Numerical Studies on the Effect of Impeller Blade Skew on Centrifugal Compressor Performance

V.C. Arunachalam[1], Q.H. Nagpurwala[2], M.D. Deshpande[3], S.R. Shankapal[4]

[1] M.Sc.(Engg.) Student, [2]Professor, [3]Professor and Head, Research, [4]Director
Centre for Rotating Machinery Design
M.S. Ramaiah School of Advanced Studies, Bangalore 560054

Abstract

Centrifugal compressors are designed for a given operating pressure and mass flow rate. But often these machines are required to be run at various off design operating conditions. The need to maintain reasonably high efficiency under off design conditions with acceptable stall margin makes the compressor design more challenging. These requirements call for improvemnet in the flow quality through the impeller by incorporating suitable changes in the vane shape. This paper examines the effect of introducing tangential skew near the trailing end of a high pressure ratio centrifugal compressor impeller. The blade skew was introduced in such a way that the blade angle distribution varied from hub to shroud near the trailing end only. Detailed, steady state, numerical simulations were carried out on the compressor, using a commercially available code, FLUENT. Results of the baseline CFD analysis on the original impeller were validated through comparison with the available experimental data. Subsequently, the impeller geometry was modified by incorporating different positive and negative skew angles, ranging from –45° to +45° and each configuration was analysed in terms of compressor overall performance and flow behaviour through the impeller blade passages. In general, the skewed impellers showed varying improvement in stall margin, pressure ratio and efficiency. It was observed that the impeller with +45° skew produced highest pressure ratio with reduced stall margin compared to baseline 0° skew impeller. On the contrary, the impeller with -45° skew produced a lower pressure ratio with increased stall margin compared to the baseline 0° skew impeller. It is concluded that a proper blade skew to the impeller needs to be imparted based on the requirement of higher pressure ratio or higher stall margin. The results have also suggested a need for further optimisation of skew angle at the impeller trailing end.

Key Words: Centrifugal Compressor, Impeller, Blade Skew, Lean, Stall Margin

Nomenclature

C	Absolute velocity, m/s
Cp	Specific heat at constant pressure
M	Mach number
N	Rotational speed, rpm
Ps	Static pressure, Pa
Po	Total pressure, Pa
To	Total temperature, K
U	Blade speed, m/s
W	Relative velocity, m/s
k	Turbulence kinetic energy, m^2/s^2
m	Mass flow rate, kg/s
α	Angle of attack, deg
β	Blade angle / Relative flow angle, deg
δ	Pressure correction factor
γ	Ratio of specific heats
ε	Dissipation rate, m^2/s^3
η	Isentropic efficiency
θ	Temperature correction factor
ω	Rotational speed, rad/sec

Abbreviations

CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
NASA	National Aeronautics and Space Administration

1. INTRODUCTION

Centrifugal compressor is one of the oldest turbo machinery, widely used in various industries like, aviation, oil & gas, refrigeration, etc. Centrifugal compressor is a radial turbomachine, which compresses air or gas with the action of centrifugal force. During the Second World War, the centrifugal compressors were used by British and American fighter aircrafts, as a part of early development of gas turbine engines. Later, during the 1950s, a large number of turboprop, turbofan, turbo-shafts and auxiliary power units started using the centrifugal compressors 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. A typical centrifugal compressor [1], used in aero-engine, is shown in Fig.1. Normally, multistaging of the centrifugal compressor is not preferred in the aviation industry because the pressure losses are higher compared to the multistage axial flow compressors [2]. However, multistage centrifugal compressors are the preferred choice in the oil and gas industry due to the requirement of large pressure ratios at high operating pressure levels of the order of 60 MPa.

There have been continuous efforts to improve the performance of centrifugal compressors. Some of the key studies are based on the modification of compressor geometry, especially impeller and diffuser. Since the impeller is an active part that adds energy to the fluid, its geometry plays a major role in the centrifugal compressor performance. Several improvements in impeller performance have been achieved by introducing splitter blades [3,4], tandem blades [5], three-dimensional impeller design [6,7], etc. Modification of the impeller geometry is considered to be one of the key approaches, as any change in the impeller geometry would have an impact on the impeller inlet or exit velocity triangles, which may result in significant performance change.

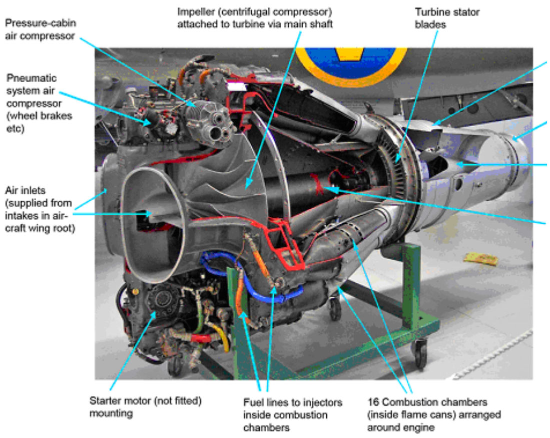


Fig. 1 Centrifugal Compressor for Aviation Application [1]

In the area of three-dimensional impeller design, studies have been reported [2,8] leading to improvement in the compressor performance by introducing skew (lean) to the impeller blades. Figure 2 shows a typical centrifugal compressor impeller having skew at the blade exit.

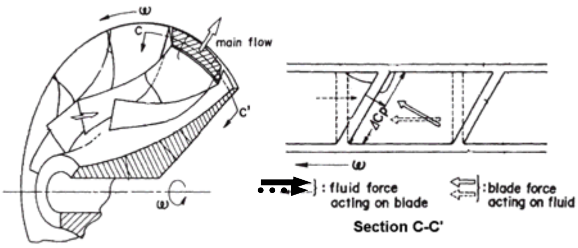


Fig. 2 Typical Compressor Impeller with Skew at the Vane Exit [2]

Imparting blade skew means introduction of variation in the blade angle from hub to shroud. The skew, either positive or negative, affects the flow through the impeller, resulting in a change in the pressure rise characteristic and stall margin.

A detailed study on the factors affecting the surging and consequent instability of a centrifugal compressor has been reported by Elder and Gill [9]. A complex mathematical model was developed and various parameters that affect the stall were studied. It is shown that the parameters, like slope of pressure rise characteristic, inducer incidence, impeller back-sweep angle, number of impeller and diffuser vanes, and diffusion rate in rotor and stator, have significant effect on the compressor stability limit. It is also established that the use of impeller back-sweep or inlet pre-swirl can provide basic improvement in the compressor stability. The compressor may reach stall point when the pressure rise characteristic reaches its peak.

Hildebrandt and Genrup [10] have presented the numerical investigations of the effect of different back sweep angles and exducer widths on the steady-state impeller outlet flow pattern of a centrifugal compressor with a vaneless diffuser. The CFD simulations were used to quantitatively investigate the influence of the compressor geometry on the flow pattern. Also, the location of the wake and its magnitude (flow angle and relative velocity) were analysed. Results have shown that the impeller with increased back sweep provides more uniform flow pattern in terms of velocity and flow deviation angle distribution, and offers better potential for diffusion process inside the diffuser.

The difference in the performance of a conventional and an inverse designed impeller has been compared in detail by Zangeneh et al [11] through numerical simulations, performed using the CFD code, TASCflow. The results of CFD analysis confirmed that the inverse design impeller had a more uniform exit flow, better control of tip leakage flow and higher efficiency than the conventional impeller. The results also showed that the shape of the trailing edge geometry had a very appreciable effect on the impeller performance.

Moore et al [12] have reported a study focusing the exit of an impeller that had large blade-to-blade flow variations, creating a highly unsteady flow field for the downstream diffuser. Impeller blade skew was used to improve the uniformity of the exit flow. A comparison is made with the turbines, where the skew has been used to control the secondary flows by tailoring the blade turning to reduce the flow overturning and underturning and to create more uniform loss distributions from hub to shroud. Blade skew is used to redistribute the flow to move the high loss fluid from the suction side to the hub, significantly reducing the blade-to-blade variations at the exit.

A patent by Harada and Shin [13] has disclosed the blade skewing techniques to improve the performance of the centrifugal compressors. It is claimed that the secondary flows can be suppressed by reducing the pressure gradient across the hub and the shroud by introducing blade skew in a closed impeller. It is also claimed that the compressor performance, including efficiency, can be improved by imparting tangential skew to the impeller blades.

The above discussion reveals that the overall performance and stability margin of centrifugal compressors can be improved by introducing tangential skew at the impeller blade exit. Although the published literature deals with the change in the global performance parameters of the compressors, there is no clear understanding of the change in the impeller passage flow, brought about by introducing the blade skew, that is responsible for the improvement in the compressor performance. This paper attempts to address the above issue. Detailed CFD simulations were carried out to predict the performance of a high pressure ratio backswept centrifugal compressor with five different skew angles, including the baseline zero skew case. Following sections explain the CFD modelling of the selected impeller with skew and without skew, validation of the CFD simulation using experimental data, and detailed discussion on the overall performance and on the flow behaviour through different impeller builds.

2. IMPELLER DESIGN SPECIFICATIONS AND GEOMETRIC MODEL

A centrifugal compressor having design pressure ratio of 4:1 and design rotational speed of 30000 rpm was chosen for the present study. The CAD model of the compressor impeller is shown in Fig.3 and the geometric data is given in Table 1. It must be noted that the compressor impeller was originally designed with a skew angle of +45° and the experimental performance data for this impeller was available at 50% design speed only. Hence, the validation of the CFD simulations was limited to 50% design speed on the impeller with +45° skew. For subsequent CFD simulation at 100% design speed, the impeller was remodeled with 0° skew and that was considered as the baseline impeller.

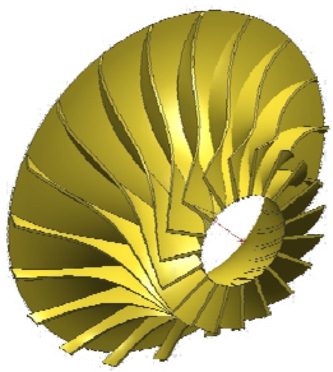


Fig. 3 CAD Model of Centrifugal Compressor Impeller

Table 1 Geometric Details of the Centrifugal Compressor Impeller

Parameters	Values
Type of impeller	Open Impeller
Number of blades	19
Impeller inlet eye dia	160mm
Impeller inlet root dia	80mm
Impeller tip dia	300mm
Back sweep angle	25-35°
Blade width at tip	9.63mm
Tip clearance	0.5mm
Operating speed	30,000 rpm

To investigate the effect of skew, the new baseline geometry was imparted different skew angles without violating other geometrical constraints and the CFD analyses were performed at the design rotational speed.

2.1 Details of the Numerical Model

Since the geometry of the impeller was cyclic symmetric, the CFD simulations were performed on one impeller passage only by applying periodic boundary condition, so that the size of the model could be reduced and correspondingly the computational time could be minimised.

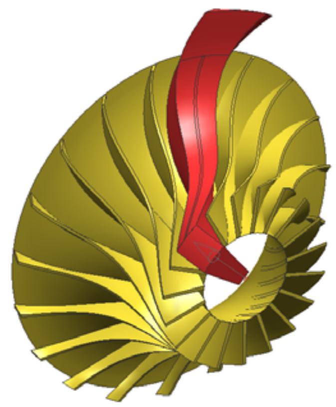


Fig. 4 CAD Model of the Impeller and the Extracted Fluid Domain

The impeller had 19 blades; hence an 18.95° sector model was created. The fluid domain was extracted for one impeller blade passage with the blade itself in the middle of the domain, as shown in Fig. 4.

The computational grid, comprising about 3.98 lakh structured hexahedral elements in multi-block environment, was generated using ICEM CFD. Sufficiently fine grid elements were created in the impeller tip clearance region, around the impeller, and at the hub and shroud walls, as shown in Fig.5. Sufficient mesh quality checks were performed by keeping the parameters like mesh angle and determinants within acceptable limits.

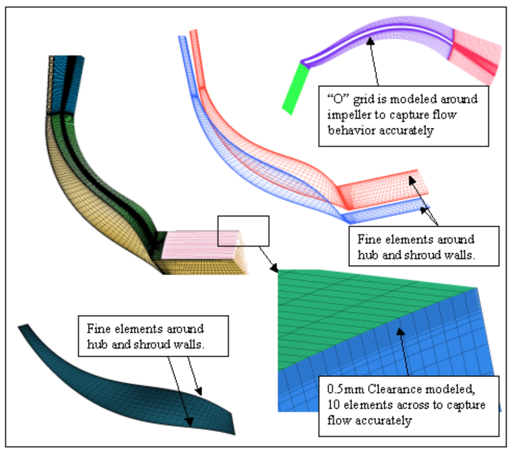


Fig. 5 Details of Computational Grid

The impeller hub and the blades were defined as rotating components with reference to the stationary domain. The shroud was defined as non-rotating domain. The periodic cutting planes were defined as periodic boundaries. At these periodic boundaries, the flow properties were computed by averaging the properties on either side of them. Working fluid in the compressor was air. Hence, standard air properties were used. The computations were performed using steady state segregated solver, with standard wall function, and also standard k-ε turbulence model.

4. GRID INDEPENDENCE STUDY

Grid size plays an important role in both convergence and accuracy of the solution. A coarse mesh is initially used to quickly examine the solver settings and boundary conditions. The grid generated should be appropriate to capture the complex flow phenomena like boundary layers, flow separation, leakage flows and secondary vortices in the blade passage. To capture all these flow characteristics, the number of cells should be large enough. But, as the number of cells increases, the computation time also increases rapidly. The grid size is, therefore, a compromise between computational time and accuracy of the results. Finer grids, in general, make the solution independent of the grid size and yield more accurate results but always require lager computational resources and time. Thus, a compromise between the grid size on one hand and convergence and accuracy on the other hand is required. Hence, a grid independence study was carried out to ensure that the numerical solutions are grid-independent. Computational grids having 398,000, 498,000 and 598,000 elements were generated and the CFD simulations were performed. The results are shown in the form of predicted compressor performance in Fig. 6. It can be observed that there is a good agreement in the results obtained from all the three mesh sizes. Though there is some difference between the results of 398,000 and 498,000 elements, there is no significant difference between 498,000 and 598,000 elements. Hence, a grid size of about 498,000 elements was used for the CFD simulations reported in this paper.

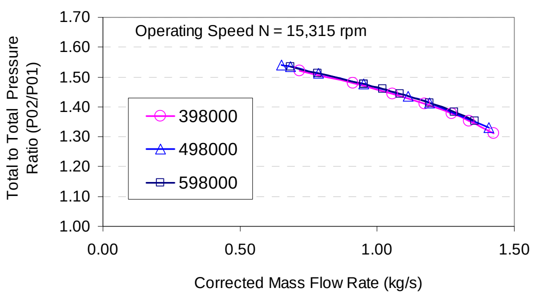


Fig. 6 Grid Independence Study

5. CONSTRUCTION OF SKEWED IMPELLERS

In order to understand the effect of blade skew, the baseline geometry was modified with different exit skew (lean) angles, both in positive and negative directions with respect to the baseline skew angle of 0°. In all, five impeller geometries were created with skew angles of +45°, +30°, 0°, -30° and -45°. Figure 7 shows two impellers with skew angles of +45° and -45° along with the baseline impeller of 0° skew. In case of positive skew, the impeller tip makes an acute angle with the direction of rotation and the point at the shroud of the tip section would *lead* in relation to the point at the hub when the impeller rotates. In case of negative skew, the impeller tip makes an obtuse angle with the direction of rotation and the point at the hub of the tip section would *lead* in relation to the point at the shroud when the impeller rotates. In the present study, the skew was not imparted to the entire impeller length from inlet to exit, but it was limited to a fraction of the impeller length near the trailing end, which was based on the energy addition rate along the impeller.

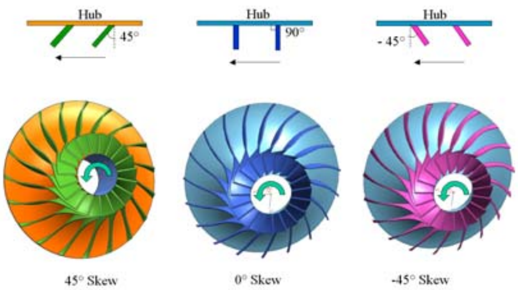


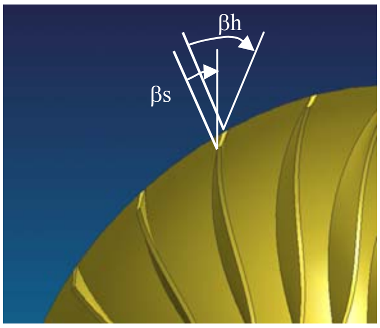
Fig. 7 Impellers with Positive, Zero and Negative Exit Skew

While implementing the skew, sufficient care was taken so that the geometrical features, like tangency and profile smoothness, were not disturbed. This was important because any deviation from a smooth blade profile would also influence the compressor performance and the effect of skew angle alone would not be isolated. Introduction of skew indirectly affects the blade angle distribution from hub to shroud as shown in the Fig 8.

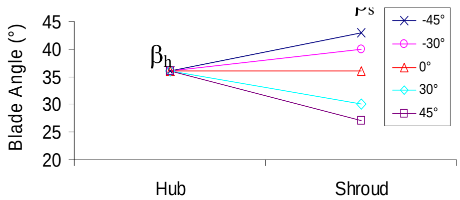
For all the skewed impeller cases the inducer angles remained unchanged. Introduction of the skew in this way was to ensure that the impeller flow profile is not changed too much, thus making the comparison of baseline and skewed impellers more meaningful. Five impeller geometries with different skew configurations were created. Computational grids were generated using ICEM CFD for the flow domain corresponding to one blade channel in each case. Hexahedral mesh was created using blocking technique and the same blocks were reused to generate CFD grids for impellers with different skew angles. The results were analysed and the same are discussed in Sections 6.

6. RESULTS AND DISCUSSIONS

The original impeller model was analysed for ~50% design speed, i.e. for 15,315 rpm and the results were compared with the available experimental data in terms of compressor performance maps. After validation of the CFD simulations at ~50% speed, the studies on skewed impellers were conducted at design speed, i.e. 30,000rpm. The effect of blade skew (five different skew configurations) is discussed in terms of compressor overall performance as well as detailed flow behaviour in the impeller blade passages.



a) Blade angle – pictorial representation



b) Blade angle variation for all study cases along impeller height at

Fig. 8 Modified Skew Geometries

6.1 Validation of CFD Simulations

The performance characteristics of the centrifugal impeller were obtained by computing the mass flow weighted flow properties like total pressure and total temperature at the impeller inlet and outlet. As one channel sector of the impeller was considered for analysis, the total mass flow rate was obtained by multiplying the mass flow rate obtained from theoretical analysis of one channel with the total number of blade channels. Isentropic efficiency was calculated using the relation,

$$\text{Isentropic Efficiency} \quad \eta = \left[T01 \frac{\left(\left(\frac{P02}{P01} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right)}{(T02 - T01)} \right]$$

In order to avoid the variations arising from inlet pressure and temperature conditions, the corrected mass flow rate was calculated by normalising the inlet total temperature and total pressure with the standard atmospheric conditions, using the following relations:

Corrected mass flow rate is given by

$$m_{corrected} = \frac{m * \sqrt{\theta}}{\delta} \text{ kg/s}$$

Pressure correction factor is given by $\delta = \frac{P_{o1}}{P_{atm}}$

and temperature correction factor is given by $\theta = \frac{T_{o1}}{T}$

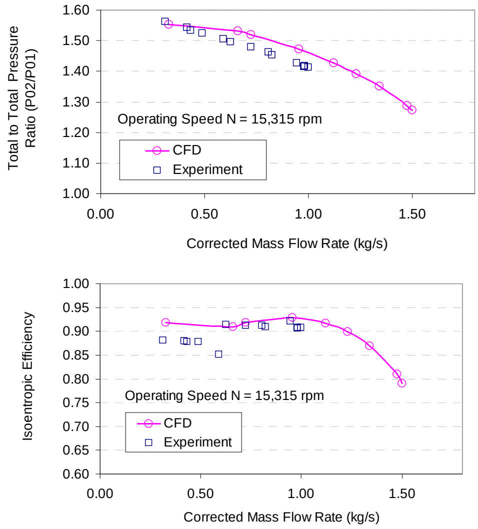


Fig. 9 Validation of CFD Simulations -- Impeller Performance at ~50% Design Speed

The performance characteristics of the original impeller with +45° skew at ~50% design speed are plotted in Fig. 9. There is a slight over-prediction of both the pressure ratio and the isentropic efficiency; however the trends are quite comparable. The difference may be attributed to the limitations of the CFD code and also to the experimental inaccuracies. Also, it is a general observation that most of the CFD codes tend to over-predict the turbomachinery performance.

6. 2 Performance of Skewed Impellers

The CFD simulations of the five impellers with different skew configurations (+45°, +30°, +15°, 0°, -30° and -45°) were performed at 100% design speed, i.e. 30,000rpm. In the following sections, the predicted performance of the impellers with different skew configurations is compared with the baseline (0°) skew case.

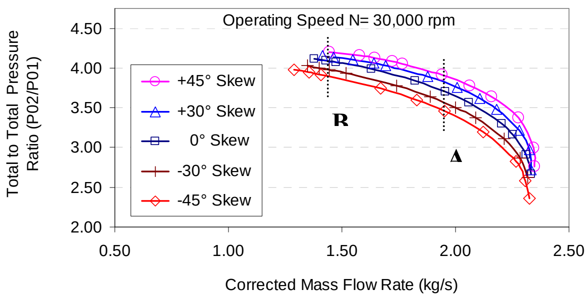
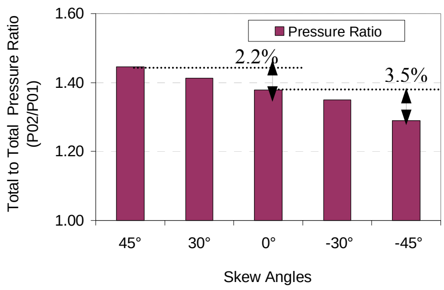
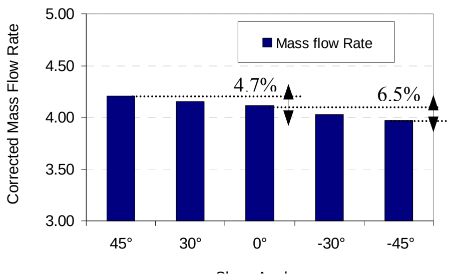


Fig. 10 Compressor Performance Map – Total Pressure Ratio for Different Skew Angles

The overall performance of the impellers in terms of total pressure ratio and mass flow rate for various skew configurations is shown in Fig.10. It is observed that there is significant increase in pressure ratio, when the impeller has positive skew at the trailing edge. The improvement in pressure ratio is about 2.2%. On the contrary, for negative skew, the compressor pressure ratio is reduced by almost 3.5% compared to the baseline 0° skew case. Also there is a notable difference in the compressor stall point pressure ratio and stalling mass flow rate by introducing the impeller skew.



a) Pressure ratio at stall point



b) Mass flow rate at stall point

Fig. 11 Variation of Stall Point Operating Conditions with Skew Angles

Variation in stall point pressure ratio and mass flow rates for various skew angles are shown in Fig.11. For a skew angle of +45°, the compressor stalls at a relatively higher mass flow rate compared to baseline 0° skew. The reduction in stall margin is ~4.7%. On the other hand, when the skew angle is changed from 0° to –45°, the stall margin improves by ~6.5%. It may be noted that the introduction of skew gives rise to a new performance map for each skew angle. The stall point pressure ratio increases when the skew angle is increased from negative to positive and the stall point mass flow rate decreases when the skew angle is changed from positive to negative.

Introduction of skew at the impeller exit has significant effect on the impeller efficiency also, as shown in Fig. 12. There is an appreciable improvement of 1.4% in the peak efficiency with +45° skew. The efficiency levels for all other skew configurations are very much comparable to the baseline impeller. The increase in isentropic efficiency for +45° skew case can be attributed to improved energy adding capability of the compressor with relatively favourable flow through the impeller passages.

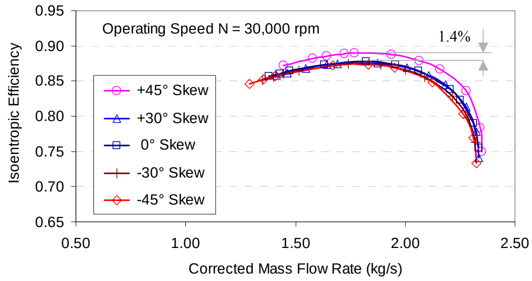


Fig. 12 Compressor Performance Map – Isentropic Efficiency for Different Skew Angles

The pressure ratio curves of the compressor with different skew angles diverge (Fig.10) when the compressor is operated at low mass flow rates i.e. near the stall region. Effect of skew is more when the compressor is operated at a high-pressure ratio. On the other hand, near the choke limit, there is not much variation in the pressure ratio between skewed and un-skewed cases, hence making the skew less effective towards the choke limit.

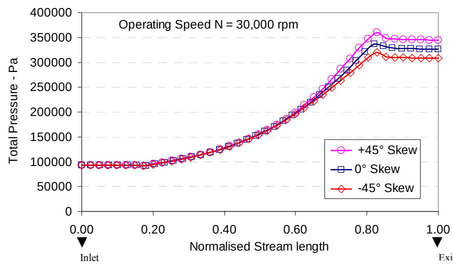
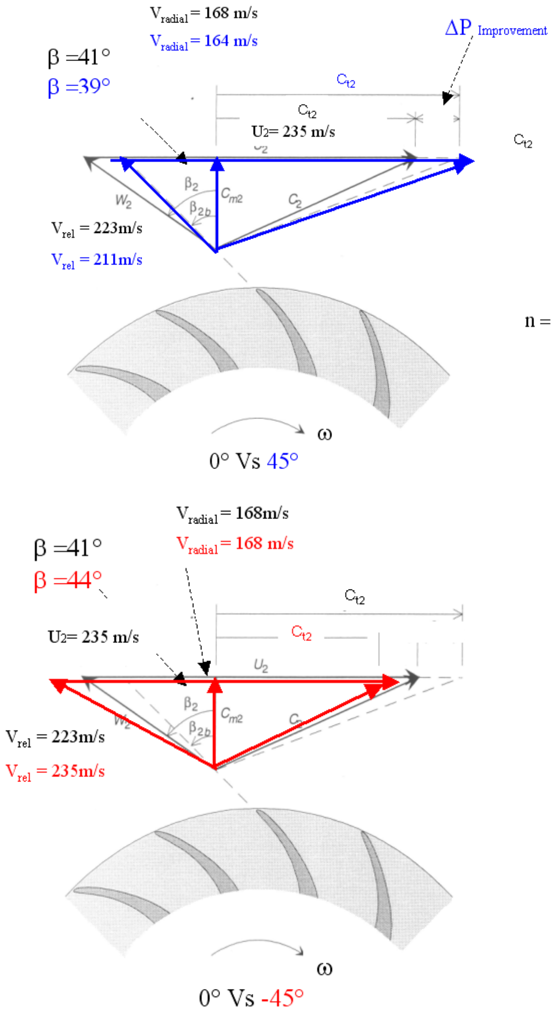


Fig. 13 Total Pressure Increase along Impeller

Figure 13 explains the variation in the energy addition to the fluid by impellers with different skew. The impeller with +45° skew is able to add higher energy from 75% of the merdional distance. Due to this enhanced work adding capability, this impeller is able to produce higher pressure ratio and also higher efficiency for the entire mass flow range.

Due to the blade skew, there is variation in the mass flow averaged relative flow angles at the impeller exit. For +45° skew case, the variation is about –2° compared to 0° skew, and for –45° skew, it is about +3°. The effect of blade angle skew on the relative flow angle is explained with the help of the velocity triangles shown in Fig.14. The lighter triangles represent 0° skew case, while the darker ones represent +45° skew (left figure) and –45° skew (right figure). In case of +45° skew, the relative flow angle β_2 has changed from 41° to 39° and the relative velocity of the flow exiting the impeller is reduced. This reduction in relative velocity is compensated by an increase in the absolute flow velocity, C_2. Higher exit absolute velocity means higher tangential component, thus leading to higher work coefficient. Consequently, the impeller with +45° skew is able to generate more pressure for the given work input. This is the reason for relatively higher pressure ratio and efficiency of the impeller with +45° skew.



Work Done = $C_{t2}U$ Efficiency $\eta = \left[T01 \frac{\left(\left(\frac{P02}{P01}\right)^{\frac{\gamma-1}{\gamma}} - 1 \right)}{(T02 - T01)} \right]$

Fig. 14 Exit Velocity Triangles – Effect of Relative Flow Angle Variation

On the contrary, when -45° skew is introduced, the relative velocity increases due to increased relative flow angle. This increase in relative velocity is reflected in a reduction in the absolute exit velocity and a lower tangential velocity at the exit. Due to smaller tangential velocity, the work addition is less, resulting in lower pressure rise. The relative Mach number distributions in the blade-to-blade planes at the impeller exit are shown in Fig. 15. The plots correspond to the stall mass flow rate of the +45° skew impeller (operating points at 'B' in Fig. 10). It is observed that the impeller with +45° skew has a larger low momentum fluid zone at the impeller exit, towards the shroud, compared to the other two cases.

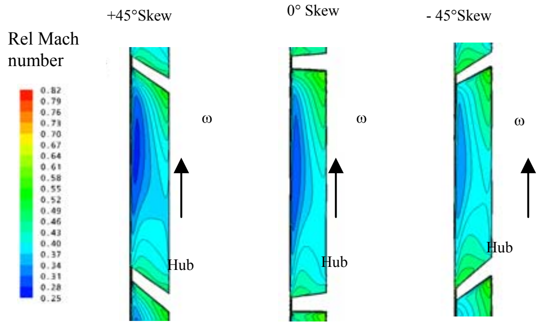


Fig. 15 Relative Mach Number Distribution at Impeller Exit for Three Skew Cases (At stall point mass flow rate of impeller with +45° skew)

It must be noted that although the +45° skew impeller is operating near stall, the other two impeller builds, with 0° and –45° skew, are away from the corresponding stall points. Hence the low momentum fluid regions are relatively smaller in extent. The larger low momentum zone in the +45° skew impeller is created due to higher diffusion near the stall point.

It can be argued that the factors responsible for reduced stall mass flow rate for –45° skew impeller are: a) lower pressure rise characteristic; b) less aggressive work addition; and c) lower diffusion rate within the impeller passage. From the above discussions, it is understood that the impeller with +45° skew is capable of generating higher pressure ratio with higher efficiency, but the stall initiation is at a higher mass flow rate than that for +0° and -45° skew impellers.

7. CONCLUSIONS

Based on the numerical simulations carried out on the centrifugal compressor impellers with different exit blade skew angles, the following conclusions are drawn:

- Predicted overall performance at ~50% design speed is compared with the available experimental data. The agreement between the CFD predictions and the experimental data is good with a maximum of 5% deviation.
- Introduction of skew at the impeller trailing edge significantly changes the work addition to the fluid. Flow behaviour in the inducer section is not affected by the skew at the trailing end.
- With positive skew, the compressor performance in terms of pressure ratio and efficiency is improved. For the +45° skew case, the simulations show a pressure ratio improvement of ~2% compared to 0° skew impeller.
- Introduction of positive skew has resulted in lesser back-sweep angle at the impeller hub, resulting in higher work addition. Hence, the efficiency and pressure ratios are better than those for the 0° and –45° skew.

- With negative skew, the compressor performance in terms of stall margin improves. For the -45° skew case, the simulations show a stall margin improvement of ~6% compared to the 0°skew case.
- Due to negative skew at the impeller exit, energy addition to the fluid becomes less aggressive with lower diffusion in the impeller passage, leading to a safe compressor operation even at reduced flow rates.

REFERENCES

[12] http://en.wikipedia.org/wiki/Image:DH_Goblin_annotated_colour_cutaway.png (January 10, 2008)

[13] Saravanamuttoo H.I.H., Rogers G.F.C. and Cohen H. (2003), "*Gas Turbine Theory*", Pearson Education, 5th Edition.

[14] Hirotaka Higashimori, Kiyoshi Hasagawa, Kunio Sumida and Tooru Suita (2004), "Detailed Flow Study of Mach Number 1.6 High Transonic Flow with a Shock Wave in a Pressure Ratio 11 Centrifugal Compressor Impeller", Trans. ASME Journal of Turbomachinery, Vol. 126, pp. 473-481.

[15] Larosiliere L.M., Skoch G.J. and Prahst P.S. (1997) "Aerodynamic Synthesis of a Centri-fugal Impeller Using CFD and Measure-ments", 33rd Joint Propulsion Conference and Exhibit AIAA–97–2878, Technical Report ARL–TR–1461.

[16] Robert D. A. and Kacker S.C. (1992) "Numerical Investigation of Tandem Impeller Designs for a Gas Turbine Compressor", Trans. ASME Journal of Turbomachinery, Vol.124, No.1, pp. 36-44.

[17] Zangeneh M. (1991) "A Compressible 3D Design Method for Radial and Mixed Flow Turbomachinery Blades" Int. Journal of Numerical Methods in Fluids, Vol. 13, pp. 599-624.

[18] Zangeneh M. (1998) "On 3D Inverse Design of Centrifugal Compressor Impellers with Splitter Blades", 43rd ASME International Gas Turbine and Aeroengine Congress and Exposition, Stockholm, Sweden, ASME paper 98-GT-507.

[19] Moore J. and Moore J. G. (1994) "Use of Blade Lean in Turbomachinery Redesign", NASA Marshall Space Flight Center, Eleventh Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion.

[20] Elder R.L. and Gill M.E. (1985) "A Discussion on the Factors Affecting Surge in Centrifugal Compressors", Trans. ASME Journal of Engineering for Gas Turbines and Power, Vol. 107, p. 499.

[21] Hildebrandt A. and Genrup M. (2007) "Numerical Investigation of the Effect of Different Back Sweep Angles and Exducer Widths on the Impeller Outlet Flow Pattern of a Centrifugal Compressor with Vaneless Diffuser", Trans. ASME Journal of Turbomachinary, Vol. 129, p. 433.

[22] Zangeneh M., Schleer M., Hildebrandt A. and Genrup M. (2004) "Investigation of an Inversely Designed Centrifugal Compressor Stage-Part I: Design and Numerical Verifi-cation", Trans. ASME, Journal of Turboma-chinary, Vol. 126, p. 73.

[23] Moore J., Moore J.G. and Timmis P.H. (1984), "Performance Evaluation of Centrifugal Compressor Impeller using 3D Viscous Flow", Trans. ASME, Journal of Engineering for Gas Turbines and Power.

[24] Harada H. and Shin Konomi (2002) "*Centrifugal Turbomachinery*" United States Patent No. 6,338,610 B1.