Global Journals LaTeX JournalKaleidoscope™

Artificial Intelligence formulated this projection for compatibility purposes from the original article published at Global Journals. However, this technology is currently in beta. *Therefore, kindly ignore odd layouts, missed formulae, text, tables, or figures.*

Numerical Studies on Centrifugal Impeller Performance with Different Lean Combinations

K V Jagadeeshwar Chary

Received: 9 December 2016 Accepted: 3 January 2017 Published: 15 January 2017

Abstract

Centrifugal compressors are designed for a given operating pressure and mass flow rate. These machines are often run at off design operating conditions depending on the requirement from industrial to Aerospace applications. The need to maintain relatively high efficiency under off design conditions with adequate stall margin makes the compressor design more challenging. These necessities demand for improvement in the flow conditions through the impeller by optimizing the vane shape. Much of the research was carried out on impeller vane shape to minimize the wake regions at impeller exit and one such effort was to introduce a blade lean at impeller inlet and exit. Investigation from the experimental studies revealed the authors that the introduction of lean at exit suppressed the wake flow regions and henceforth improved the impeller performance either with improved pressure rise or with increased stall margin. Though many of the research studies have proven the influence of lean on the change in the Centrifugal impeller performance, the study was pertained to a lean with a combination of positive inlet and exit leans or negative inlet and exit leans that has shown no change in surge margin improvement. Moreover, the study was pertained to a transonic impeller and the flow investigations are confined to two operating conditions only. Considering the authors knowledge/experience on usage of lean, our current research is aimed to investigate the performance of a subsonic impeller that has a combination of positive inlet and negative exit lean. This impeller model is further compared for performance with the other two different impeller models viz., an impeller with negative exit lean and the impeller with zero lean. All these three different lean angle combinations are focused to understand the aerodynamic performance, stability margin for steady state operation and flow behavior throughout the operating range. The performance results revealed a noticeable improvement

---

Index terms— lean angle, stall margin, centrifugal impeller

1 Introduction

entrifugal compressors are widely used in various applications viz., aviation, oil & gas, refrigeration, etc. These type of roto dynamic machines are majorly used in turboprop, turbo-shafts and auxiliary power units for air compression due to their high pressure raising capability in a single stage and their robustness in case of foreign object damage. Centrifugal compressors are capable of producing pressure ratio up to 6:1 in a single stage made of high strength metal alloys. Multistage centrifugal compressors are not preferred in aviation industry because of the huge pressure losses accompanied when compared to multistage axial flow compressors. There have been continuous efforts to improve the performance of centrifugal compressors. To begin with, the impeller design is initially constrained by selection of specific speed that predetermines the impeller characteristics. With the available design concepts viz., blade backswept/radial, impeller shroud &inducer arrangements and the use of splitter blades resulted in the centrifugal impeller advancements to be exhausted. Despite this tremendous design changes identified for the best performance, the flow distribution from the impeller has always become complex and inevitable.

Compressor performance improvements were reported by introducing splitter blades [1,2], tandem blades [3], three-dimensional impeller design ??4, ??]. Elder and Gill ??6] showed that the parameters viz., inducer incidence, impeller back-sweep angle, number of impeller and diffuser vanes have the significant effect on the compressor stability limit. Hildebrandt and Genrup ??7] investigated the influence of different back sweep angles and exit widths on the impeller outlet flow pattern of a centrifugal compressor with vaneless diffuser through numerical simulations. It was revealed from their studies that the impeller with increased back sweep provides more uniform flow pattern at the outlet that would provide better diffusion process in the downstream.

Further, Moore et al. ??8] [9] investigations realized that the impeller with 0° lean (also, defined as Skew in some literature) at outlet realized vast secondary flow losses near shroud region than in any other interior zone. To suppress these wakes, introduced a lean concept that allows redistribution of flow and moving the high loss fluid from shroud region to hub. By incorporating a lean at the exit indirectly affects the blade angle distribution as the tangency and profile smoothness are not to be disturbed as shown in Fig [1]. A patent by Harada and Shin [10] has disclosed the lean blade techniques. He claimed that the compressor performance, including efficiency, can be improved by imparting tangential lean to the impeller blades. Based on the above findings, it reveals that the overall performance and stability margin of centrifugal compressors can be improved by introducing negative lean at the impeller blade exit. Arunachalam and Nagpurwala [11] have given better approach in understanding the positive and negative lean affect on centrifugal impeller performance.

Figure [1]: Blade angle distribution from hub to shroud

[14]

Another study from, Howard et al.

[12] has also reported a study with the simple lean angle at impeller inlet and outlet as shown in Fig [2]. The numerical results revealed a marginal improvement in impeller efficiency with reduced leakage flow at the shroud, while it lowered the impeller head rise. It was also noticed that lean angle can control the distribution of static pressure along the blade height, especially in the rear part of the cascade which not only reduces the energy losses in the impeller passage but also improves the downstream flow.

Based on the author's observations, impeller with negative lean identified a decent stall margin improvement compared to 0°. Investigations by analysts performed on a transonic impeller with positive inlet lean realized improved operating range with increased stability margin and acceptable pressure ratio while, negative inlet lean at the shroud provided the worst performance. [13] have already provided an understanding on change in the Centrifugal impeller performance, it was for a transonic inlet conditions. Moreover, the investigations so far contributed was on combination of positive lean at inlet and outlet or a negative lean at inlet and outlet with a transonic flow conditions. our current research is herewith intended to design a subsonic impeller that has combination of positive inlet lean and negative exit lean to provide enough surge margin for steady state operation throughout the operating range. Henceforth, it is proposed to develop an impeller model with the combination of 10° inlet positive lean and 45° negative lean at the blade exit. These magnitudes are based on the authors investigations performed in improving stability margin. Impeller performance with detailed flow passage investigation is carried out and compared with the other two models viz., an impeller with No lean and an impeller with exit lean.

An Eckardt-A type, 1976-80 [15] [16] impeller operating at a design speed 14000 rpm with a mass flow at 4.52 kg/s and total pressure ratio of 1.92 is chosen as BASELINE for our current study. This impeller has 30° backswept with 0° lean. To validate the tested impeller, basic sizing of an impeller is obtained by performing 2D analysis using well-known Jet-wake theory proposed by Japikse. With the basic geometry details obtained and with the available blade geometry information including blade profile and wrap angle distributions As the lean is obtained by moving the shroud section relative to the tangential direction, will not disturb the existing blade profiles and hence minor changes required to be imposed on deriving Models A and B. 13.0. It is a Commercial CFD code capable to solve 3D compressible Navier-Stokes equations using a finite volume discretization method. It uses a range of turbulent models with both logarithmic wall function and two-layer approaches to model the boundary layer. A 3D geometric model of the radial compressor stage has been developed using ANSYS Blade-Gen. Model created on Blade-Gen platform is imported into ANSYS Turbo Grid for grid generation. For the current study, a single passage impeller with 5% radial space is simulated with low solidity vaned diffuser (LSVD) configuration at the downstream. A 3Dmesh has been developed using the H-Grid and O-Grid topologies. The O-Grid provides a good mesh around the blades while rest of the passage used H-Grid. Adequate grid is developed with enough number of cells to capture the complex flow phenomena like boundary layers, flow separation, leakage flows and secondary vortices in the blade passage. Grid independence study was carried out to get the optimum grid for numerical solution to be independent of grid. Computational grid with 4, 43,000 elements were generated and the CFD simulations were performed.

2 Numerical Analysis Setup

After the successful grid complexion completion of every individual model, numerical analysis is planned to setup for all these configurations. The inlet of the computational domain is kept 200mm ahead of the eye of the impeller to ensure that the inlet boundary conditions are not affected by the back pressure of the impeller blade. Ambient states are defined at impeller inlet duct with zero Pascal as relative pressure and total temperature of 288 K. The fluid used for simulation here is air and assumed to be an ideal gas.

K-? turbulence model is preferred with "Stage" interface for rotor-stator interaction with 5% of turbulence intensity. Considering a single passage configuration, the computational domains are separated into inlet and impeller domains. The side walls of the impeller domain are specified with rotational periodicity. At the impeller solid walls, no-slip boundary condition is applied and all the solid walls are assumed to be adiabatic. The hub wall of the impeller is presumed to be moving with the rotor blade, while the upstream and downstream hub is made stationary. The impeller shroud is defined as counter-rotating that allows the relative motion between the rotating impeller. As mentioned earlier, Inlet boundary conditions with total pressure and static pressure distribution at the outlet are applied. The fluid time step is given as 0.1/?, where ? is the angular velocity in radian per second. Throughout the calculation the values of maximum residuals, a pressure at the outlet, eventually mass flow imbalances and efficiency were monitored. The convergence criterion is set for the maximum residuals below 10-4. Applying the following boundary conditions, a Numerical analysis is performed by assuming the flow conditions as axisymmetric with periodic conditions imposed. Concrete solution is obtained by analyzing the flow through a single blade impeller passage for all these three models.

V. The other factor for the performance deviation can be due to the aerodynamic influence of Impellerdiffuser interaction. Concluding that the performance divergence between numerical model and experimental values are acceptable at the design point, the off-design performance trend was also satisfactory. After the successful validation of BASELINE MODEL, the improvement techniques revealed from authors knowledge ??11] [12] are applied to investigate the aerodynamic behavior on MODEL A and MODEL B for comparing the performance results. Observations reveal that by imparting lean on the MODEL A, impeller exit has noticed a considerable reduction in total pressure losses with an improved stability, while with added inlet lean to the MODEL B impeller has shown the additional increase in stall margin with no significant loss in total pressure. Performance plots are generated for our discussion herewith. When inlet lean is provided to MODEL B in addition to the exit lean, the variation of area from Hub to shroud at inducer influence on the inlet blade angle variation and hence on the relative flow diffusion along the blade passage. The consequence of inlet lean feature resulted to further increase in relative velocity (reduction in tangential velocity) at outlet, which is observed on total pressure rise and appreciable increase in stall margin over MODEL A. Fig ??12] and [13] substantiate the above discussions. Aerodynamic flow studies at impeller exit on all the three models are relatively compared at design and two off-design conditions. At design point, the highest relative Mach number is observed with MODEL B Impeller over Baseline and MODEL A impellers. This indicates the increase in relative Mach number reduces the tangential velocities (energy transfer) and hence decreases the static pressure rise as evident from the flow studies.

3 Numerical Model Validation

The calculated efficiencies obtained using numerical model shown in ??ig [8] is always unpredictable and complex. However, when these magnitudes are compared looks to be acceptable as the pressure losses are function of impeller blade loading. Higher work-input coefficient with baseline impeller obviously envisages lower total pressure losses in relative to other two configurations. Following are the performance parameter observations tabulated below at design and Off-design conditions for three different impeller models. Readers are to be noted herewith, the performance values for Baseline Impeller at = 0.95 is not reported as the impeller underwent the surge occurrence near to design point condition.

4 VII. Flow Studies Through Centrifugal Impeller

From the numerical simulations, stall was detected in the diffuser ring as flow break down in between the passage with decrease in stage flow coefficient. Overload conditions are identified by flow break down in the diffuser as the flow coefficient is increased, while the impeller pressure ratio drops to exceedingly low values. No solutions were pursued beyond these limits and the below discussion is within these boundaries to examine the flow behavior.

5 a) Flow behavior through Meridional plane

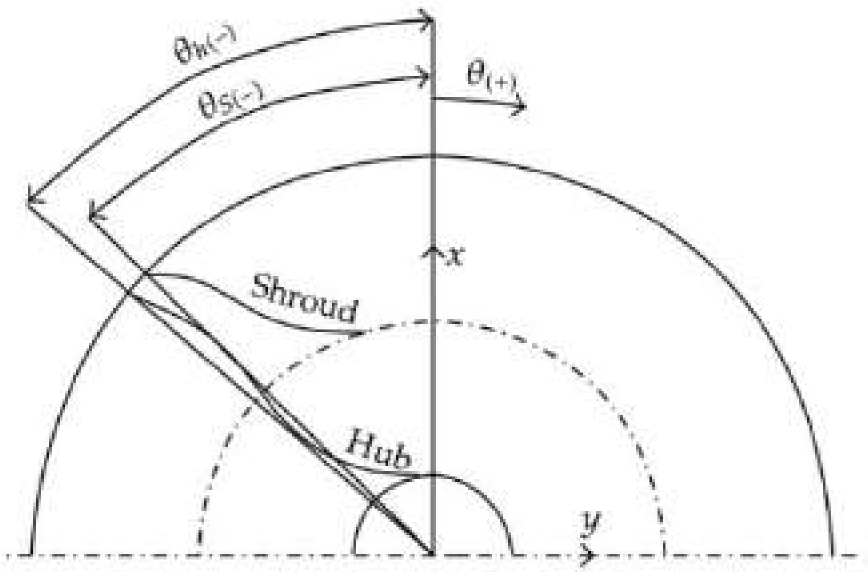
The flow field of the impeller without inlet lean (MODEL A) and impeller with compounded effect of inlet and outlet lean (MODEL B), are examined through relative Mach number distribution in meridional planes at three different flow coefficients. The purpose is to find ? possible explanations for the effect of inducer leading edge on the performance of the compressor. It should be understood that the change in relative Mach number along the impeller meridional passage indicates the Global Journal of Researches in Engineering ( ) Volume XVII Issue II Version I growth in diffusion process. Higher the diffusion rate, higher will be the pressure rise.

The progressive diffusion along the passage for MODEL B impeller is slow when compared with the Baseline and MODEL A impellers as evident from ??ig [9]. This indicates the MODEL B impeller has a lower decay rate of relative Mach number distribution along the meridional passage. Accordingly, this flow phenomenon is visualized with lower pressure rise with MODEL B impeller when compared to rest of the other impeller models as realized from Fig ??10]. The static pressure rise at different flow coefficients envisaged the influence of relative Mach number distribution growth. The inlet relative Mach number distribution has no significant change between the three impeller models as evident from Fig ??11]. With inlet lean on MODEL B impeller, the inducer inlet has variation of area from Hub to shroud that required the change in inlet blade angle variation. This feature is believed to influence the diffusion control along the impeller passage.

6 b) Relative Mach number envelops and Static pressure distribution at impeller outlet

The relative Mach number and static pressure distribution at impeller outlet is understood implicitly with the change in static pressure imposed at diffuser outlet. Flow phenomenon observed for the baseline impeller follow a similar trend as noticed by author Krain H. **??**1984]. Further, the provision of inlet lean at impeller inlet (MODEL B), the compound benefit is clearly verified from the flow dynamics showing higher relative Mach number at impeller exit for design and off-design conditions while, the Baseline impeller demonstrate the lowest relative Mach number. This phenomenon reflects on the pressure rise development for the respective impeller models as shown in Fig **??**13]. Similar interpretation can be made or judged based on realized relative Mach number at impeller exit. combinations on the impeller performance and stall margin. The results of the impeller are agreeable when looked onto the performance parameters like Work-input coefficient, Total pressure ratio and stall margin. The outcome of these parameters realized using MODEL B is comparable with BASELINE and MODEL A impeller except the computational values obtained for polytropic efficiencies. Although the commutated efficiencies are overestimated throughout the operating range by various factors, the conclusions drawn are as follows:

? MODEL A impeller reveals an improved stall margin by 3.1% over baseline impeller, while MODEL B envisage further better stall margin by 7.4% compared with Baseline impeller. [1] [2]



2

Figure 1: Figure [2]D

[1] © 2017 Global Journals Inc. (US)

[2] Numerical Studies on Centrifugal Impeller Performance with Different Lean Combinations © 2017 Journals Inc. (US)

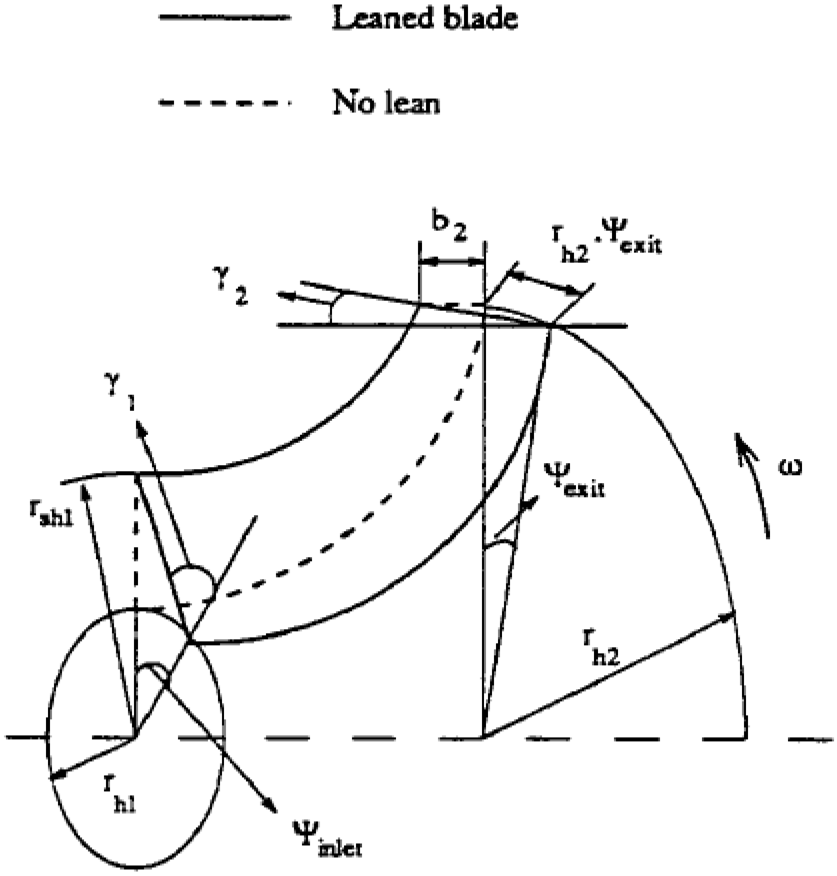


Figure 2:

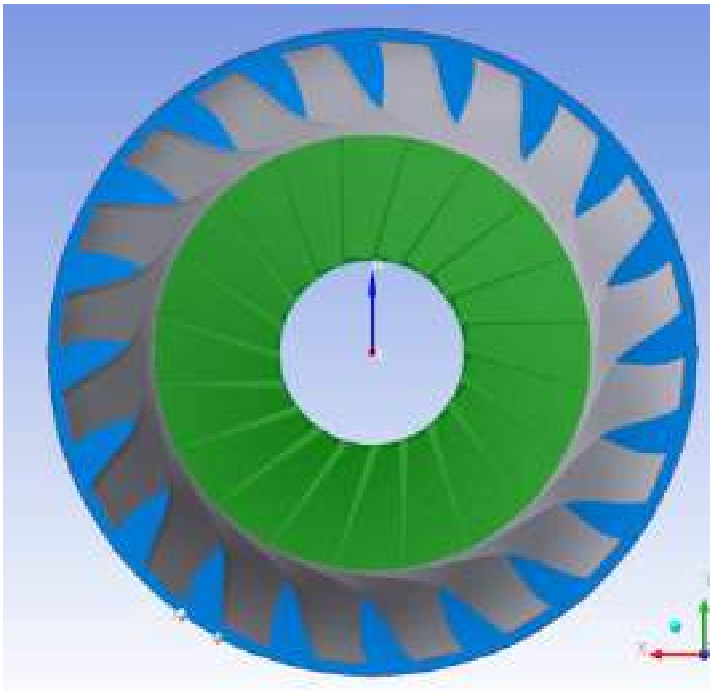


Figure 3: Fig

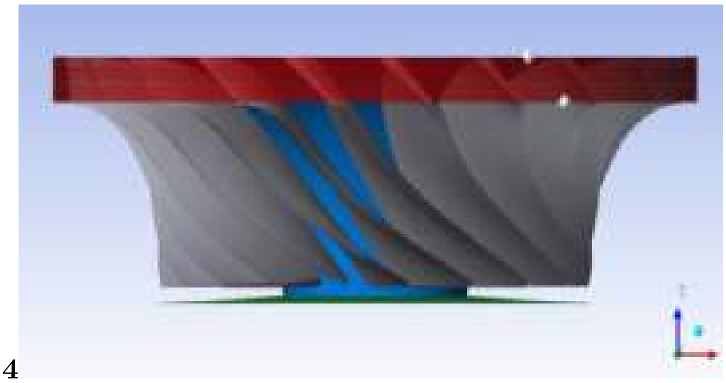
4 

Figure 4: Fig. [4

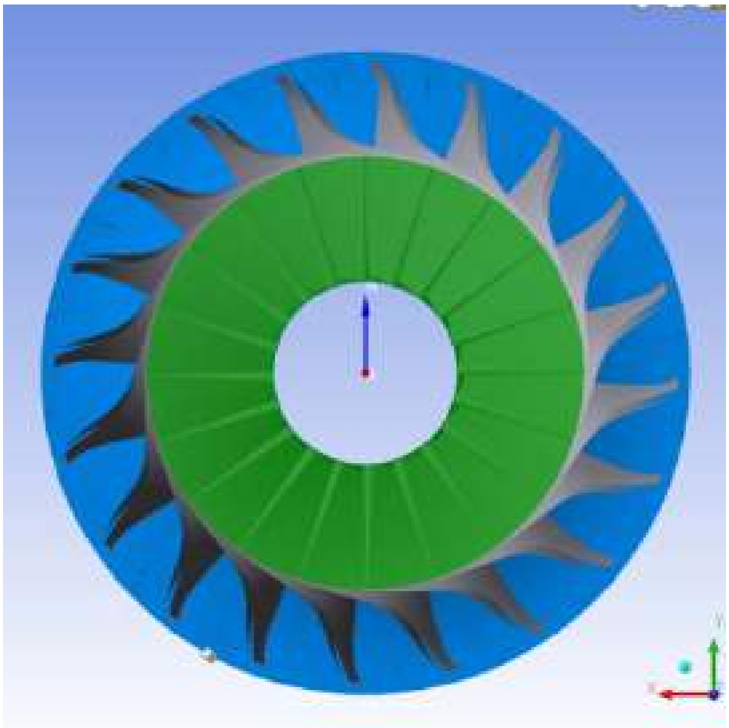


Figure 5:

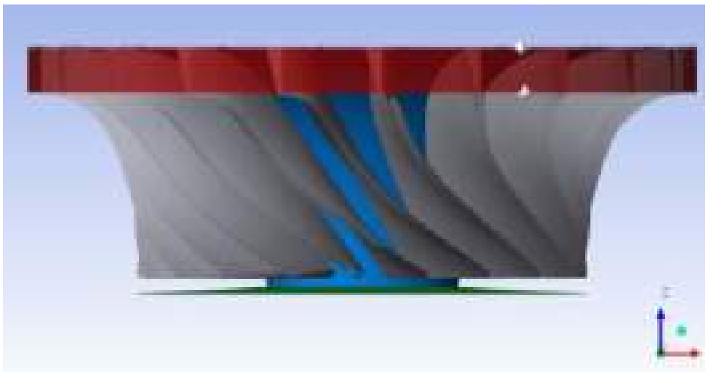


Figure 6: (

8

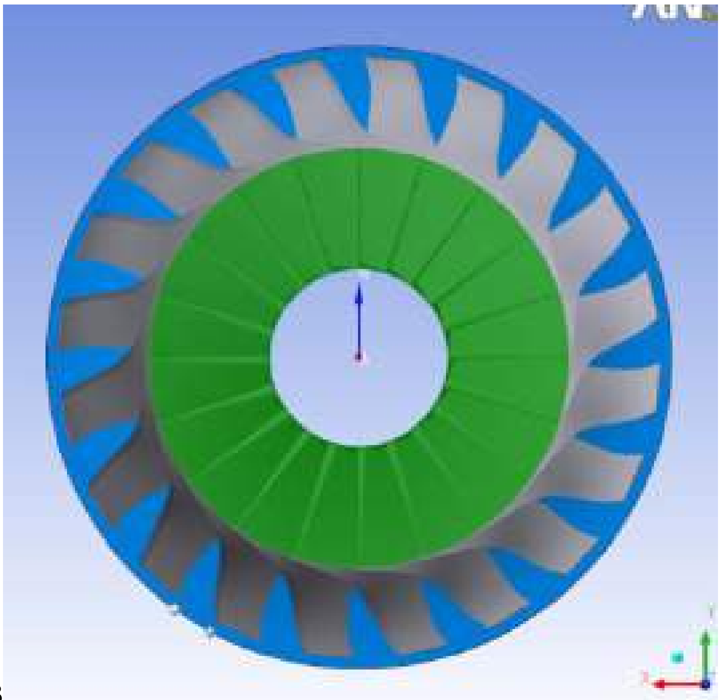


Figure 7: Fig. [8

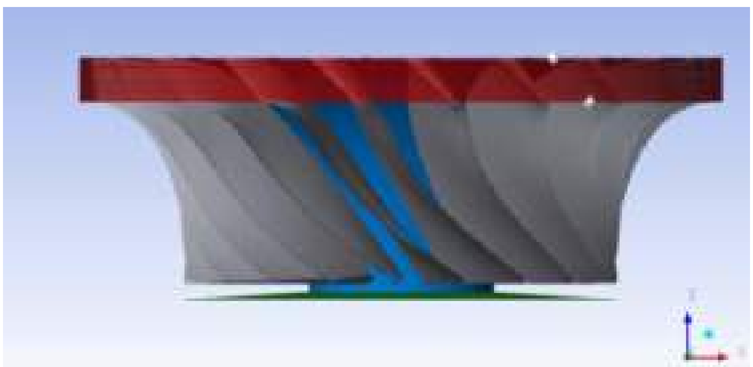


Figure 8:

Impeller Type	Parameter	Φ = 0.95	Φ = 1.00	Φ = 1.10
Baseline	Mass flow (kg/s)		4.48	4.89
	Pressure Ratio		1.92	1.91
	Head Coeff.		0.65	0.64
	Impeller Efficiency		93.00	93.15
Model A	Mass flow (kg/s)	4.34	4.56	4.85
	Pressure Ratio	1.88	1.87	1.85
	Head Coeff.	0.63	0.63	0.61
	Impeller Efficiency	93.5	93.5	92.3
Model B	Mass flow (kg/s)	4.15	4.54	4.90
	Pressure Ratio	1.87	1.85	1.82
	Head Coeff.	0.63	0.62	0.59
	Impeller Efficiency	92.70	92.30	92.00

91011112

Figure 9: Fig. 9 :Fig. 10 : 1 Fig. 11 :Fig. 12 :

? Numerical model realizes narrow operating range than Eckardt impeller. It is fairly expected as the Test impeller has VLD configuration in the downstream that has no influence on incidence effect in off-design condition, since the fluid follows a logarithmic spiral path.
? Eckardt impeller reported 88.6% gain in isentropic efficiency and total pressure ratio of 1.90 with design mass flow of 4.54 kg/s at design speed. The test data obtained with vaneless diffuser configuration for Eckardt makes sense when compared with the numerical model, as the VLD encounters higher pressure losses over LSVD configuration.

Figure 10:

13. Xiao He and Xinqian Zheng, M echanism of lean on the performance of transonic centrifugal compressor

Centrifugal Compressor Impeller Using Evolutionary Algorithms” Hindawi Publishing Corporation Mathematical Problems in Engineering Volume 2012, Article ID 752931, 22 pages doi:10.1155/2012/752931
15. Eckardt, D. Flow field analysis of radial and backswept centrifugal compressor impellers. Part I:

Figure 11:

.1 Conclusions

Numerical investigations have been performed to determine the effect of the inlet and exit lean

[Larosilierel et al. ()] 'Aerodynamic Synthesis of Centrifugal Impeller using CFD and Measurements'. M Larosilierel , G J Skoch , P S Prahst . ARL-TR-1461. *33rd Joint Propulsion Conference and Exhibit AIAA-97-2878*, 1997. (Technical Report)

[Higashimori et al. ()] 'Detailed Flow Study of Mach Number 1.6 High Transonic Flow with Shock Wave in a Pressure Ratio 11 Centrifugal Compressor Impeller'. Hirotaka Higashimori , Kiyoshi Hasagawa , Kunio Sumida , Tooru Suita . *Trans. ASME Journal of Turbomachinery* 2004. 126 p. .