Proceedings of the Forty-First Turbomachinery Symposium
September 24-27, 2012, Houston, Texas

CENTRIFUGAL STAGE PERFORMANCE PREDICTION AND VALIDATION FOR HIGH MACH NUMBER APPLICATIONS

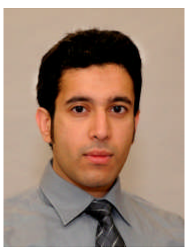
Steven C. Kowalski
Aero/Thermo Design Engineer
Dresser-Rand Company
Olean, NY, USA

Jorge E. Pacheco, PhD
Manager, Aero/Thermo Design
Dresser-Rand Company
Olean, NY, USA

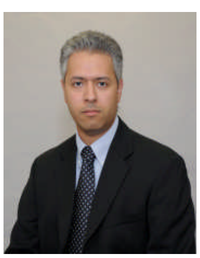
Syed Fakhri
Aero/Thermo Design Engineer
Dresser-Rand Company
Olean, NY, USA

James M. Sorokes
Principal Development Engineer
Dresser-Rand Company
Olean, NY, USA

Steve Kowalski is an Aero-Thermo Engineer in the Research and Development Department of Dresser-Rand. His responsibilities include the design, development and analysis of aerodynamic components of centrifugal compressors. He has been employed at Dresser-Rand since 2007. He holds a B.S. degree from Michigan State University (Mechanical Engineering, 2002) and an M.S. degree from Michigan State University (Mechanical Engineering, 2005). Prior to joining Dresser-Rand, he worked as a teaching/research assistant at Michigan State University.



Syed Fakhri is an Aero-Thermo Engineer in the Research and Development Department of Dresser-Rand. His responsibilities include the design, development and analysis of aerodynamic components of centrifugal compressors, in addition to development of 1D compressor selection and performance prediction codes. He has been employed at Dresser-Rand since 2010. He holds a B.E. degree from Manipal University (Mechanical Engineering, 2007) and an M.S. degree from The Pennsylvania State University (Mechanical Engineering, 2009). Prior to joining Dresser-Rand, he worked as a research assistant at The Pennsylvania State University while pursuing his M.S. degree.



Dr. Jorge Pacheco is Manager of the Aero-Thermo group for Dresser-Rand in Olean, New York. His team is involved in new centrifugal compressor stage development, multiphase modeling for gas-liquid separation, full-scale and sub-scale compressor testing, and axial compressor and turbine product development support. He has been employed at Dresser-Rand since 2007, working as Development Engineer and Supervisor of Solid Mechanics. Dr. Pacheco received his B.S. degree (mechanical engineering, 1997) from Universidad Simon Bolivar and Ph.D. degree (mechanical engineering, 2003) from Carnegie Mellon University. Before joining Dresser-Rand, Dr. Pacheco was a Professor at Universidad Simon Bolivar for three years in the Thermodynamics and Transport Phenomena Department. Dr. Pacheco is a member of ASME and has authored or co-authored more than 16 technical papers.

James M. "Jim" Sorokes is a Principal Engineer at Dresser-Rand with over 35 years of experience in the turbomachinery industry. Jim joined Dresser-Clark (now Dresser-Rand) after graduating from St. Bonaventure University in 1976. He spent 28 years in the Aerodynamics Group, became the Supervisor of Aerodynamics in 1984 and was promoted to Manager of Aero/Thermo Design Engineering in 2001. While in the Aerodynamics Group, his primary responsibilities included the development, design, and analysis of all aerodynamic components of centrifugal compressors. In 2004, Jim was named Manager of Development Engineering whereupon he became involved in all aspects of new product development and product upgrades. In 2005, Jim was promoted to principal engineer responsible for various projects related to compressor development and testing. He is also heavily involved in mentoring and training in the field of aerodynamic design, analysis, and testing.

Jim is a member of AIAA, ASME, and the ASME Turbomachinery Committee. He has authored or co-authored over forty technical papers and has instructed seminars and tutorials at Texas A&M and Dresser-Rand. He currently holds three U.S. patents and has two others patents pending. He was elected an ASME Fellow in 2008.

ABSTRACT

Centrifugal compressors used for process applications like Liquefied Natural Gas (LNG) plants require operation at high machine Mach numbers (>1) due to the heavy gas and reduced footprint requirements. Performance guarantees are commonly specified as per API-617, but some end users are requiring tighter performance tolerances. This paper describes Computational Fluid Dynamics (CFD) performance predictions and their validation using test rig results.

The experimental results presented in this paper were obtained using a heavily instrumented compressor test rig. The instrumentation used allows measurement of necessary stage performance parameters such as total and static pressure, total temperature and flow angle among others.

Due to the target tolerances, CFD was used to predict the performance rather than alternative methods (mean-line codes, 2D prediction codes, etc.). The results of the CFD analyses are described in the paper, including flow field characteristics and performance parameters.

The performance results from the tested stages matched well with the CFD prediction. The performance predicted using CFD was also well within API-617 tolerances. The comparison using the tighter end user tolerances showed that CFD predictions can meet the requirements for efficiency, head and overload margin. There is still more work to be done to meet the tighter tolerances regarding surge/stall margin. Overall, CFD analysis has been shown to have the necessary accuracy to predict performance within tight tolerances for challenging applications such as high machine Mach number.

INTRODUCTION

Original Equipment Manufacturers (OEMs) that provide centrifugal compressors for the process market industry; *i.e.*, oil and gas, petrochemical, gas transmission applications and the like; have seen an increasing demand for stages operating at higher flow coefficients and higher machine or inlet relative Mach numbers. These demands are driven by a desire to reduce the footprint of the compressor or to compress larger amounts of gas within a smaller casing. Further, as heavier gases are being used, higher speeds are being required for similar compression. As a direct result, process centrifugal compressors now operate at machine Mach numbers, U2/A0s, in excess of 1.2 and shroud inlet relative Mach numbers of 0.95 and higher. While very commonplace in gas turbine or turbocharger applications, such transonic designs have been, until now, quite uncommon in the process market. This is due, in large part, to the reduced operating range associated with such Mach number levels as addressed by Sorokes (1993) and others. However, many end users are now willing to accept the restricted range in their desire to handle greater flow capacity in smaller sized compressors.

The authors' company, as well as many others, have completed a number of high flow coefficient, high Mach number impeller designs over the past several years. Most of the factors considered in these designs have been addressed in textbooks by Cumpsty (1989), Japikse (1996), Aungier (2000) and others and in papers by Bammert et al (1980), Al-Zubaidy et al (1992), Sorokes and Kopko(2007), Sorokes et al (2009), and many others. As many have discovered, it is insufficient to only optimize the impeller design. While the impeller is certainly the most critical component in a well-performing stage, failure to give proper attention to the stationary components associated with high flow coefficient, high Mach number applications will severely compromise the resulting performance.

This paper will describe an effort undertaken to predict and validate the performance of a new family of high flow coefficient, high inlet relative Mach number impellers developed for applications such as LNG.

The paper will provide a general description of the new stages. The discussion will then turn to the analytical assessment of the new stages and the flowfield trends observed in the flow path. This section of the paper will include: a description of: the computational domain; CFD software used; boundary conditions applied; turbulence model employed; etc. Next, a description of the test vehicle and test loop will be offered including a brief description of the rig itself, the instrumentation, test gases, operating conditions, etc. The test results for the new stages will be presented and, again, the focus will be on the performance curve characteristics of the stages. The comparison of the results from test and CFD will be presented next. Finally, recommendations regarding improvement of the performance predictions will be offered and concluding remarks will be presented.

DESCRIPTION OF NEW STAGES

The stages described in this work were designed as part of an overall initiative to develop new high performance staging, operating at different head levels, for the OEM's products. The primary intent was to increase the stage efficiency, increase overall flow range for high flow coefficient impellers (ϕ > 0.100), and match the design head levels. The full-inducer, arbitrary-bladed impellers were developed for use in any of the stages (*i.e.*, first, intermediate or last) of a multi-stage centrifugal compressor. The target flow coefficient for both impellers used in the study and addressed in this paper was 0.12. Each stage was designed for a specific head, low and medium heads respectively. Both new stages were required to operate over a machine Mach number range of 0.85 to 1.20. The stages also included an upstream inlet guide, diffuser, return bend and return channel (or de-swirl cascade). The parallel wall diffuser width was set such that the flow angle did not violate any of the widely-accepted stall avoidance criteria such as those of Kobayashi-Nishida (1990). The return channel system was also developed using well-established design guidelines, ensuring proper leading edge incidence, passage area distribution, vane shape, etc. to avoid any untoward flow anomalies.

Finally, both impellers were designed to have the same flow coefficient and the same Mach number. .

ANALYTICAL RESULTS

The new stages, both of which have a shroud inlet relative Mach number(Mrel1S) of 0.94, were developed using a typical design methodology; *i.e.*, a blend of 1-D, 2-D, and 3-D analytical methods. It was the 3-D CFD assessment of the new stages that determined whether the stages met the design targets.

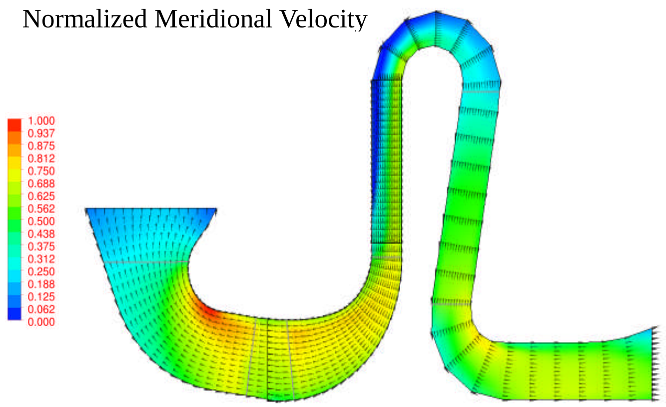


Figure 1. Meridional velocity profile (Low Head Stage) – 100 percent flow

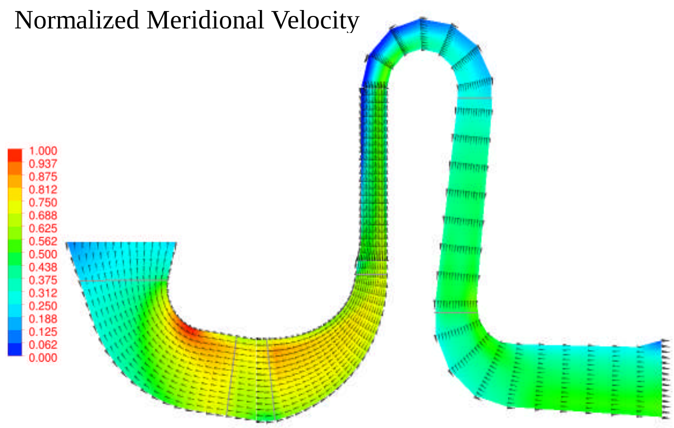


Figure 2. Meridional velocity profile (Medium Head Stage) – 100 percent flow

CFD Approach

All analyses described in this paper were conducted using the commercially-available solver, ANSYS/CFX. The computational domain was a sector or "pie-slice" model that included the upstream inlet guide, impeller, diffuser, return bend, return channel, and exit section. The grids were composed of sufficient elements to ensure grid independence.

All interfaces between the sector components were modeled using a "stage" or circumferentially averaged interface. The k-E turbulence model and a high-resolution discretization scheme were applied as this OEM has had good success using these approaches.

The overall flowfield in the new stages were assessed at design flow and at incrementally lower flow rates until CFD predicted that the stage head coefficient had peaked and decreased with further reductions in flow. Based on the OEM's experience, the point at which the head coefficient curve is predicted to change from a negative to a positive slope is indicative of the flow rate at or near which the stage will stall on test.

It must be noted that all analyses presented in this paper were run assuming steady state conditions. It is recognized that the conditions associated with stall and the like are highly unsteady, so the nature of the simulated flowfield at the near stall conditions might deviate somewhat from the real flowfield. Still, based on the OEM's extensive experience comparing CFD to test results and in comparing steady state and unsteady CFD analyses, steady state results have been found to provide reasonable results relative to the assessment of minimum stable flow on test. Further investigations using unsteady CFD are being considered in order to improve the prediction even further.

The circumferentially averaged normalized meridional velocity profiles for both stages at design flow are provided in Figure 1 and Figure 2.

In reviewing the trends in the normalized meridional velocity, a low momentum region was observed along the shroud surface of the diffuser at design flow for both designs (100 percent flow-- See Figures 1 and 2). The low head coefficient stage exhibited a larger low momentum region than the medium head coefficient stage. This is due to the fact that the diffuser of the medium head stage was pinched and hence the flow was accelerated through it, unlike the low head stage. Pinching the diffuser is an approach used to adjust the performance level and range of stages. The effect of this pinching would be apparent in the performance curves and flowfield contour plots of the stages, which will be discussed later. These two stages were tested and the results of these tests were used to validate CFD's ability to accurately predict the performance of these two different stages.

OVERALL TEST VEHICLE DESCRIPTION

The tests were conducted in the OEM's sub-scale test rig, which can be configured to either a single or two-stage arrangement. The rig is comprised of a series of stackable rings that form both the aerodynamic flow path and the rig casing. The ring concept allows all instrumentation leads to be extracted thorough the outside diameter of the rings, facilitating instrumentation connections to the data acquisition system.

The rig is driven by a 1.12MW (1500HP) electric motor and speed increasing gear, offering a wide range of operating speeds. The rig is installed in a closed-loop system similar to the schematic shown in Figure 3. Available test gases include nitrogen, carbon dioxide, R-134A refrigerant, and, if necessary, helium-nitrogen mixtures.

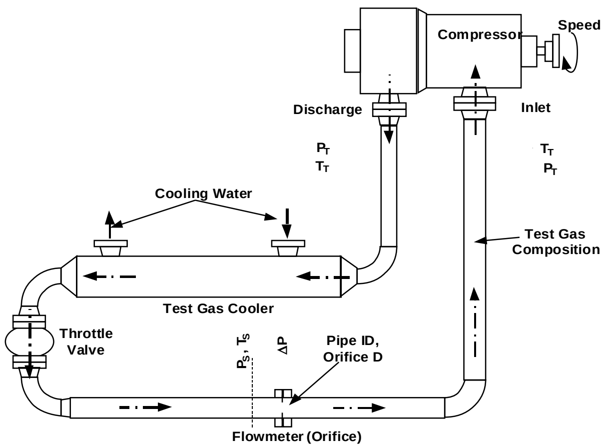


Figure 3. Schematic of test loop with main components

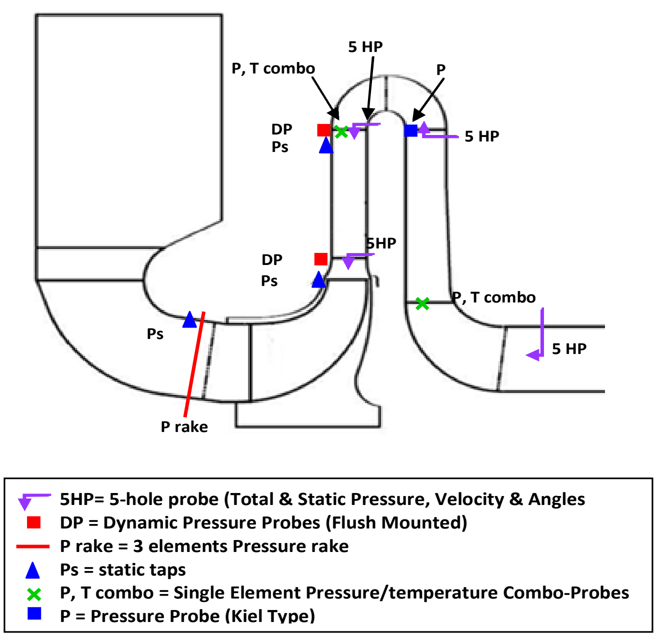


Figure 4. Typical test rig internal instrumentation layout

The stackable ring construction allows greater flexibility in rig configurations. It is possible to test a first stage configuration (*i.e.*, following a main inlet), an intermediate stage, and a discharge stage (*i.e.*, with a volute or collector, two stages with an intermediate sidestream, etc.

The testing of the stages presented in this paper was done in a single stage configuration as shown in Figure 4. The stage was operated in accordance with ASME PTC-10 (1997) at machine Mach numbers ranging from a maximum U2/A0 of 1.2 down to a minimum U2/A0 of 0.6. However, the stationary components were optimized for a U2/A0 of 1.15, as the stage was primarily targeted for heavy mole weight applications. For each machine Mach number, a full speed line of data was taken; typically six to twelve thermodynamically-settled flow points from overload (choke) to stall/surge. Further, the data acquisition system captures performance data every four seconds, accumulating so-called "transition data" that provides additional insight into the compressor operation between settled points.

The stage was extensively instrumented, as shown in Figure 4. Included were so-called "combo probes" that include a half-shielded thermocouple and Kiel-head pressure probes to measure total pressure and total temperature, individual Kiel-head pressure probes, total pressure rakes, as well as 5-hole probes, static pressure taps, and dynamic pressure transducers. In addition, proximity probes were used to monitor rotor vibrations. The dynamic pressure transducers (which detect pressure fluctuations) and the 5-hole probes (which measure both pressure and flow angle) at the diffuser inlet and exit are critical in the assessment of the diffuser flow physics.

For this test program, there were two 5-hole probes at the diffuser inlet and two at the diffuser exit. At each location, the probes were placed at 33 percent and 66 percent immersion from hub to shroud respectively. The probe at 33 percent immersion is closer from the hub wall while the one at 66 percent immersion is closer to the shroud wall. The probes were used to measure the absolute gas angle at the two depths.

There were three dynamic pressure transducers mounted at the same radial location as the 5-hole probes; *i.e.*, three at the diffuser inlet and three at the diffuser exit. The dynamic probes were distributed circumferentially so that it would be possible to detect stall cells as well as the number of cells and their direction of rotation

Several observers monitored the vibration and dynamic pressure transducer signals as well as the real-time display of the diffuser flow angles (and other aerodynamic parameters) to take note of any interesting phenomena that occurred during the testing. In short, every effort was made to ensure that all critical data was captured during the run.

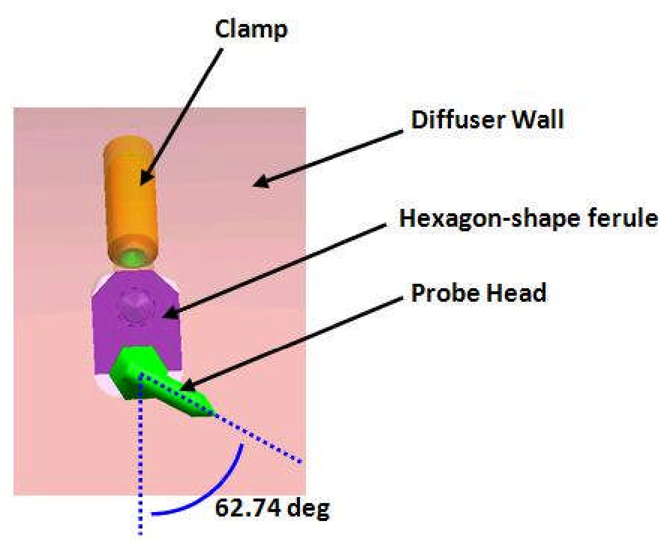


Figure 5. -- 5-hole probe

COMPARISON OF ANALYSIS AND TEST RESULTS

A comparison of the performance curves obtained from the CFD analyses and the measurements taken during the rig testing was completed in order to assess the capability of steady-state CFD to accurately predict performance. Overlays of the CFD and test data for the 1.15 U2/A0 speed line are shown in Figures 6 through 10. The predicted and tested performance curves (Efficiency and Head coefficient – see nomenclature) of the medium and low head coefficient stages are shown in figures 6 and 7. These figures show a very good agreement between the CFD prediction and the test results. For the low head coefficient stage (Figure 6), the CFD is slightly over predicting efficiency at low flow and under predicting at higher flow. The head coefficient matches very well across the entire curve. The flow at which peak head coefficient occurs is slightly under predicted by the CFD

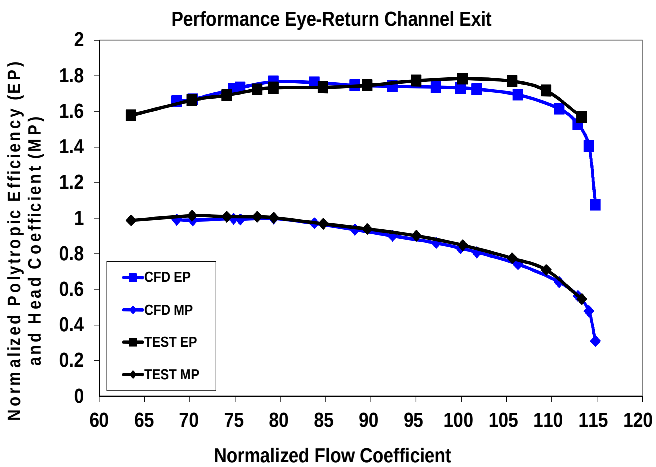


Figure 6. Normalized performance from impeller eye to return channel exit versus normalized flow coefficient at U2/A0=1.15–Test vs. CFD (Low head coefficient stage)

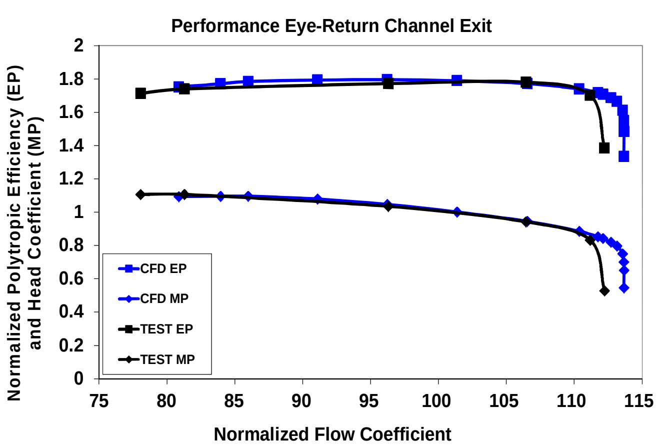


Figure 7. Normalized performance from impeller eye to return channel exit versus normalized flow coefficient at U2/A0=1.15–Test vs. CFD (Medium head coefficient stage)

For the medium head coefficient stage (Figure 7), the agreement both on efficiency and head coefficient is excellent. The CFD is over predicting the flow at overload (choke) and under predicting the flow at peak head coefficient.

Most end users require that the centrifugal compressor meets API-617 (2009) standard guarantees, which require the compressor, without the use of a bypass, be suitable for continuous operation at any capacity at least 10% greater than the predicted surge capacity shown in the proposal while for overload there is no requirement. The overall power is required to be within +-4%.

For LNG applications (which utilize high flow coefficient stages at high Mach number) certain end users are requiring tighter tolerances: efficiency and head coefficient at design point within ± 2%, flow at overload (choke) within ± 4% and flow at stall (or peak head) within ± 4%. The values of the parameters that were used in the evaluation of the performance prediction (CFD) of the stages in order to determine if the required tolerances were satisfied are shown in Tables 1 and 2.

Table 1. Low head coefficient stage (see Figure 6)

Parameter	Test	CFD	Deviation
Norm. Efficiency @ design	1.7750	1.7306	-2.50%
Norm. Head coeff. @ design	0.8437	0.8295	-1.69%
Norm. Flow @ overload	NA	114.79	
Norm. Flow @ stall	70.241	75.567	7.58%

The efficiency at design flow predicted by CFD for the low head coefficient stage is well within the API-617 requirement for power of ± 4%. The deviation of -2.5% is almost within the tighter tolerances required by some clients. Test results could not be obtained at normalized flows larger than 113% of design flow due to loop resistance limitations. Therefore, the flow at overload cannot be exactly determined for the test. If the farthest point into overload from test is used as a reference, the CFD is under-predicting the normalized flow at overload by just 1.2%.

As previously indicated, the most challenging prediction for the CFD is at stall or surge conditions (or peak head coefficient) due to the intrinsic unsteadiness of the flow. The steady-state CFD is capable of predicting within the commonly used ± 10%. For this stage the CFD is under predicting the flow at stall/surge by 7.6%, which is above the stricter target of ± 4%. The test showed a better stall/surge margin. Consequently, the machine will have a larger operating range than initially predicted which leads to the observation that the CFD is being too conservative. Further transient simulations are on-going and preliminary results indicate even better correlation between test and CFD.

Table 2. Medium head coefficient stage (see Figure 7)

Parameter	Test	CFD	Deviation
Norm. Efficiency @ design	1.7741	1.7897	0.88%
Norm. Head coeff. @ design	1.0001	1.0119	1.18%
Norm. Flow @ overload	112.22	113.65	1.27%
Norm. Flow @ stall	81.278	85.977	5.78%

The results for the medium head coefficient stage are even more encouraging. The deviation in efficiency and head coefficient at design flow are well below the target of ± 2%. The flow at overload is over predicted by just 1.27%, which is less than half of the target range of ± 4%. The deviation in flow at stall/surge is lower for the medium head coefficient stage, but it is still above the target. Transient CFD simulations are expected to improve the deviation within the target range.

Other comparison parameters between CFD and test that are used to evaluate CFD prediction capabilities are shown in Figures 8 through 10. The results shown here are for the low head coefficient stage. The results for the medium head coefficient stage are very similar and have not being included for conciseness.

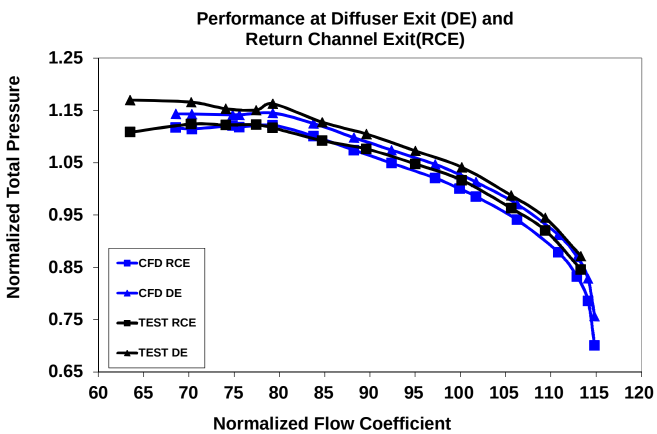


Figure 8. Normalized total pressure at diffuser exit and return channel exit versus normalized flow coefficient at U2/A0=1.15–Test v. CFD (Low head coefficient stage)

Normalized total pressure at the exit of the diffuser and at the exit of the return channel versus normalized flow coefficient are shown in Figure 8. CFD is able to predict the total pressure at design flow within 0.9% at the diffuser exit and within 1.1% at the return channel exit.

The normalized pressure recovery and loss coefficient for the return channel are shown in Figure 9. These two parameters help evaluate the frictional losses and the pressure conversion from overload to stall/surge. There is good agreement between CFD and test, both show similar trends.

The comparison of flow angles between test and CFD at the diffuser inlet are shown in Figure 10. Flow angle is generally used to evaluate the quality of flow entering the diffuser and exiting it. Five-hole probes were placed near the hub and shroud at diffuser inlet during testing. When comparing with CFD flow angles at these locations, we can observe from Figure 10 that there is very good agreement between CFD and test. CFD was able to accurately predict the magnitude and general trend of the flow angle at these locations. Some deviation between the test and CFD can be expected near overload and surge due to probe incidence error and increasing hub to shroud flow quality variation.

These results, in conjunction with the overall performance curves for these stages, show that the OEM's CFD analysis can accurately predict the performance of a stage at various stations/locations within it. For applications with multiple incoming flows (sidestreams) that mix with the core flow of the compressor; being able to accurately predict the total pressure at the flange is crucial. Since the discharge pressure of the stage preceding the sidestream is instrumental in setting the sidestream flange pressure, it is critical that the stage discharge pressure be predicted accurately.

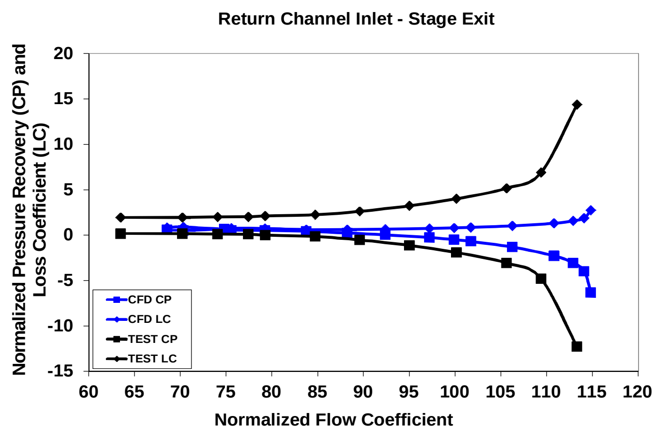


Figure 9. Normalized performance from return channel inlet to stage exit versus normalized flow coefficient at U2/A0=1.15–Test v. CFD (Low head coefficient stage)

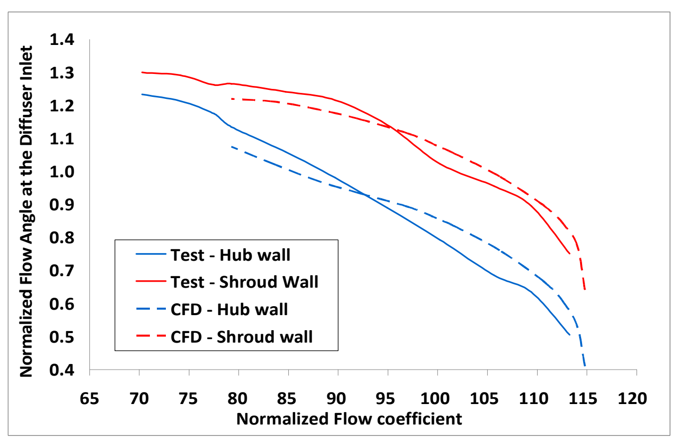


Figure 10. Flow angle at diffuser inlet versus normalized flow coefficient at U2/A0=1.15–Test v. CFD (Low head coefficient stage)

FURTHER PREDICTION IMPROVEMENTS

The results shown above validate the capability of CFD to predict stage performance within the API-617 and standard guidelines requirements for efficiency (power), head and overload and stall flow margin. A tremendous effort has been made over the years to reduce the deviation between CFD and test. However, in order to reach the tighter tolerances specified for certain applications, more work needs to be done. The current steady-state CFD setup was able to meet the tighter tolerances for efficiency, head and overload flow. The intrinsic unsteadiness of the flow near stall/surge restricts the capabilities of the steady-state CFD. Transient CFD simulations have been limited in the past due to amount of time required given hardware and software constraints. Improvements in CFD solver capabilities and faster hardware have eased the limitations on transient Turbomachinery CFD. Currently, fully transient and pseudo transient (non-linear harmonic) CFD simulations are being explored by this OEM to better reproduce the conditions near stall/surge. It is expected this will aide in the improvement of the prediction of the stall/surge margin within the tighter requirements specified for certain applications (i.e. LNG).

CONCLUDING REMARKS

This paper presented the results of analytical and experimental work undertaken for the design of new high flow coefficient, high Mach number stages. This study demonstrated the capabilities of CFD to predict performance by comparing the analytical results to test results generated in the OEM's development test rig. Two centrifugal compressor stages of different head coefficient targets were evaluated. The CFD performance predictions for these two stages met API-617 and standard guidelines for efficiency (power), head, overload margin and stall/surge margin. In addition to this, CFD was shown to accurately predict several performance parameters at various locations within the stage, namely the diffuser and return channel. This allows the OEM to analyze flow behavior qualitatively and quantitatively at various locations within the stage prior to actual testing of the stage.

The paper noted that clients require that centrifugal compressors be able to perform at high efficiency levels over a wide flow range for their processes. For this, it is critical that the OEM be able to not only accurately predict performance at design conditions but also at off-design conditions across the range. Certain applications with large complex equipment like LNG plants are pushing the boundaries of previous guarantee requirements into tighter limits. Compressors used for several LNG applications comprise of complex sections with incoming/outgoing sidestreams. In such cases, it is imperative that stage/internal performances be accurately predicted as the first step to predicting sidestream sectional performance. The CFD performance predictions were evaluated against these tighter constraints and were found to be capable of predicting efficiency, head and overload margin satisfactorily. The one area that requires more development is performance predictions at near stall/surge conditions. Current investigations on fully transient and pseudo transient CFD have shown encouraging results.

Having accurate prediction tools (CFD) leads to greater benefits for OEMs and also for the end users. These prediction tools can provide cost effective evaluations of multiple scenarios and conditions and provide increased confidence in the performance even without testing. Most experimental based tools (1D and 2D) have good agreement over the range in which they were tuned for. The downside to using these types of tools is having to go through the laborious task of tuning them post-test. CFD, on the other hand, is generally more trusted pre-test, especially for applications that require new out of the box designs for which the empirical tools haven't yet been tuned for. CFD based tools have the potential of predicting over a wider range, prior to testing, given its adaptability to changes in the flow behavior. An OEM's overall art-to-part cycle time can be reduced with use of CFD tools during the design and testing portions of the process. Greater configuration flexibility can be obtained for this type of highly engineered-to-order equipment; as geometry can be optimized for users' conditions.

NOMENCLATURE

ϕ = flow coefficient = $700.33 \dfrac{Q}{N D_2^3}$

A0 = sonic velocity of gas in feet per second

D_2 = impeller exit diameter in inches

N = operating speed in rotations per minute (rpm)

PT Combo =

Q = volumetric flow in cubic feet per minute

U_2 = impeller tip speed in feet per second

EP = Polytropic Efficiency

MP = Polytropic head coefficient

CP = Pressure Recovery

LC = Loss Coefficient

ACKNOWLEDGMENTS

The authors thank the Drafting and Test Departments for their support in this study. Finally, the authors thank Dresser-Rand for funding this research and for granting the authors permission to write and present this paper.

REFERENCES

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

American Petroleum Institute, 2009, API Std 617, "Axial and Centrifugal Compressors and Expander-compressors for Petroleum, Chemical and Gas Industry Services", Seventh Edition.

ASME, 1997, PTC 10, "Performance Test Code on Compressors and Exhausters", ASME Press.

Aungier, R. H., 2000, Centrifugal Compressors – A Strategy for Aerodynamic Design and Analysis, ASME Press.
Bammert, K.; Rautenberg, M.; Knapp, P., 1980, “The influence of the meridional impeller shape on the energy-transfer in centrifugal compressors,” ASME IGTI
Cumpsty, N. A., 1989, Compressor Aerodynamics, Longman Scientific & Technical.
Japikse, D., 1996, Centrifugal Compressor Design and Performance, Concepts ETI, Inc.
Kobayashi, H., Nishida, H., Takagi, T., Fukushima, Y., 1990, “A Study On The Rotating Stall Of Centrifugal Compressors,” (2nd Report, Effect of Vaneless Diffuser Inlet Shape On Rotating Stall) Transactions Of JSME (B Edition), 56(529): 98-103.
Sorokes, J.M., 2003, “Range Versus Efficiency – A Dilemma for Compressor Designers and Users,” ASME paper no. IMECE2003-55223.
Sorokes, J.M., Kopko, J.A., 2007, “High Inlet Relative Mach Number Centrifugal Compressor Impeller Design,” ASME paper no. GT2007-27864.
Sorokes, J.M., Kopko, J.A., Geise, P.R., Hinklein, A,L., 2009, “The Influence of Shroud Curvature and Other Related Factors on Impeller Performance Characteristics,” ASME paper no. GT2009-60109.