow is separating from the suction surface downstream of the shock and high curvature. The large wake or secondary flow region is also evident when reviewing the relative Mach number distribution at the impeller exit plane (see Figure 1). An in-depth review of the analytical results pointed to rapid turning along the cover exacerbated by excess area at the impeller exit as the root cause of the problem. Such flow anomalies in CFD results are strong indicators that the impeller will experience premature stall, thereby limiting the usable operating range. In addition, the large wake region at the impeller exit will likely cause problems for the downstream diffuser, further reducing the stage performance. The OEM had prior experience on the benefits of reducing the tip dimension. However, it was unclear how much further improvement was possible by reducing the shroud curvature, especially in the leading edge region. Therefore, an effort was undertaken to determine the potential benefits of improving the shroud contour.

The curvature along the cover is driven by the need to turn the flow from an axial direction at the leading edge to a radially oriented flow at the exit. Curvature is defined as the inverse of the turning radius (see Figure 3). The length over which this turning takes place becomes a critical factor. The most effective way to reduce the curvature is to increase the distance over which the turning takes place; *i.e.* make the impeller longer. This was suggested in the works of Brammert *et al* [1980], Sapiro [1982] and Al-Zubaidy [1992].

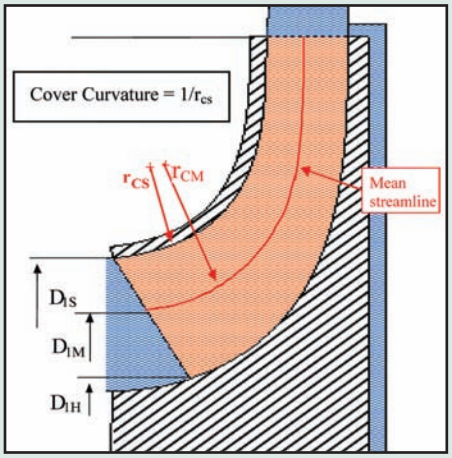


Figure 3 – Definition of Cover Curvature.

Figure 4 – Impeller Ax/D2 v. Flow Coefficient (Aungier – 2000).

The literature survey also uncovered some correlations used to specify the axial length for impellers as a fraction of the impeller exit diameter for various flow coefficients. One such correlation suggested by Aungier [2000] is shown in Figure 4. It is important to recognize that many of these correlations were developed for impellers used in turbo-chargers or other single stage, overhung arrangements. As such, these correlations tend to suggest impellers that are much longer axially than those applied in multi-stage centrifugal applications. Still, it was felt these correlations could serve as a reference for the multi-stage impeller design process.

The correlation shown in Figure 4 suggests that an Ax/D_2 of 0.27 should be optimal for an impeller with a flow coefficient of 0.12. This would yield an impeller with a 28.6% longer axial length than the original.

At the beginning of the study, it was unclear how much longer the impeller could be made before rotordynamic concerns would override the potential performance benefits. Therefore, an assessment was made using four different lengths: 0.21, 0.24, 0.27, and 0.30 Ax/D_2; *i.e.*, designs both shorter and longer than recommended by the criteria in Figure 4.

For all of the designs, the hub and shroud diameters at the front face of the various impellers were kept constant. The exit widths varied slightly to optimize the exit flow distribution but all designs fell within a tip width variation of 2.54mm (0.1") for a 686mm (27") diameter impeller or a maximum variation of approximately 5%. The blade angle distributions were altered to account for the increased axial length. The blade-to-blade loading (as assessed by Concepts ETI's AxCent streamline curvature code) were held to the same general limits by varying the number of blades in the impeller; *i.e.*, shroud loading less than 0.6 and hub loading less than 0.9. Herein, "blade loading" is defined as the suction surface relative velocity minus the pressure surface relative velocity divided by the midpitch relative velocity (or $\Delta W/W_A$). For reference, the maximum blade-to-blade loadings for the hub and shroud streamtubes are given in Table 1.

Parameter	Original	0.21 Ax/D	0.24 Ax/D	0.27 Ax/D	0.30 Ax/D
Max Load Shroud	0.85	0.50	0.52	0.49	0.49
Max Load Hub	0.67	0.62	0.57	0.53	0.69
Number Of Blades	19	19	17	17	15

Table 1 – Impeller Parameters.

As the length of the impeller was increased, it was possible to achieve acceptable loading levels with a lesser number of blades. Note that it was critical to maintain or decrease the blade loading (and other velocity distribution criteria) to ensure that the impeller met the required flow range, specifically the surge/stall margin. However, it was important to not install too many blades (*i.e.*, drive the loading even lower) as this would result in excess wetted surface and higher frictional losses. Therefore, the number of blades was selected to maintain a blade loading that had historically provided good flow range for the OEM.

CFD analyses were conducted to determine the relative merits of the various impeller configurations. Before discussing the results, a brief overview will be given of the CFD methodology used.

CFD MODEL / METHODS

The problem domain for all the CFD analyses consisted of a vaneless turning region upstream of the impeller, a rotating impeller, a vaneless diffuser, followed by a return bend and return channel incorporating de-swirl vanes. See Figure 5 for a representative picture of the problem domain. All domains were assumed to be axi-symmetric in nature, and as such only one bladed impeller sector and one bladed return channel sector were considered in all of the analyses.

All of the grids used in the analyses consisted of hexahedral elements. In all cases, the minimum face angle was greater than 20 degrees and the maximum face angle was less than 160 degrees.

Care was taken to ensure that the grid sizes between analyses would be consistent. The target impeller grid size was between 100k-150k nodes, while the target grid size for the de-swirl vanes was around 100k nodes. Total problem size ranged between 400k-500k nodes for all cases.

Independent of this study, a grid refinement study was performed on the original Ax/D = 0.21 geometry to study losses associated with near wall effects. Such effects are obviously important as the geometry is changed. Additionally, any regions of flow separation that might occur, as the grid was refined, were of definite interest. The grid refinement study covered grid densities from 75k nodes to 500k nodes per impeller passage. Briefly, the results of the grid refinement study showed no changes in global qualitative flowfield characteristics beyond 100k nodes per passage.

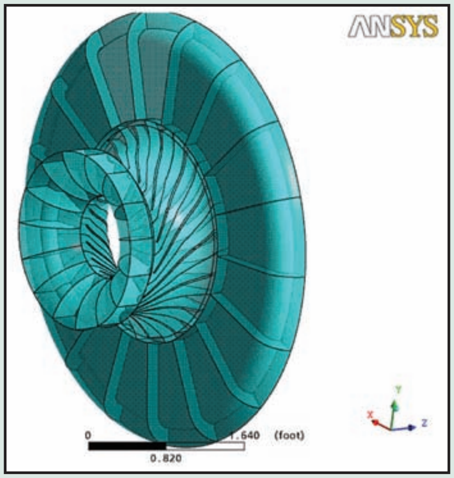


Figure 5 – CFD Domain.

Further, somewhat counter-intuitively, the quantitative performance levels (efficiency and head coefficient) actually increased as the grid was refined. This trend also seemed to contradict reality as it was recognized that the Ax/D = 0.21 offered several different opportunities for improvement in terms of aerodynamic design. Therefore, based on the results of the grid refinement study, it was decided to 'standardize' on a node count of between 100-150k nodes per impeller passage for the purposes of this investigation.

These grid densities are coarse by most standards but the OEM has found such grids to be more than adequate when conducting comparative analyses. That is, this study was done to assess the relative differences between the different impeller designs, not to determine with supreme accuracy the absolute magnitude of the performance parameters.

Because one cannot know exactly what geometry will be installed upstream of the new impellers, it is difficult to predict the turbulent intensity and length scales that will enter the impeller. Once again, when comparing geometries it is important to keep the parameters as consistent between analyses as possible in order to help ensure reliable results. As such, it was decided to use a turbulence intensity value of 5% and allow the CFD code to compute the turbulent length scale.

A frozen-rotor interface was used between the stationary and rotating components for all of the analyses. In each case, care was taken to try and ensure a good area match between rotating and stationary components at the grid interface plane. Further details of the frozen-rotor interface methodology can be found in the Ansys/CFX v11.0 documentation.

Continued on page 18

ENGINEER'S notebook

All CFD analyses employed air treated as an ideal gas for a working fluid. Total pressure of 100.0 psia and total temperature of 100^{0}F was specified at the inlet plane of the domain. The rotational speed of the impeller was set to achieve a machine Mach number of approximately 1.15 calculated in air using ideal gas assumptions. For simulations near the design point and to the left on the performance curve, a specified mass flow rate was used as an outlet boundary condition. However, for flow points near the choke point, the outlet boundary condition was switched to a specified static pressure at the exit plane, as this boundary condition often yields better results in the overload portion of the performance curve. The k-epsilon turbulence model with scalable wall functions was employed. The solutions were considered converged when the maximum values of all momentum and mass flow residuals dropped below a maximum value of 1.0e-4.

Once the aerodynamic analyses were completed, it was also necessary to assess the stress levels of the various impeller configurations. Clearly, it does little good to develop a more efficient impeller design if it cannot run mechanically. The follow section describes the mechanical analyses performed.

FEA MODEL / METHODS

The finite element impeller models used in this study were a one-blade pitch model (asymmetric model) and will be referred to here as sector models. The ANSYS finite element program was used to analyze all the impeller models. The models were created using solid brick elements (element type 45). The models ranged from 2277 to 2682 nodes and 1474 to 1802 elements. All models had a uniform blade mesh.

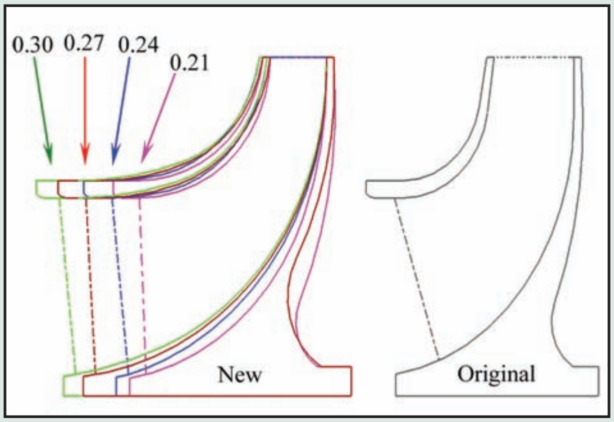


Figure 7 – Cross-Sections of New and Original Impellers.

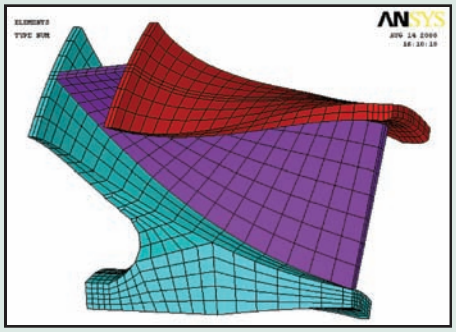


Figure 6 – Typical FEA Sector Grid.

As with the CFD analyses, these coarse meshes have been found to provide very satisfactory results when doing comparative analysis. Further, fillet radii and the like were not included in the analyses as these typically result in a fairly constant "offset" to the stress levels. Because all impellers were analyzed without the fillets, *etc.*, the relative changes in stress level are still relevant and sufficient.

All boundary nodes were rotated into the cylindrical coordinate system and coupled to their corresponding locations. This boundary condition simulates an exact 360-degree impeller model. An axial and tangential restraint was applied to the blade centerline node at the impeller inside diameter (impeller bore at the inlet). A rotational speed of 1100 feet per second was applied as the loading condition.

The analyses showed that as the axial length of the impeller increased the stress level of the impeller increased (based on the Von Mises nodal stress). Not all of the stress components followed that trend. Some individual stress components were relatively high when compared with the other designs. It depends on what structural criterion is being considered as the limiting factor. A yield-based criterion was the driving factor behind these designs. Therefore, the model with the lowest overall Von Mises nodal stress was being sought to determine the best impeller design.

Having discussed the aerodynamic and mechanical analysis procedures, the discussion will now turn to the results obtained for the four impellers.

ANALYTICAL RESULTS -- AERODYNAMICS

The meridional profiles for the four new and the baseline impellers are shown in Figure 7. The $Ax/D_2 = 0.30$ impeller is in green, while the $Ax/D_2 = 0.27$, 0.24, and 0.21 impellers are in red, blue, and magenta, respectively. Examination of this figure shows the effect of varying impeller length on shroud curvature, particularly in the critical leading edge region. The change in curvature along the cover and, in particular, near the leading edge is obvious but for clarity, the curvature distributions for the various impellers are shown in Figure 8. Note the significant reduction is curvature in the 0.30 Ax/D_2 design as compared to the 0.21 design and versus the original design. As the figures may be difficult to read, the curvatures at the leading edge are 0.03, 0.04, 0.06, 0.16,

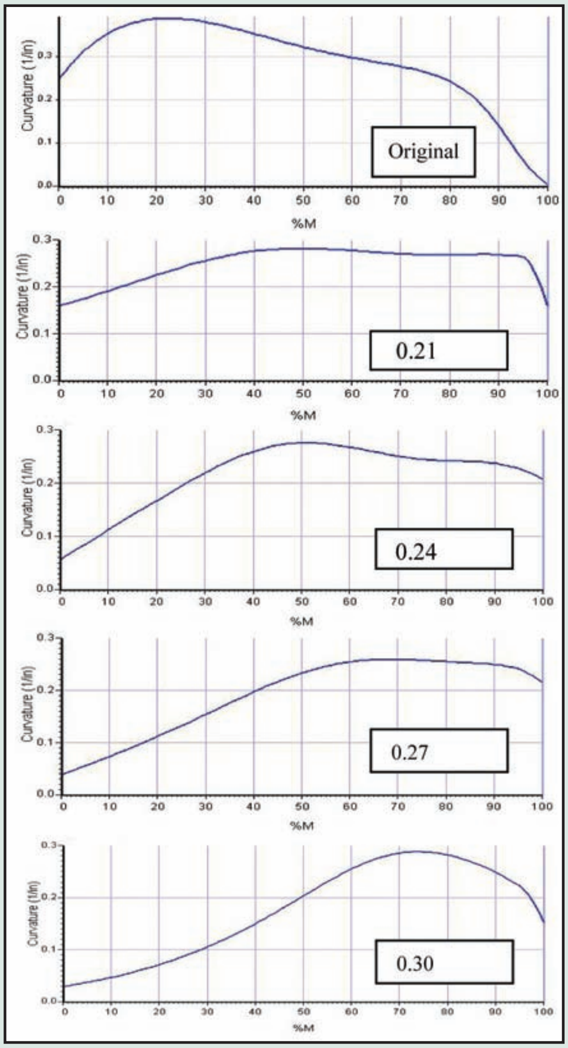


Figure 8 – Shroud Curvature – All Impellers.

and 0.25 for the 0.30, 0.27, 0.24, 0.21 Ax/D2 and original, respectively. Note that there is a y-axis scale change between the original impeller and all other designs.

The reduced turning near the leading edge is very important in suppressing the shroud inlet relative Mach number for high flow coefficient impellers. The larger the turning radius is made near the leading edge, the lower the resultant inlet relative Mach number (Sorokes and Kopko [2007]).

As noted earlier, the blade angle distributions were changed for the four new designs. To illustrate the impact on the flowfield, the isentropic head rise for the shroud streamtube (2-D basis) is shown in Figure 9 for the four new impellers. Note that the head rise at the exit is virtually identical for the four impellers. There is a subtle difference in the rate of change in head rise in the inducer region between the $0.24 Ax/D_2$ design and the remaining impellers due to the rate of change of blade angle in those regions. This small difference was felt to be relatively insignificant to the study being conducted.

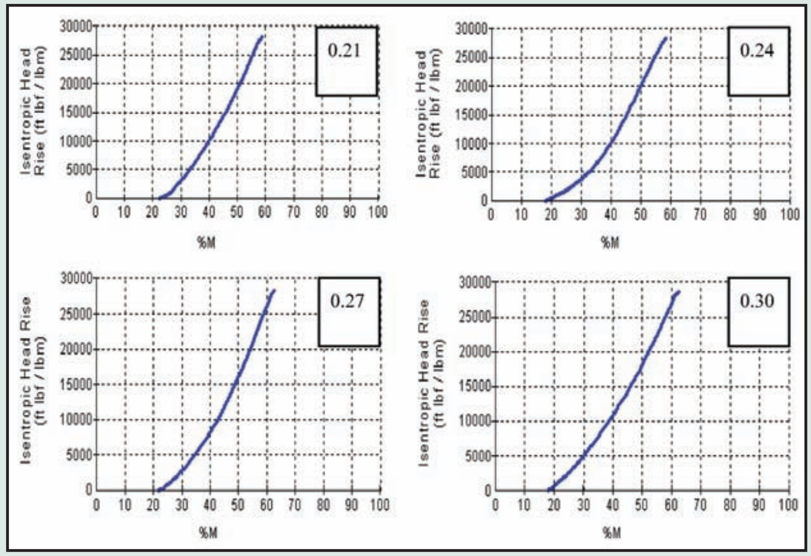


Figure 9 – Isentropic Head Rise – Shroud Streamtube (2-D) – New Impellers.

Vector plots at the impeller exit plane for the four designs are shown in Figure 10. The vectors are colored by relative Mach number and reflect the impeller exit conditions at the design flow rate. Recall that the machine Mach number for all of these cases is fixed at 1.15. Velocity vector plots near the suction surface for each impeller, also at the design flow rate, are shown in Figure 11. By examining Figures 10 and 11, one can see that the impeller exit velocities and/or relative Mach number are more uniform as a result of reduced shroud curvature. Note also the reduction in relative Mach number along the shroud between the $0.21 Ax/D_2$ and the $0.30 Ax/D_2$ impellers in Figure 11. The reduction in local curvature has reduced the Mach number. While not obvious from the figures provided, the longer impeller did provide wider flow range than the shorter versions due in part to the lower Mach number. The additional range would be advantageous for many process compressor applications.

It is also interesting to compare the results in Figures 10 and 11 with those in Figures 1 and 2; *i.e.*, similar plots for the original impeller. The improvements of the new designs over the original are striking as the new designs all show significantly less low momentum fluid at the impeller exit and much more well-behaved flow along the shroud surface.

The downstream stationary flow path components also benefit from the improved impellers. The meridional projection of velocity throughout the analytical domain is given in Figure 12. Examining this figure, one can see the improvement in the diffuser flow field as a result of the reduction in the impeller exit wake

Continued on page 20

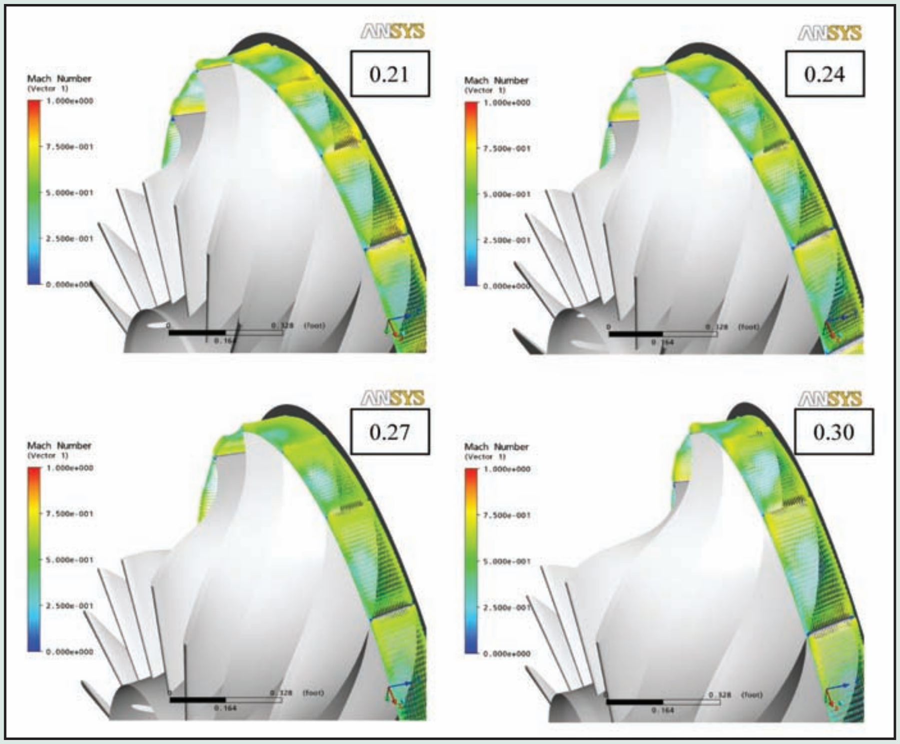


Figure 10 – Impeller Exit Velocity Vectors Colored by Relative Mach Number.

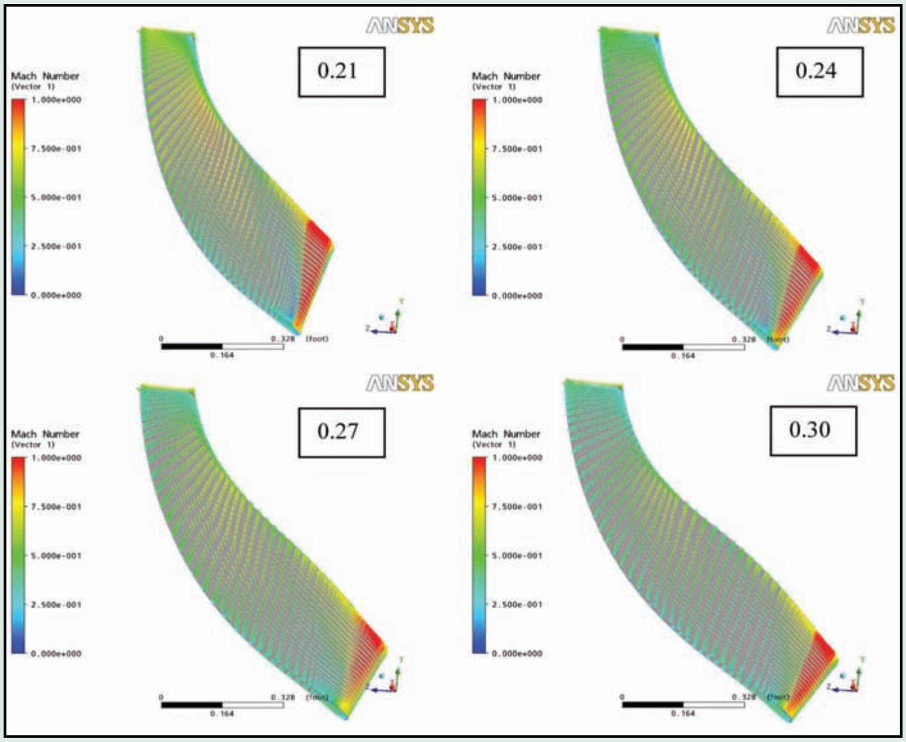


Figure 11 – Velocity Vectors Colored by Relative Mach Number – Near Suction Surface.

region. Note the absence of the low velocity region along the diffuser shroud surface in the 0.30Ax/D_2 design versus the 0.21Ax/D_2 stage. Recall that the reduction in the impeller exit wake exit was achieved through both the reduction in shroud curvature and the narrower tip opening.

Assessments of the new impellers were also made at off-design conditions. In the interest of brevity and space, results for those cases are not presented herein. Suffice to say that the results mimicked those for the design case. That is, the Mach number along the shroud surface near the leading edge is considerably lower for the 0.30 Ax/D_2 case than on the 0.21 Ax/D_2 case. Likewise, the level of secondary flow along the cover and at the impeller exit is reduced for the impeller with the lowest curvature along the cover; *i.e.*, the 0.30 Ax/D_2 impeller. This was particularly obvious on the 80% flow case (see Figure 13), suggesting that the longer impeller would have more surge/stall margin. While difficult to see in the reduced-size figures in the paper, the flow along the suction surface near the impeller exit is significantly more uniform in the 0.30Ax/D_2 impeller than in the 0.21Ax/D_2 design.

Finally, non-dimensional performance curves for efficiency and head coefficient for all of the new impellers are given in Figure 14. In both plots, the efficiency and head coefficients have been normalized using the design point efficiency and head coefficient from the 0.21 Ax/D_2 design. As noted above, several factors contributed to the improved efficiency of the longer impellers over the original design. The reduction in tip dimension was a major contributor as said reduction was instrumental in the significant reduction in the level of secondary flow along the blade suction and shroud surfaces. However, the flowfield improvements realized by reducing the shroud curvature have also translated into aerodynamic performance gains as can be seen by comparing the various designs with the narrower tip dimension. Of the four, the $Ax/D_2 = 0.30$ stage shows the highest performance and widest range. The 0.21 Ax/D_2 has the lower efficiency with the 0.24 Ax/D_2 and 0.27 Ax/D_2 being between the two extremes. In other words, the longest does provide the greatest increase in performance. However, one must consider the possible mechanical implications of the extra impeller length. This will be discussed later in the paper.

RELATED AERODYNAMIC DESIGN FACTORS

In developing the new impellers, it is, of course, impossible to change only the cover curvature and maintain an acceptable aerodynamic design. One could attempt to extend the constant radius shroud surface further into the impeller to reduce the curvature through the leading edge region but this would result in much higher curvature in the exducer portion of the impeller. The result will be separation from the cover or some similar untoward flow phenomenon. Therefore, in an attempt to reduce shroud curvature, other geometric factors must change as well, such as the impeller axial length, the blade angle distribution, and the number of blades. These changes will certainly have an impact on the impeller performance characteristics.

Recall the objective of this study was to develop a new design that could be used for high Mach number applications. Consequently, the new configurations needed to conform to all of the OEM's assessment criteria including, but not limited to: blade-to-blade loading ($\Delta W/W_A$), minimum pressure surface relative velocity, relative velocity ratio (W_{1S}/W_2), wake area fraction, and various other criteria related to velocity distributions and secondary flow. Therefore, in manipulating the shroud curvature, changes were also required in the blade angle distribution to: (a) achieve the desired blade loading; (b) conform to manufacturing requirements on blade lean; (c) match

Continued on page 22

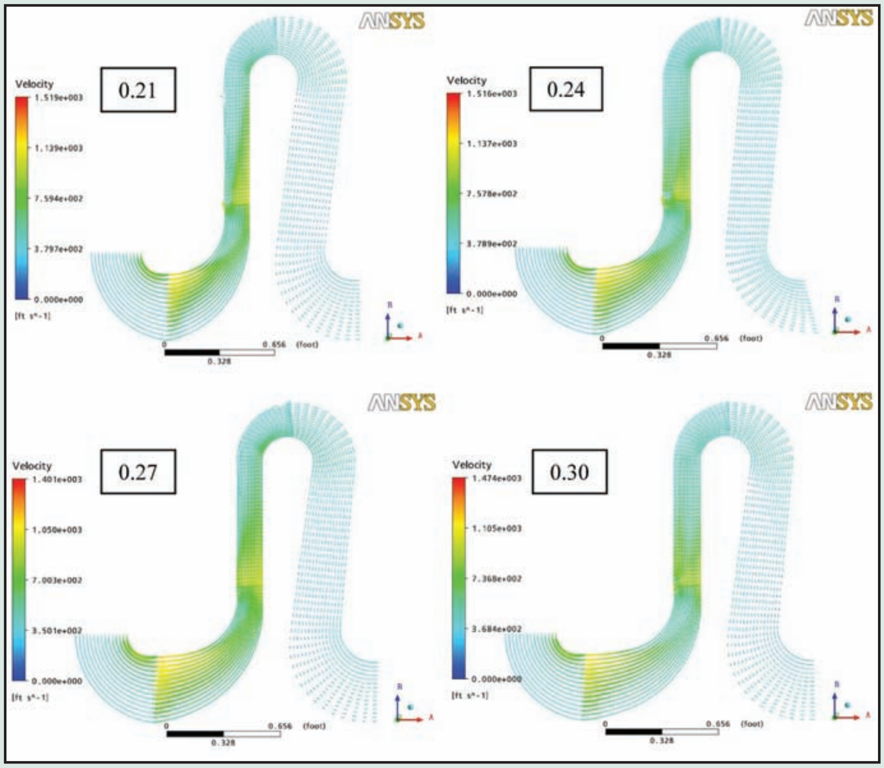


Figure 12 – Average Meridional Velocity Vectors.

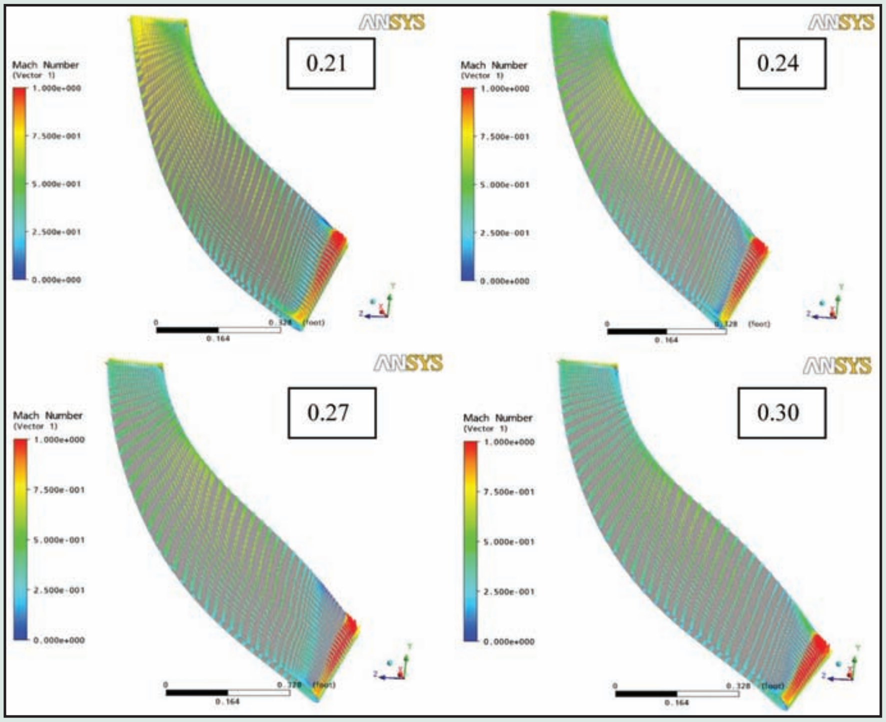


Figure 13 – Velocity Vectors Colored by Relative Mach Number – Near Suction Surface – 80% Flow.

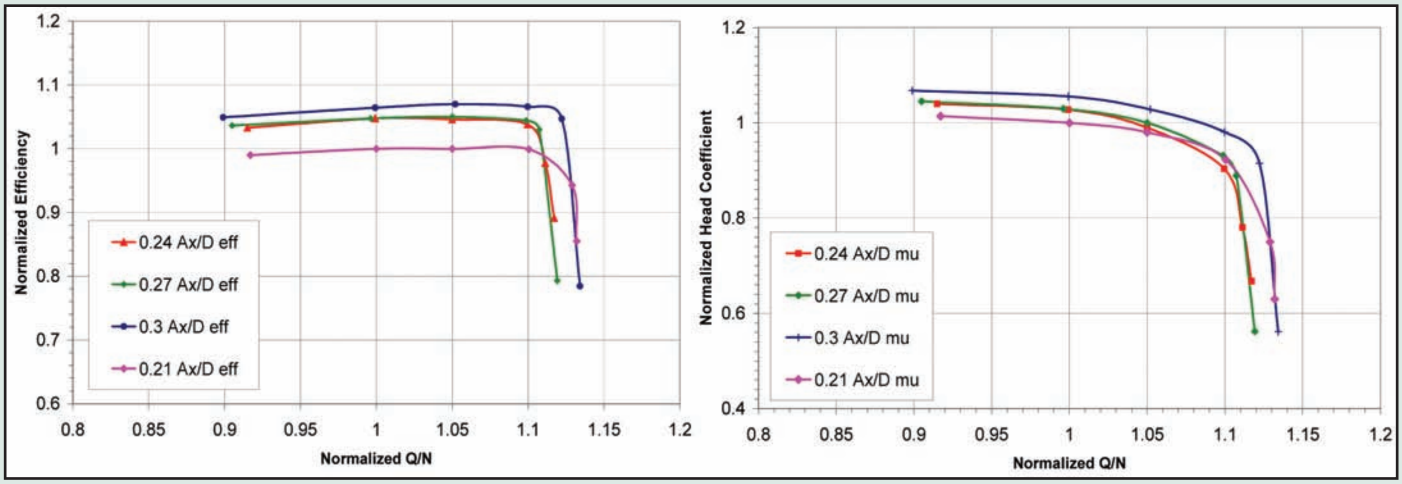


Figure 14 – Normalized Efficiency & Head Coefficient – New Impellers.

the required head coefficient; (d) *etc.* These changes certainly impacted the flowfield within the impeller and, therefore, influenced the performance characteristics.

As noted earlier, in reducing the curvature and increasing the axial length, fewer impeller blades were required to maintain the desired blade loading. Despite the increased length, with fewer blades there is less wetted surface and less friction loss in the impeller. Lower friction loss results in higher impeller efficiency. Further, with fewer blades, it is possible to have a larger throat area due to a reduction in blade blockage. The result will be an improvement in the overload capacity of the impeller (as seen in the 0.30 Ax/D_2 design).

In short, the improvement in impeller performance obtained during this study cannot be solely attributed to the reduction in cover curvature. However, the curvature reduction is primarily responsible for a reduction in the shroud inlet relative Mach number… a key consideration in the design of the new impellers.

MECHANICAL / FABRICATION CONSIDERATIONS

The maximum values of several critical stresses were compared to determine the relative mechanical properties of the impellers. These values are tabulated in Table 2. The values in the table were normalized using the stresses for the original impeller. That is, the stresses for the original impeller are shown as 100% and the various stress levels for the other designs are shown as percentages of those for the original impeller.

As can be seen, the stress levels in the longest impeller (*i.e.*, Ax/D_2 of 0.30) are, on average, the highest while the lowest stresses are found in the 0.21 Ax/D_2 impeller. In particular, the OEM uses the Von Mises stress as a critical assessment parameter in comparing new designs and the maximum Von Mises stress is nearly 11% higher in the 0.30 Ax/D_2 impeller than in the original. In short, while the impeller with the lowest curvature or longest axial length provided the highest performance, it also yielded the highest Von Mises stress level. This could be of concern if the impeller is required to operate at high tip speeds. However, in some applications, the impeller does not operate at high tip speed because the sonic velocity of the gas is low. Therefore, high Machine Mach number designs are required without necessarily requiring a very high tip speed. For example, the sonic velocity of propane is on the order of 740 feet per second. To achieve a machine Mach number of 1.2, the impeller only need operate at a tip speed of 890 feet per second or less. Therefore, the longer impellers, despite their higher stress levels, can be very effective in some applications.

Also of potential concern, as noted earlier, the impeller with the lowest cover curvature ($Ax/D_2 = 0.30$) has the longest axial length. This means that the bearing span for a collection of such impellers would be considerably longer than a collection of 0.21 Ax/D_2 impellers. It is possible that the longer impellers and resulting longer stage axial length would result in an overall bearing span that was unacceptable from a rotordynamics perspective.

Further complicating matters, the longer impellers also will weigh more than their shorter counterparts. This increases the rotating mass and moment of inertia. In short, the potential benefits derived from the reduced curvature or longer length must be weighed against the possible rotordynamic consequences.

One further matter is the potential difficulties one might encounter in fabricating the longer impeller. Revisiting Figure 3, consider the possible complexities in fabricating the 0.3 Ax/D_2 impeller. If the impeller were constructed from two pieces; *i.e.*, a cover welded or brazed to a bladed disk; it might be a challenge to: (a) weld the impeller without causing excess distortion due to the heat input; (b) obtain line of sight to fillet weld the impeller; or (c) achieve the tight fit between the two parts necessary for brazing. Likewise, it might not be possible to single-piece mill this impeller again because of line-of-sight or tooling issues. In short, it does little good to develop a sophisticated aerodynamic design if it cannot be manufactured.

Ultimately, a reasonable compromise between aerodynamic, stress, rotordynamic, and manufacturing must be found to ensure the total effectiveness of any new impeller and/or stage design.

FUTURE WORK

Since the course grids analyses suggested improvement with the reduced curvature (and other factors), additional CFD and FEA work will be completed with more refined grids to obtain a more accurate assessment of the improvements possible with the reduced curvature. The rotordynamics group will also review the potential compromises associated with the increased stage spacing.

Ultimately, final verification of any new impeller and/or stage is achieved when the new design is subjected to performance testing. In this regard, the OEM is currently working toward a test of the new longer designs. Unfortunately, these test results were not available prior to the publication deadline. Once the data are available, a detailed comparison will be completed between the test results and CFD predictions to further refine the model for the influence of shroud curvature.

CONCLUSIONS

The paper has presented the results of a study to determine the impact of cover curvature and axial length on impeller performance. CFD and FEA results were offered to show the aerodynamic and mechanical performance of four impellers of varying shroud curvature and axial length as compared to an older, obsolete design. The analytical results showed the potential aerodynamic benefits derived from lower shroud curvature and the related changes; *i.e.*, axial length, blade angle distribution, number of blades, *etc.* Comments were also offered regarding possible mechanical issues that might preclude use of the longer impellers as well as on the manufacturing considerations that must be addressed.

In closing, there are clear aerodynamic benefits to decreasing the curvature along the impeller shroud but these benefits must be weighed against the impact on the mechanical and manufacturing considerations. If a reasonable compromise can be found between the aerodynamic, stress, rotordynamic, and manufacturing disciplines, a superior design can be derived.

ACKNOWLEDGMENTS

The authors acknowledge our colleague Robert E. Kunselman for his help in developing some of the figures used in this paper. We also thank Mr. Nick D'Orsi of Turbo Solutions Engineering, Dr. Ed Bennett of Mechanical Solutions, and Dr. Chris Robinson and Dr. Mick Casey of PCA Engineering for some very fruitful discussions related to impeller design. Finally, we thank Dresser-Rand for allowing us to publish this paper.

DISCLAIMER

NOMENCLATURE

Φ = flow coefficient =

ΔW = difference in relative velocity suction surface to pressure surface

Ax = axial length from shroud leading edge to hub plane at impeller exit

D_2 = impeller exit diameter in inches

D_{shaft} = compressor shaft diameter in inches

N = speed in rpm

Q = volumetric flow in ACFM

Sx = maximum stress in x-direction

Sy = maximum stress in y-direction

Sz = maximum stress in z-direction

S1 = maximum principal stress

SEQV = maximum Von Mises nodal stress

VONM = maximum Von Mises elemental stress

W_A = mean relative velocity

W_{1S} = shroud inlet relative velocity

W_2 = impeller exit relative velocity

REFERENCES

1. Al-Zubaidy, S.N.J, "Axial length influence on the performance of centrifugal impellers," Journal of Propulsion and Power (ISSN 0748-4658), vol. 8, no. 6, p. 1245-1251, 1992

2. Aungier, R. H., Centrifugal Compressors – A Strategy for Aerodynamic Design and Analysis, ASME Press, 2000.

3. Bammert, K.; Rautenberg, M.; Knapp, P., "The influence of the meridional impeller shape on the energy-transfer in centrifugal compressors," ASME IGTI, 1980

4. Casey, M.V., 1994, "The Industrial Use of CFD in the Design of Turbomachinery", AGARD Lecture Series 195, Turbomachinery Design Using CFD.

5. Cumpsty, N. A., Compressor Aerodynamics, Longman Scientific & Technical, 1989

6. Japikse, D., Centrifugal Compressor Design and Performance, Concepts ETI, Inc., 1996

7. Raw, M.R., Galpin, P.F. and Hutchinson, B.R., 1989, "A Co-Located Finite Volume Method For Solving The Navier-Stokes Equations For Incompressible And Compressible Flows In Turbomachinery: Results And Applications", J. Canadian Aeronautics and Space, v35, No. 4.

8. Sapiro L. "The effect of extended shrouds on centrifugal compressor performance as a function of specific speed" ASME 82-GT-228, 1982.

9. Sorokes, J.M., "The Practical Application of CFD in the Design of Industrial Centrifugal Impellers," Texas A&M Turbomachinery Symposium Proceedings, 1993.

10. Sorokes, J.M., Kopko, J.A., "High Inlet Relative Mach Number Centrifugal Compressor Impeller Design," ASME paper no. GT2007-27864, 2007.