

GT2011-46505

Optimization of 6.2:1 Pressure Ratio Centrifugal Compressor Impeller by 3D Inverse Design

M. Zangeneh¹
University College of London
London, UK

N. Amarel , K. Daneshkhah
Advanced Design Technology
London, UK

H. Krain
DLR, Köln²
Germany

ABSTRACT

In this work, the redesign of a centrifugal transonic compressor impeller with splitter blades by means of the three-dimensional inverse design code TURBObdesign-1 is presented.

The basic design methodology for impellers with splitter blades is outlined and is applied in a systematic way to improve the aero/mechanical performance of a transonic 6.2:1 pressure ratio centrifugal compressor impeller. The primary design variables are the main and splitter blades loading and their thickness distributions, the splitter to main blade work ratio, as well as the span-wise swirl distribution. The flow in the original and redesigned impellers are then analyzed by means of a commercial CFD code (ANSYS CFX). The predicted flow field for the original impeller is compared with detailed L2F measurements inside and outside the impeller. The validated CFD results are used to compare the flow field in the optimized and original impeller. It is shown that the inverse design method could be effectively used to control the position and strength of the shock waves, eliminate flow separation and hence obtain a more uniform impeller exit flow in order to improve the aerodynamic performance. In addition, some results are presented on the comparison of stress and vibration in both impellers.

NOMENCLATURE

deg	Degree
EFF	Total-Total isentropic Efficiency
LE	Leading edge
TE	Trailing edge
Main	Main (full) blade
M_{rel}	relative Mach number
PR	Pressure ratio
PS	pressure side
Splitter	splitter blade

SS	suction side
m	mass flow rate (kg/s)
n	shaft speed (1/min)
n_0	design shaft speed (1/min)
r	radius
u	local circumferential velocity (m/s)

Subscripts

m	normalized meridional distance
red	corrected to standard conditions
rel	relative
tt	total/total
1	rotor inlet
2	rotor exit
3	diffuser inlet
4	diffuser exit

INTRODUCTION

Transonic centrifugal compressors with pressure ratios higher than 5:1 are commonly used in marine diesel turbocharger applications as well as in helicopter engines and other gas turbine applications. These compressors, which tend to operate at relatively high tip speeds of 500-600 m/s, need to provide high efficiency over a broad range of operation as well as being reliable structurally.

In recent years there has been a number of studies of the flow field in these types of impellers by using detailed LDV and/or L2F (Laser-Two-Focus) measurements, see Eisenlohr, et al. (2002) and Ibaraki et al (2003). Such measurements coupled with recent numerical analysis of the flow through these impellers, see Marconcini et al (2008) and Ibaraki, et al. (2008) have helped to explain the nature of the flow inside these

¹ Corresponding author m.zangeneh@ucl.ac.uk

² Currently retired

impellers. The flow seems to be dominated by strong 3D interaction of inducer shock with tip clearance flow, secondary flows and possible incidence effects on the splitter blade leading edge. Despite the complexity of the flow field, these impellers are generally designed by using straight filaments, partly due to requirements for flank milling of impellers and partly due to perceived possible structural integrity issues with 3D blade profiles. However, in recent years many centrifugal compressor manufacturers, especially in turbocharger and helicopter engine applications, have been changing their manufacturing system to point milling and have found that the extra cost of point milling can be only 10-20% more than equivalent flank milling process, Thiele (2000). Furthermore, it has become increasingly clear that further performance improvement is only possible by using 3D blade profiles, especially for the impeller, see Zangeneh et al (2010).

Robust and reliable design of impellers with 3D geometry also poses difficulties for designers. The 'direct' design approach based on iterative modification of the blade shape results in a time consuming design process, especially if various multi-objective requirements of the impeller design for turbocharger applications are to be met. These difficulties are compounded for impellers with splitter blades, which are commonly used in turbocharger applications.

In conventional design practice, it is customary to use the same blade angle on the splitter blade as on the main blades with splitter camber line being placed at mid-pitch between the two main blades. There is, however, considerable evidence that this practice leads to poor performance of the splitter blade as a result of the mis-match between the splitter leading edge blade angle and the local flow angles.

In recent years there have been some attempts by designers to modify the splitter leading edge slightly to improve the matching of the splitter to the local flow. An example of this is the work of Drtina et al (1993), who showed that by modifying the pitchwise location of the leading edge of a splitter blade they could improve the pressure recovery of a vaned diffuser. However, in the design of 3D blades it is rather difficult to arrive at an optimum pitchwise distribution of the splitter across the span.

An alternative approach for the design of centrifugal impellers with 3D geometry is to use a 3D inverse design method such as TURBObdesign1(2009). In this inverse design method the blade geometry is computed for a specified distribution of blade loading ($\partial r \bar{V}_\theta / \partial m$), which is the meridional derivative of the tangentially mean swirl velocity and is directly related to the blade bound circulation $2\pi r \bar{V}_\theta$. In this method, in addition to the blade loading the normal thickness distribution is specified. The basic theory behind this approach is also presented in Zangeneh (1991) and its extension to design of blades with splitters is presented in Zangeneh (1998). The inverse design method TURBObdesign1 allows the designer to design the blade geometry on the full and splitter blades subject to a specified distribution of blade loading and

therefore automatically optimizes the pitchwise location of the splitter blade across the span.

The method has already been applied to the design of industrial centrifugal compressor impellers, where simple design guide-lines have been developed for suppression of secondary flows and the resulting jet/wake flow effects at the impeller exit (see Zangeneh et al, 1998 and 1999). This method has also been applied to the design of compact high performance vaned diffusers for centrifugal compressors (see Zangeneh et al, 2002) and industrial compressors with splitters, see Zangeneh et al (2004) and Schleer et al (2004) and turbochargers for heavy duty diesel applications, see Zangeneh et al (2010).

In this paper, this inverse design approach is applied to the redesign of a 586 m/s tip speed impeller which has been the subject of extensive detailed experimental measurements and numerical analysis at DLR, see Krain and Hofmann (1998), Eisenloher et al (2002) and Krain et al (2007).

BASELINE IMPELLER

The baseline impeller used in this study is the SRV2AB impeller that has been subject of detailed L2F (Laser-Two-Focus) measurements and flow predictions at DLR Cologne, see Krain and Hoffman (1998) and Eisenlohr et al (2002). The impeller has been tested both with vaneless (SRV2ABunb) and vaned diffuser (SRV2ABD1) configurations. The basic rotor design parameters are given in table 1.

Shaft speed	n=50000 rpm
Design Mass flow rate	m=2.55 kg/s
Impeller Tip radius	r ₂ =112 mm
Rotor Tip speed	u ₂ =586 m/s
Rotor pressure ratio	P _{t12} =6.2:1
Blade count full/splitter	13/13
LE hub radius	30 mm
LE tip radius	78 mm
Blade angle LE tip	26,5 deg.
Blade angle TE	52 deg.
Exit blade height	8.7 mm
Diffuser inclination against radial	13 deg.
Tip clearance	0.5 mm at inlet to 0.3 mm at exit
Material	Titanium Alloy TiV6A14V

Table 1: Rotor Design Parameters

L2F measurements were made at 50,000 rpm and design flow rate of 2.55 kg/s at locations shown in Figure 1.

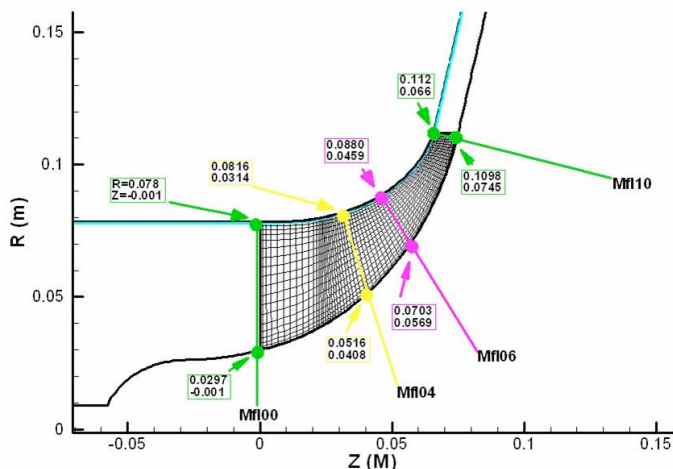


Fig. 1 Location of L2F measurement planes for Baseline Impeller.

Prediction of flow in Baseline impeller and comparison with L2F measurements

Steady 3D calculations were carried out for the original SRV2AB impeller using the same vaneless diffuser as was used in the experiments. ANSYS CFX11 was used for this study. The mesh used for the final computations were arrived at after a mesh independency study using a coarse, fine and very fine mesh. The coarse mesh used about 300 thousands nodes for one blade channel bounded in the circumferential direction by two main blades, medium mesh used 880 thousands nodes and the fine mesh used 1.4 million nodes. The effect of using different meshes for the prediction of the stage characteristic is shown in Figure 2. The medium mesh seems to provide a mesh independent solution for the whole characteristic and hence this mesh was used for the rest of the study. The medium mesh consists of 235 streamwise, 49 spanwise and 76 pitchwise nodes, and is shown in Fig. 3.

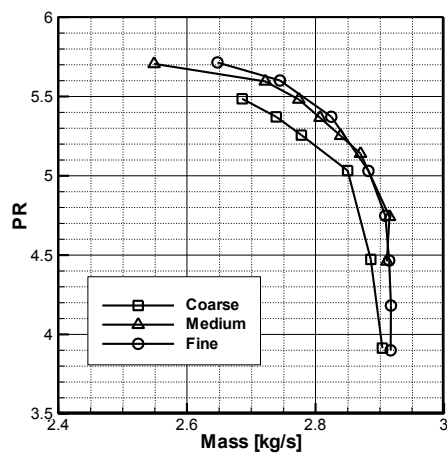


Fig. 2- Mesh independency study, $n = 50\,000$ rpm

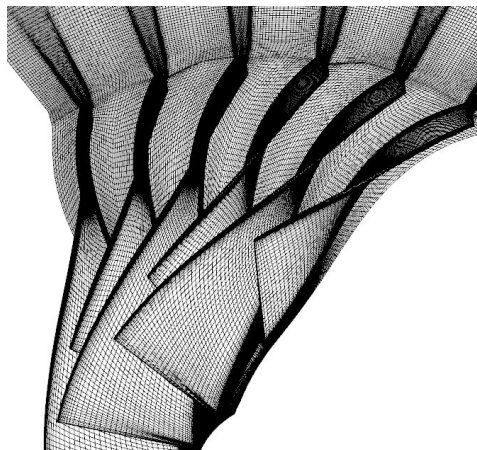


Fig. 3 Computational Mesh used

The computational domain consists of an unshrouded impeller followed by a vaneless diffuser. A frozen-rotor interface is used between the rotating and stationary components. The mesh has a structured H-O topology. For flow analysis compressible RANS equations were used. Both $k-\epsilon$ and SST turbulence models were used initially at design point and design speed and since no significant differences were observed the $k-\epsilon$ model was for rest of the computations. In general the value of Y^+ was around 40-50 and was everywhere less than 100.

Figure 4 shows a comparison of predicted pressure ratio versus mass flow characteristics of the stage consisting of the impeller and vaneless diffuser at 50,000, 40,000 and 30,000 rpm, corresponding to 100%, 80% and 60% of design speed. Generally the prediction matches the experimentally measured performance quite well at all operating speeds.

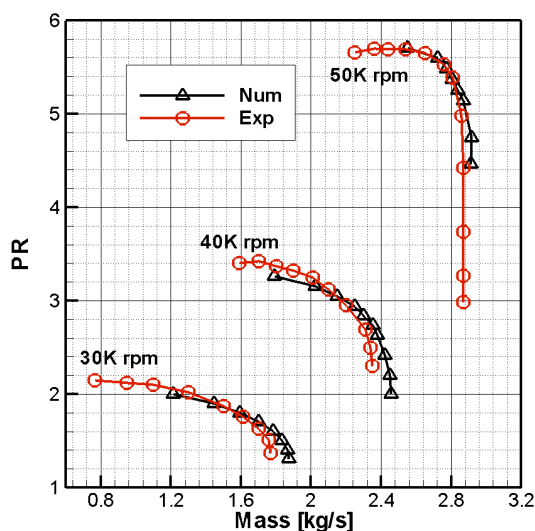
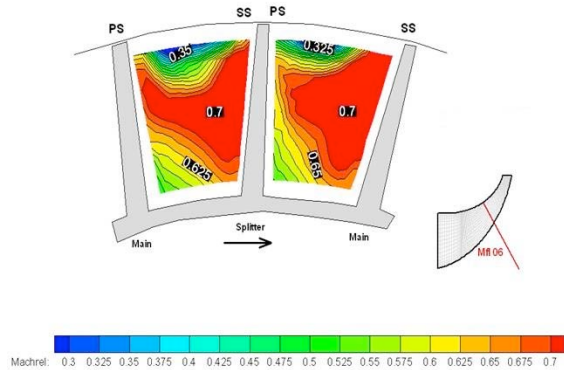
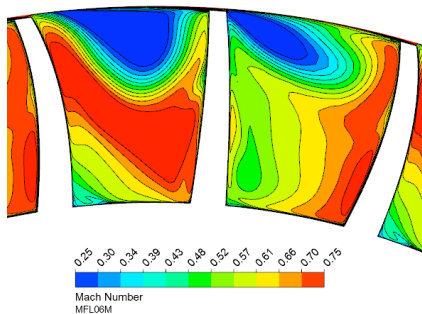


Fig. 4 Comparison of predicted and measured total/total stage pressure ratio versus corrected mass flow

The CFD predictions at the 100% speed and design flow rate of 2.55 kg/s were compared with the L2F measurements at the two most critical measurement planes MFL06 and MFL10 (Fig.1). At MFL06 the flow is turned from axial to radial which is usually related with the onset of wake flow close to the shroud. The jet/wake structure at rotor exit (MFL10) determines the unsteady diffuser inlet flow in the absolute frame. Both effects are fairly well predicted by the CFD approach and acceptably good correlations were found between the CFD predictions and the L2F measurements at these locations. The results are shown in Figures 5 and 6 below. The L2F measurements could not be made very close to endwalls and blade surfaces. But the data in Figure 5 shows that the extent of the wake region in both channels are well predicted by CFD and compare quite well with the L2F measurements. The same observation can be made at the impeller exit plane as shown in Figure 6. These results confirm the accuracy of the CFD code used in this study to predict the complex flow field inside high pressure ratio transonic compressors. The same type of mesh topology and CFD setting will then be used to study the flow in the redesigned inverse designed impeller.

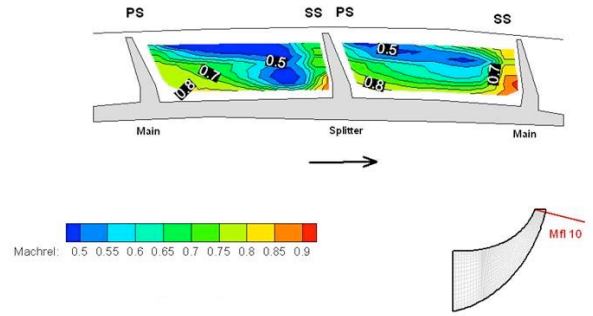


a) L2F Measurements at location Mf106

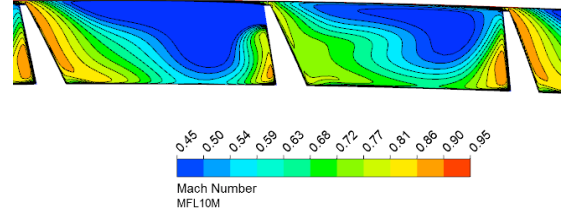


b) CFD Predictions at Location Mf106

Fig. 5 Comparison of predicted and measured relative Mach number for Baseline impeller at location Mf106



a) L2F measurement near TE (Mf110)



b) CFD predictions at Location Mf110

Fig. 6 Comparison of predicted and measured relative Mach number for Baseline impeller at Location Mf110

FLOW IN BASELINE IMPELLER AND DESIGN OBJECTIVES

The SRV2AB impeller has been the subject of a redesign study by Krain et al (2007). In this redesign study a number of parameters such as meridional shape, axial impeller shape, wrap angle at rotor exit and blade angle distribution and diameters at hub and shroud were varied in order to improve the exit flow non-uniformity. All designs had straight filament geometry. The optimum design of this study was labeled as SRV4 impeller and its performance was evaluated experimentally. The new impeller was found to improve the stage performance at design speed by about 2 points but at part load conditions the performance of the new impeller was less than the original SRV2AB impeller, see Krain et al (2007) for further details. The results presented for SRV4 impeller in Krain et al (2007) are for tests with vanned diffuser. However, additional tests were also done with vaneless diffuser (not presented in the paper) which essentially confirmed the same drop in performance of the SRV4 impeller at part load conditions.

The main objective of the present study is to redesign the SRV2AB impeller to improve its efficiency at both design and part-load speed conditions. Another important aspect of this study is to investigate the advantages and any possible disadvantage in terms of aerodynamics or structural integrity in using 3D impeller geometries for these high tip speed centrifugal compressor applications.

In order to achieve these objectives the first step is to understand better the nature of the flow in the baseline SRV2AB impeller and identify the main causes of loss. In previous section, it was shown that the CFD predictions in the baseline impeller matches well the L2F measurements. So the

CFD results can be used to investigate the main sources of loss in the baseline impeller. In Figs. 7 and 8, the blade to blade relative Mach number contours in the original SRV2AB impeller is shown at 50% and 95% span for the design point ($n=50\,000\text{rpm}$ $m=2.55\text{ kg/s}$).

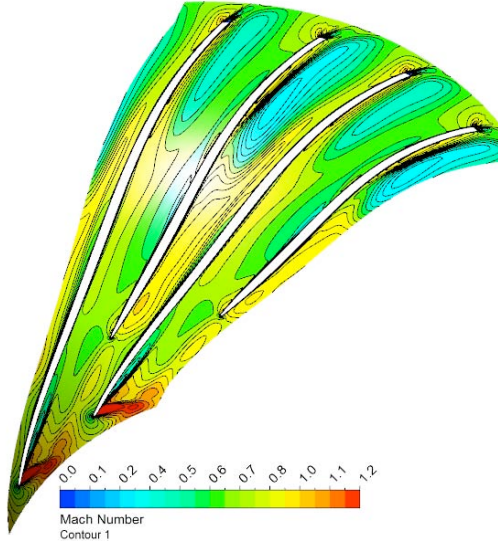


Fig. 7 – Relative Mach number Contours at Mid-span

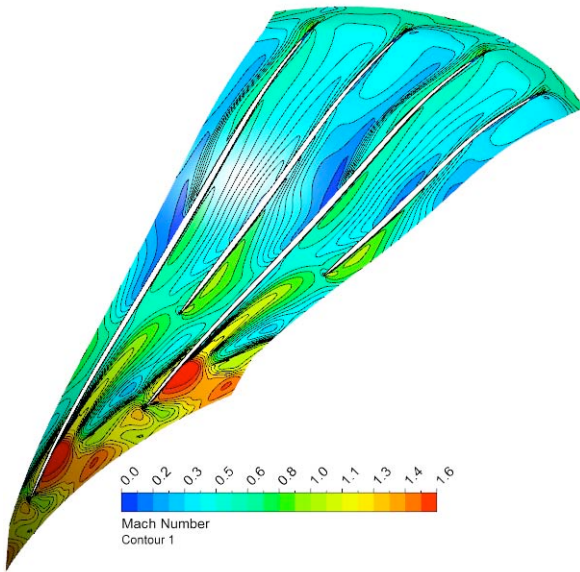


Fig. 8 – Relative Mach number contours at 95% span

The predictions indicate that the flow in the impeller is to a large extent dominated by the inducer shock, which extends from the tip down to mid-span position. At the 95% span maximum Mach number on suction surface reaches 1.6. The shock seems to be interacting strongly with the clearance vortex. Also a region of almost stagnant flow can be observed on the pressure surface of the full blade. Velocity vectors near

the suction and pressure surface (inside the boundary layer) of main blade, see Fig. 9, show an area of flow separation near the tip on pressure surface and strong spanwise secondary flows on the suction surface, especially in the inducer. Also a small area of flow separation can be observed on pressure surface of the splitter blade (not shown).

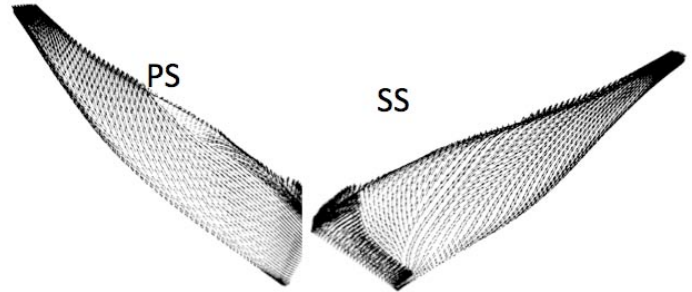


Fig. 9 – Velocity Vector on the pressure and suction surface of main blade

The results indicate that the losses in the impeller are dominated by the presence of secondary flows, strong shock in the inducer and its interaction with the leakage flow as well as flow separation on the pressure surface of the main blade and to a lesser extent splitter blade. The main aim of the redesign is to reduce the inducer shock strength and eliminate the separation on the pressure surface.

REDESIGN OF IMPELLER BY 3D INVERSE DESIGN

The 3D inverse design method TURBODesign1 allows the designer to design the blade geometry on the full and splitter blades subject to a specified distribution of blade loading. The code takes as input the basic design parameters such as rpm, mass flow rate and specific work together with a specified choice of meridional shape and normal thickness. In TURBODesign1 it is possible to modify the meridional shape during the design process but in this project no attempt was made to modify the meridional shape. The main aim was to investigate the effect of blade loading on the impeller flow field. So the same meridional geometry as that of the baseline SRV2AB impeller was used in the design. The meridional geometry used is shown in Fig.1. The splitter leading edge location is typical of high tip speed applications where the splitter is almost radial.

The choice of design point, in terms of operating speed and flow rate has an important effect on part-load performance. In this case, an important requirement was to ensure good performance at most operating conditions including part load speeds. Hence it was decided to base the design point at near peak efficiency of the conventional SRV2AB impeller at 44,000rpm or 88% of nominal speed. The CFD predictions of baseline impeller at this speed were used to obtain the specific work and hence the impeller exit $r\bar{V}_\theta$.

The spanwise distributions of impeller inlet and exit $r\bar{V}_\theta$ are important design inputs in the inverse design code. In

TURBODesign1, as the blade loading on the full and splitter blades are specified separately, the overall impeller $r\bar{V}_\theta$ is split between the full and splitter blades through the specification of the $r\bar{V}_\theta$ ratio which is defined as:

$$r\bar{V}_\theta \text{ ratio} = \Delta(r\bar{V}_\theta)^{\text{splitter}} / \Delta(r\bar{V}_\theta)^{\text{total}}$$

where $\Delta(r\bar{V}_\theta)^{\text{splitter}}$ is the change in specific work on the splitter blade and $\Delta(r\bar{V}_\theta)^{\text{total}}$ is the change in specific work in the overall impeller. Normally the $r\bar{V}_\theta$ ratio is based on the ratio of the meridional length of splitter (at the shroud) over the overall meridional length. This type of definition of $r\bar{V}_\theta$ ratio will essentially ensure similar levels of maximum loading between the full and splitter blades. However, it is equally possible to unload the splitter at the expense of full blades or vice versa. This provides additional flexibility in the design of impellers with splitter blades. In this case, initially the $r\bar{V}_\theta$ ratio was set at 0.43, which is the value of based on meridional length at the shroud. However, by reducing this ratio down to 0.4 at the hub and shroud the performance of the impeller was found to improve.

The other important design specification is the blade loading ($\partial r\bar{V}_\theta / \partial m$). The blade loading is directly proportional to static pressure jump across the blade and hence can be used to control 3D pressure field in the impeller and influence the shock strength and tip leakage flow. The blade loading specified in the design of the impeller, with $r\bar{V}_\theta$ non-dimensionlised by the tip speed and tip radius, is shown in Fig. 10.

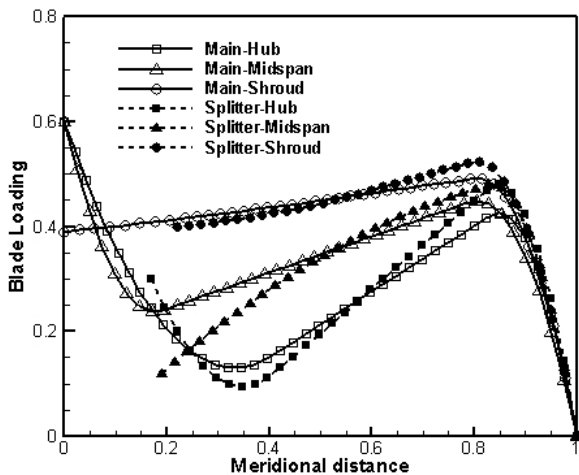


Fig. 10 Specified Blade loading

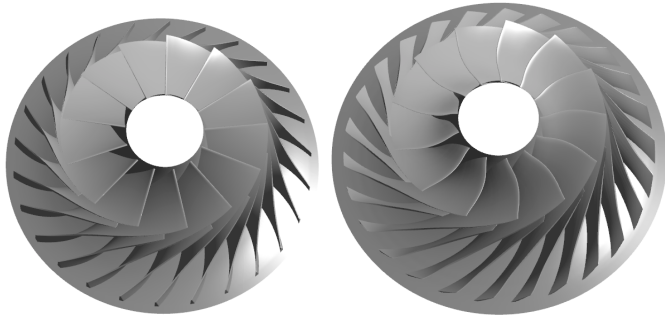
In Turbodesign1, it is possible to specify the blade loading on the full and splitter blades separately. In this case the blade

loading was specified at 3 different streamlines at hub, mid-span and shroud. The choice of blade loading is dictated by a number of contrasting requirements. Firstly, to minimize secondary flows on the suction surface it will be useful to have an aft-loaded distribution at the hub and fore-loaded distribution at the shroud, see Zangeneh et al (1998). However, to use a fore-loaded distribution at the shroud of the main blade will actually result in a very strong shock in the inducer and hence high inducer losses. So in fact in this case a moderately aft-loaded distribution was used at the shroud of the main blade and a strongly aft-loaded distribution at the hub. The high value of leading edge loading on the main blade especially at the hub and midspan helps to provide a wide operating range and a choke margin similar to the original impeller. Also this type of loading has an added benefit of reducing the blade lean at the leading edge of the main blade, which is very important for controlling the stresses in this high speed impeller. The rapid unloading at the hub and mid-span of the main blade for the first 20% of meridional chord helps to reduce the shock strength and in fact has a major impact on reducing the shock losses at the midspan area. The loading on splitter blade is generally made to follow that on the main blade apart from the region close to the leading edge, in order to ensure the resulting geometry is not too complex.

The final important input specification in inverse design code is the stacking condition, where the spanwise variation of wrap angles at one quasi-orthogonal are specified. The inverse design code uses the wrap angles specified on the stacking quasi-orthogonal as the initial condition of the hyperbolic partial differential equation used to compute the blade wrap angles. In this case the blades were stacked at the trailing edge plane with a wrap angle stacked in such a way that the shroud leads the hub in the direction of rotation. This type of stacking condition is similar to conventional rake and is used primarily to reduce the stresses at the impeller tip region. The impeller was designed with the same normal thickness on the full and splitter blade as the original SRV2 impeller. Using the above loading and stacking distribution the impeller geometry was designed by using TURBODesign1. Initially the surface Mach numbers from the code are used to check the diffusion on the blade surfaces and the blade loading was slightly adjusted to obtain reasonable blade surface diffusion. Also by comparing the throat area of the inverse designed impeller with the original SRV2 impeller, the leading edge loading, which has dominating effect on throat area, was modified to match the original impeller's throat area, to ensure the same choke margin. Once an acceptable diffusion is obtained and the throat area is matched, CFD computations were performed at 100%, 80% and 60% of design speed. By using feedback from CFD, only minor adjustments were made to the exit $r\bar{V}_\theta$ to adjust the pressure ratio back to the pressure level of the original SRV2 impeller.

The solid model of the resulting blade geometry computed by inverse design is compared to that of the conventional SRV2 impeller in Fig. 11. The inverse designed impeller has a 3D

geometry but with very little lean at the leading edge. In fact the difference in wrap angle between hub and shroud at leading edge of full blade is only 0.5 degrees. However, the leading edge is slightly bowed. The splitter leading edge has more lean (about 5 degrees difference in wrap angle from hub to shroud) and is also bowed.



a) Original Impeller b) Inverse Designed Impeller

Fig. 11 – Solid model of Original and inverse designed impeller

STRUCTURAL INTEGRITY OF INVERSE DESIGNED IMPELLER

The SRV2 impeller has a very high tip speed of 586 m/s at the rated design condition and hence it is essential to check the viability of any new design by confirming the structural integrity of the impeller. For this purpose the structural and modal analysis of the original SRV2 impeller and the inverse designed impeller were performed by using ANSYS structural. The blades and splitters were modelled separately, each using the same 10-node tetrahedral solid element and an overall element length of approximately 2–2.5 mm. The original SRV2 impeller was manufactured using an annealed Titanium Alloy, Ti6Al4V, which had a density, elastic modulus, yield stress, ultimate tensile strength and poisson's ratio of 4430 kg/m³, 113.8 GPa, 880 MPa, 950 MPa and 0.342³, respectively. This material was used for analysis of both original and inverse designed impeller. The models did not include the hub or shroud geometry, and were thus constrained at the hub surface of the blade. In order to obtain more reasonable stresses at the hub, a second model was also created for the redesigned blade which included a 3mm radius fillet at the hub surface with a tighter mesh size.

The stresses and deflections experienced due to a rotational speed of 50,000 rpm (5236 rad/s) were determined by using a rotating reference frame and including Coriolis and centrifugal forces. Figure 12 displays the Von Mises equivalent stress experienced by the main blade and splitter of both designs. The same contour interval was used for the main and splitter blade of both original and inverse impeller, respectively. The

extremely high stress at the main blade's leading edge hub is due to the model's lack of fillet; a second model with a 3mm radius fillet at the hub shows a 35% reduction in stress at this location. Due to the curvature of the leading edge in the inverse designed impeller, the real maximum stresses are experienced mid-span of the pressure side for both the blade and splitter. Overall, the stress of the main blade is comparable between the two designs.

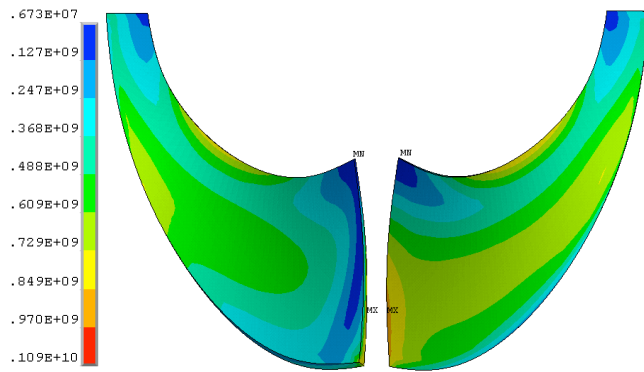
The splitter of the inverse designed impeller, however, does experience a 40% increase in stress near the leading edge due to the curvature. The high stresses along the hub are again due to the lack of fillet and would be significantly alleviated upon the inclusion of one. Despite the increase in stress, the splitter still experiences a lower maximum value than the blade, and for both the inverse designed impeller blade and splitter the predicted stress has a reasonable safety margin from the yield strength (880 MPa) of the material.

These structural results were then used to determine the modal response of the blades by specifying in ANSYS that the models were pre-stressed. This is a necessary step, as without the pre-stressed command the analysis would not incorporate the changes in blade shape or stress due to the rotation. Figure 13 displays the resultant displacements for the first four mode shapes of the Original case, which are nearly identical to those of inverse designed impeller with only a slight difference in amplitude (red represents high displacement and blue low displacement). The results show that the first two natural modes are bending, while the third and fourth modes are a combination of torsional and bending. Table 2 lists the calculated natural frequencies of the two designs, which indicates that the inverse designed impeller blade has higher natural frequencies and therefore is the stiffer blade. As mentioned earlier, the inverse designed impeller uses exactly the same normal blade thickness as the original SRV2 impeller. Clearly, by modifying the normal thickness distribution of the inverse designed impeller it should be possible to further improve the static and dynamic stresses. However, no such attempt was made in this study.

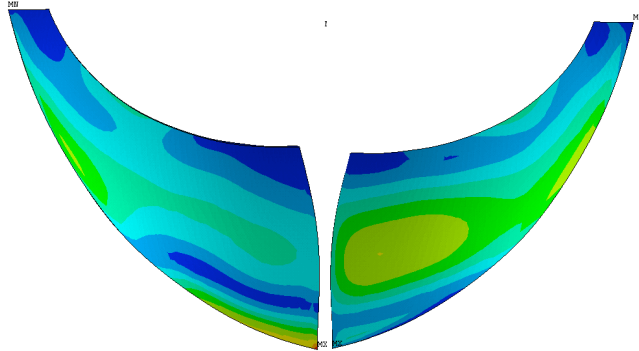
	1 st mode (Hz)	2 nd mode (Hz)	3 rd mode (Hz)	4 th mode (Hz)
Original	2614	4463	5915	6946
Inverse	3386	5528	6174	7952
% change	+29%	+24%	+4.4%	+14.5%

Table 2: Calculated Natural Frequencies of the first four modes for the Original and inverse designed impeller

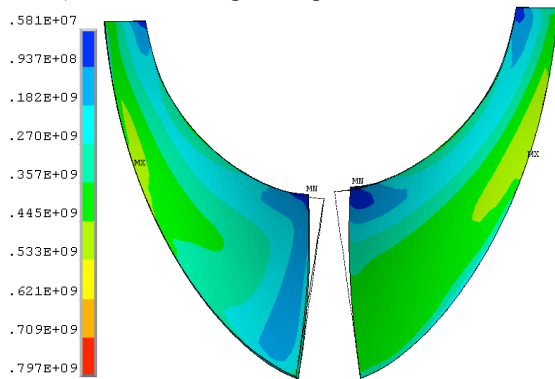
³ Titanium Ti-6Al-4V (Grade 5), Annealed. MatWeb Online database:
<http://www.matweb.com/search/DataSheet.aspx?MatGUID=a0655d261898456b958e5f825ae85390&ckck=1>



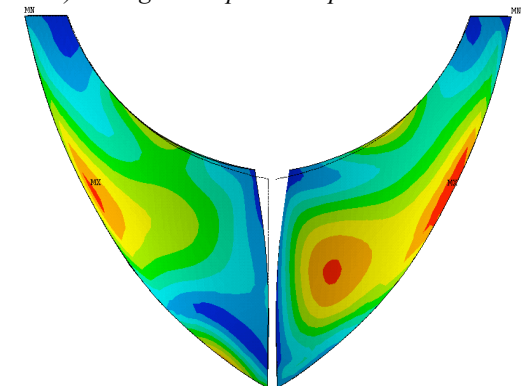
a) Original Impeller – Main Blade



b) Inverse designed impeller – Main Blade



c) Original impeller – Splitter blade



d) Inverse Design impeller – Splitter blades

Fig . 12 Comparison of Von Mises stresses in Original SRV2 and inverse design impeller – Units are in Pa

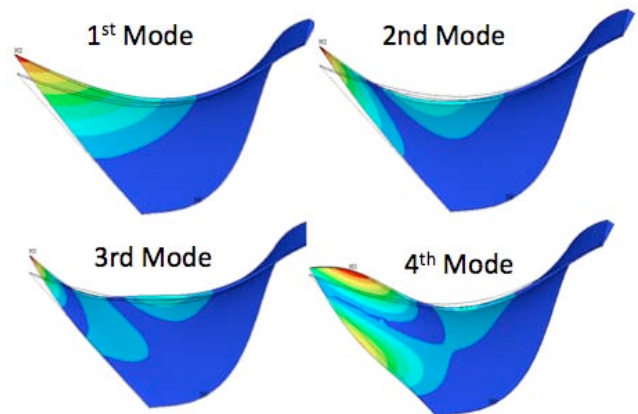


Fig. 13 Modal shapes for the original SRV2 impeller

PREDICTION OF PERFORMANCE OF THE INVERSE DESIGNED IMPELLER

The same mesh topology and mesh size as the medium mesh used for the analysis flow through the original SRV2 impeller was used to analyse the flow through the inverse designed impeller. The analysis was performed with the same vaneless diffuser geometry as the original SRV2AB stage. The resulting performance map comparison with the original SRV2AB stage is shown in Figs. 14 and 15, where the stage efficiency and stage pressure ratio are compared at 100%, 80% and 60% of design speed. The results show that the inverse designed impeller has a stage efficiency that is higher by 2.5 points at 100% speed and about 2 points at 80% and 60% speed. The stage pressure ratio is slightly higher at 100% speed but matches the original stage at part load.

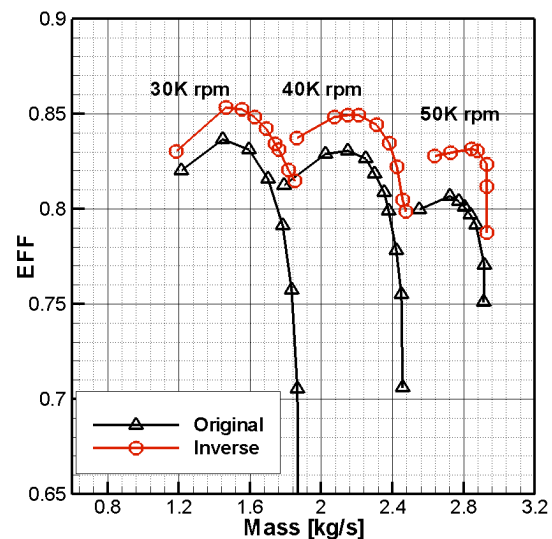


Fig. 14 Comparison of the predicted total stage efficiency of inverse designed impeller with original SRV2AB

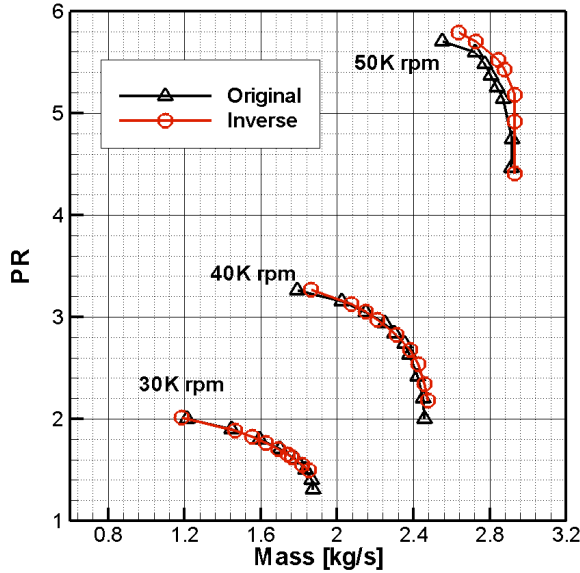


Fig. 15 Comparison of the predicted stage Total pressure ratio of inverse designed impeller with original SRV2AB

Fig. 16 compares the predicted impeller exit relative velocity corresponding to MF110 section with SRV2AB impeller at 100% speed and flow rate of 2.7 kg/s, which corresponds to peak stage efficiency of SRV2AB stage. The results confirm a significant reduction in the low momentum region at impeller exit especially in the channel between the splitter suction surface and full blade pressure surface.

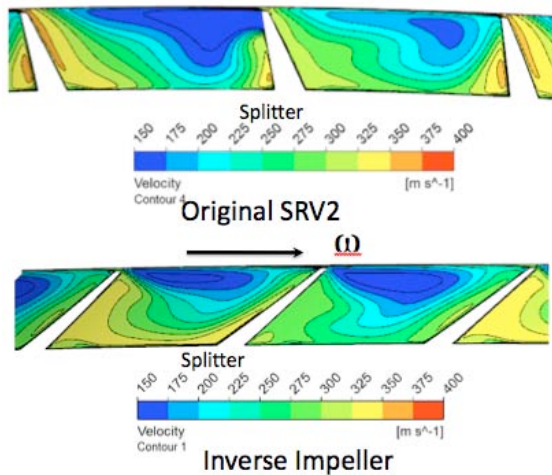


Fig. 16 – Comparison of exit relative velocity at MF110 location (just before impeller exit)

The Mach number contours for design point operation at midspan and 95% span of inverse designed impeller are shown in Figs. 17 and 18. When compared with the original impeller (Fig. 7) the midspan contours, indicate a significant reduction

of shock strength in the inducer and also the absence of low momentum region on the suction surface of the splitter which can be seen in the original impeller. In fact the same observation can be made from 40% span to 90% span.

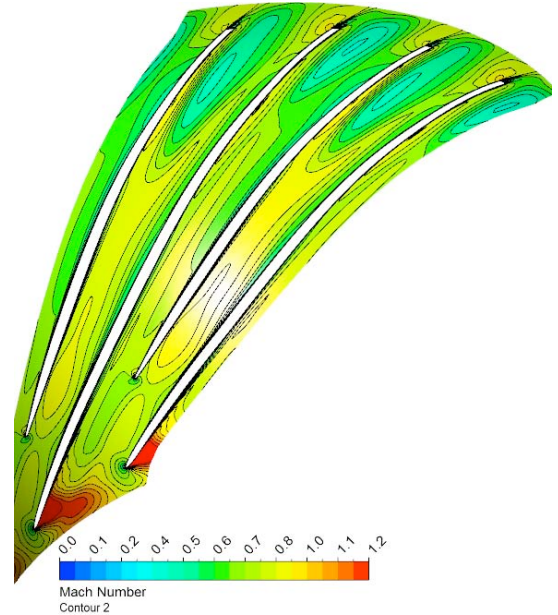


Fig. 17 Contours of relative Mach number at mid-span – Inverse impeller at 100% speed and 2.7 kg/s

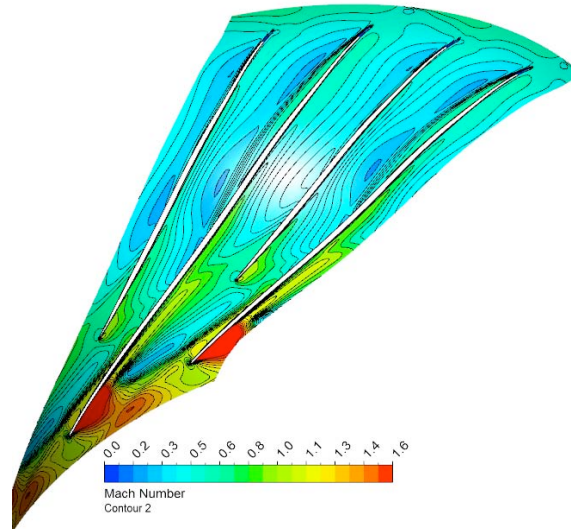


Fig. 18 – Contours of Relative Mach number at 95% Span – Inverse impeller at 100% speed and 2.7 kg/s

At 95% span the main improvement is the elimination of the separation zone on the pressure surface of the main blade. This is confirmed by velocity vectors near the pressure and suction surface of the inverse impeller, shown in Fig. 19.

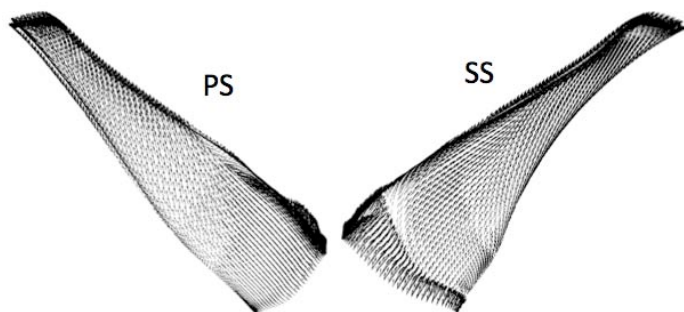


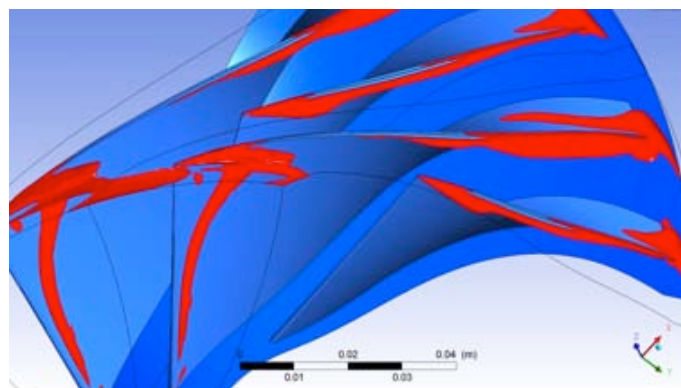
Fig. 19 – Velocity vectors near PS and SS of inverse impeller

Velocity vectors on the pressure surface confirm that there is no large scale separation present. The vectors on suction surface indicate a significant reduction in the secondary flows, especially in the inducer. In order to understand better the nature of the vortical flows in the two impellers, iso-surfaces of absolute helicity (which measures streamwise vorticity, i.e. dot product of velocity with vorticity) are presented in Fig. 20. Helicity is a good measure for distinguishing primary and secondary vortices and provides information on the degree of concentration of vortices, further details on helicity can be found in Levy et al (1990). In the inducer of the original SRV2 impeller one can see a region of high helicity near the leading edge extending from hub to shroud. In fact this region coincides with the shock location in the inducer. The spanwise extent of this region is reduced considerably in the inverse impeller confirming that the blade loading specified has resulted in substantial reduction in the inducer shock strength from the hub to mid-span. Also they show considerable reduction in helicity in the tip region of the full and splitter blade.

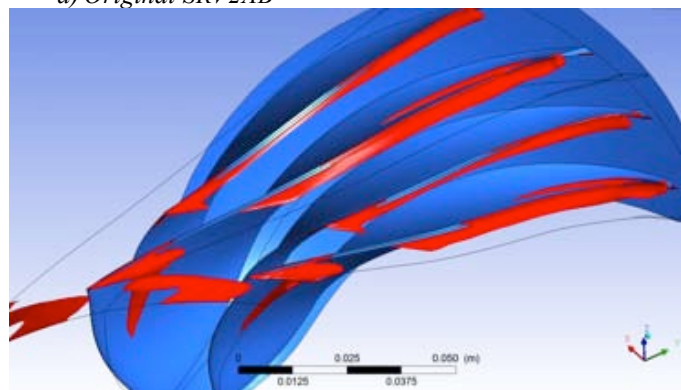
Finally the spanwise variations in isentropic efficiency at the trailing edges of the inverse and original impellers are compared in Fig. 21. They confirm that the most significant improvement in efficiency occurs from hub to 60% span. Additionally a small improvement can also be observed very close to the tip.

CONCLUSIONS

The L2F data available for the transonic high pressure ratio centrifugal compressor SRV2AB were used to validate CFD predictions for the impeller. The CFD predictions were used to understand better the nature of the flow losses in the impeller and to guide the choice of design specifications of a 3D inverse design code. By appropriate choice of loading distribution and spanwise work distribution and by making adjustment to relative work ratio between the impeller and splitter blades an impeller was designed that resulted in 2-2.5 point improvement in stage efficiency at 100%, 80% and 60% speeds. In particular the choice of a strongly aft-loaded hub and mildly aft-loaded shroud for the main blade helped to provide good compromise between suppression of secondary flows and decreasing the shock losses in the inducer. Furthermore by unloading the hub and mid-span rapidly in the inducer area it was possible to reduce the shock strength in the inducer from mid-span to hub.



a) Original SRV2AB



b) Inverse Impeller

Fig. 20 Comparison of Iso Surfaces of Absolute Helicity

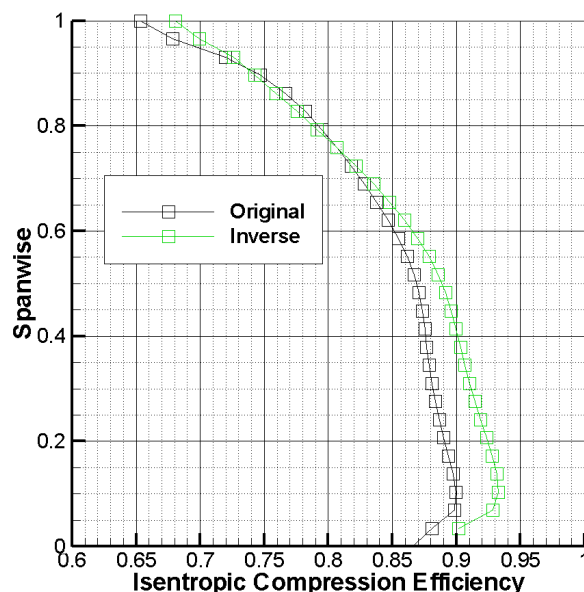


Fig. 21 Comparison of Spanwise variation of isentropic efficiency at impeller TE.

The CFD predictions confirm a more uniform exit flow for the inverse impeller. Despite the fact that the impeller geometry is 3D and impeller tip speed is rather high, the structural analysis of the impeller confirmed the maximum stresses overall are at similar level as the conventional straight filament design. Furthermore, modal analysis results confirmed that the 3D impeller is much stiffer with 1st natural frequency being 29% higher than the corresponding value for conventional impeller.

ACKNOWLEDGEMENT

The authors wish to thank Ms Molly Rice of Advanced Design Technology Ltd for her help and guidance on the structural analysis of the impellers.

REFERENCES

- Drtina, P., Dalbert, P., Rutti, K. and Schachenmann, A., (1993), 'Optimisation of a diffuser with splitter by numerical simulation', ASME paper 93-GT-110.
- Eisenlohr, G., Krain, H., Richter, F. and Tiede, V. "Investigations of the Flow Through a High Pressure Ratio Centrifugal Impeller ." *ASME GT-2002-30394*, Jan 2002: 9.
- Higashimori, Hirotaka, Susumu Morishita, and Tooru Suita. "Detailed Flow Study Of Mach Number 1.6 High Transonic Flow In A Pressure Ratio 11 Centrifugal Compressor Impeller " *ASME GT2007-27694*, Nov 2008: 10.
- Ibaraki, S., Matsuo, T., Kuma, H., Sumida, K., and Suita, T., 2003, "Aerodynamics of a Transonic Centrifugal Compressor Impeller". *ASME Journal of Turbomachinery*, **125** (2), pp. 346–351.
- Ibaraki, Seiichi, Masato Furukawa, Kenichiro Iwakiri, and Kazuya Takahashi. "Vortical Flow Structure And Loss Generation Process In A Transonic Centrifugal Compressor Impeller ." *ASME GT2007-27791*, Nov 2008: 10.
- Ishida, M., H Ueki, and Y Senoo. "Effect of blade tip configuration on tip clearance loss of a centrifugal impeller." *Journal of Turbomachinery*, Jan 1990.
- Krain, H, Hoffmann, B., Rohne, K.H., Eisenlohr, G. and Richter, F.A., "Improved High Pressure Ratio Centrifugal Compressor ." *ASME Proceedings GT2007-27100*, Jan 2007: 9.
- Krain, H. "Review of Centrifugal Compressor's Application and Development." *Journal of Turbomachinery*, Jan 2005.
- Levy, Y., Degani, D. and Seginer, A., (1990), "Graphical visualization of vortical flows by means of helicity", *AIAA Journal*, Vol 28, No 8., pp. 1347-1352.
- Marconcini, Michele, Filippo Rubecchini, Andrea Arnone, and Seiichi Ibaraki. "Numerical Analysis Of The Vaned Diffuser Of A Transonic Centrifugal Compressor ." *ASME GT2007-27200*, Nov 2008: 9.
- Marconcini, Michele, Filippo Rubecchini, Andrea Arnone, and Seiichi Ibaraki. "Numerical Investigation of a Transonic Centrifugal Compressor." *Journal of Turbomachinery* 130 (Jan 2008): 9.
- Krain, H and Hofmann, B, 1998, 'Flow Physics in High Pressure Centrifugal Compressors', ASME FEDSM98-4853.
- Thiele, M., (2000), ABB Turbosystem, 'Private communication'. TurboDesign-1 version 3.1, Advanced Design Technology Ltd, 2009.
- Zangeneh, M., 1991, "A compressible three dimensional Blade design method for radial and mixed flow turbomachinery blades", *Int J. Numerical Methods in Fluids*, Vol. 13, pp. 599-624.
- Zangeneh, M., 1998, "On 3D inverse design of centrifugal compressor impellers with splitter blades", *ASME Paper 98-GT-507*.
- Zangeneh, M., Goto, A. and Harada, H., 1998, "On the design criteria for suppression of secondary flows in centrifugal and mixed flow impellers", *ASME Journal of Turbomachinery*, Vol 120, pp. 723-735.
- Zangeneh, M., Goto, A. and Harada, H., 1999, "On the role of three-dimensional inverse design methods in turbomachinery shape optimization", *Proceedings of IMECHE, J. of Mech. Eng. Sci., Part C*, Vol. 213, No. C1, pp. 27-42.
- Zangeneh, M., Vogt, D., Roduner, C., 2002, "Improving a vaned diffuser for a given centrifugal impeller by 3d inverse design", *ASME Paper GT-2002-30621*
- Zangeneh, M., M., Schleer, M., Ploger, F., Hong, S.S., Roduner, C., Ribi, B. and Abhari, R., 2004, "Investigation of an inversely designed centrifugal compressor stage part 1: Design and numerical verification" *Trans. Of ASME, J. of Turbomachinery*
- Schleer, M., Hong, S.S., Zangeneh, M., Ploger, F., Roduner, C., Ribi, B. and Abhari, R "Investigation of an inversely designed centrifugal compressor stage part 2: experimental investigations", *Trans. Of ASME, J. of Turbomachinery*, Vol 126, pp. 82-90 (2004).
- Zangeneh, M., Nikpour, B and Watanabe, H. (2010), "Development of High performance centrifugal compressor impeller by using a 3D inverse design technique", 9th International Conference on turbochargers and turbocharging, London, May 2010.