

ACGT201694

Flow studies in a Mixed Flow Compressor Stage for Small Gas Turbine Applications

Logesh N
Project Fellow, CSIR-NAL
Bangalore, Karnataka, India

Emandi Rajesh
Scientist, CSIR-NAL
Bangalore, Karnataka, India

Dr.R.Rajendran
Professor, DSCE
Bangalore, Karnataka,
India

ABSTRACT

In a small gas turbine engine where the requirement of the high thrust to weight ratio is of prime importance, centrifugal compressor play a major role. In a single stage centrifugal compressor can develop high pressure ratio compared to axial flow compressor. Recently mixed flow compressors are being used for the application of moderate flow and high work coefficient applications. A mixed flow stage is designed for 1.75 kg/s and 5 pressure ratio to fit in a geometric envelope of 220mm diameter and axial length of 135mm. Numerical analysis on the model shows that 4.42 total to total pressure ratio for 73.8 % isentropic efficiency. Reason for the drop in efficiency and pressure ratio against the targeted value is predicted through numerical analysis. Detailed flow studies and component performance prediction is carried out and which shows the recommended modifications in the geometry to be carried out so that performance and stall margin improves.

Keywords: Mixed flow impeller, Diagonal diffuser, OGV, Flow angle, incidence, performance, efficiency.

NOMENCLATURE

Mr Relative mach number
 Stall Margin $\left[\frac{(PR)_{stall}}{(PR)_{design}} * \frac{(massflowrate)_{design}}{(massflowrate)_{stall}} - 1 \right] * 100$
 i (incidence) flow angle-blade angle
 n fi number of blades at impeller
 α Absolute flow angle

α' Diffuser inlet blade angle
 β Relative flow angle
 β' Impeller inlet blade angle
 Ψ Work coefficient
 Ψ $(h_{02}-h_{01})/U_2^2$

1.Introduction:

Mixed flow compressor development plays a major role in the application of short range missiles, unmanned vehicles, drones and UCAVs due to its high thrust to weight ratio. Smaller frontal area and diameter satisfies the requirements of Small Gas turbine engine. Computational analysis of compressor stage helps the optimization process and performance prediction of designed compressor. Predicting the flow behavior in complex flow regime is a challenging task but development of computational solving techniques and schemes helps to study such complex flow behaviors. Present work details the simulation and performance prediction of compressor at design and off-design conditions.

Mixed flow compressors come into importance with its moderate flow and pressure rise coefficients. Axial flow compressors are preferred for high flow coefficient with less pressure rise coefficient and also it has less operating range and efficiency, whereas radial flow compressors are preferred with less flow and high pressure rise coefficient. It has wide operating range and high efficiency. Mixed flow compressors are preferred for higher flow and pressure rise coefficient with moderate operating range.

Compressor numerical analysis was reported by many researchers, but there is lack of information about the interaction of mixed flow impeller with diagonal diffuser. M.Govardhan, and S.Ramamurthy[1] details the CFD analysis of mixed flow impeller and compared with experimental results and found good agreement with experimental results.

A mixed flow compressor is designed for a small gas turbine engine which can produce 100kgf thrust. The mixed flow compressor stage geometry is mentioned in the Table 1. Figure 1 provides the meridional view of the compressor stage which has rotor, stator and outlet guide vane. Due to geometrical constraints the diffuser is positioned in such a way that it is seen as curved diffuser. The present work deals with the detailed numerical flow study on the compressor stage at different speeds.

Geometrical parameters	
Impeller	
Inlet hub diameter (m)	0.050
Inlet tip diameter (m)	0.123
Exit tip diameter (m)	0.1994
Cone angle (deg.-axial)	31.93
Back sweep (deg.-axial)	-25
Number of vanes (M+S)	11+11
Diffuser	
Number of vanes	15
LWR	5.56
AR	2.7
Outlet guide vane	
Number of vanes	45
Blade height (m)	0.010
Stage diameter (m)	0.220
Stage axial length (m)	0.1796

Table 1: Geometrical detail of MFC

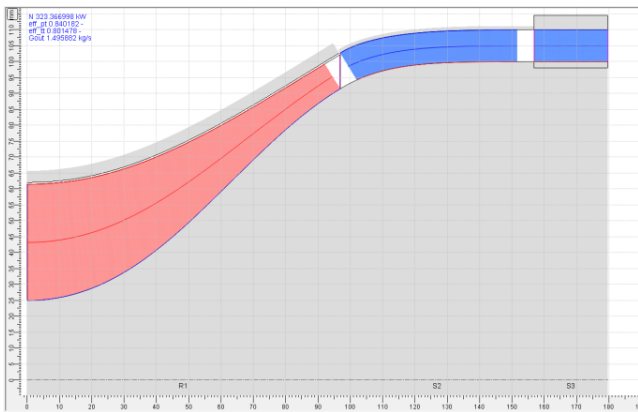


Fig 1: Meridional view of mixed flow compressor stage

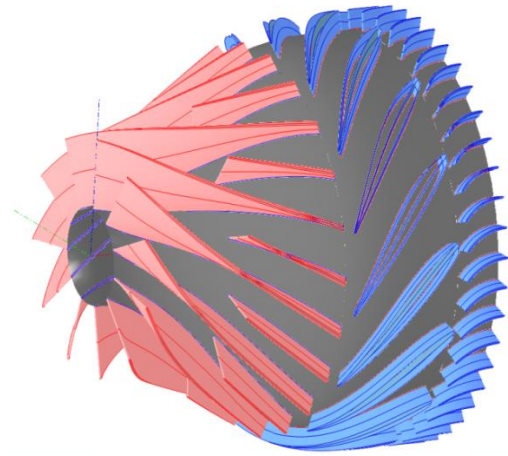


Fig 2: 3D model of mixed flow compressor stage

2. Numerical Analysis

3D geometry of compressor model shown in Figure 2 is exported to 'Bladegen' to model flow field. 'Turbogrid' is used for meshing, which gives automated high quality hexahedral structured mesh. Meshing is done separately for mixed flow impeller, diagonal diffuser and outlet guide vane. Uniform tip clearance of 0.5mm is given for impeller and high dense mesh is constructed to capture the secondary flow. Impeller is highly twisted from hub to tip, for quality mesh generation multiple layers are created. By keeping minimum and maximum face angle in control impeller mesh is generated with high aspect ratio. Traditional control points were used for topology definition using H/J/C/L-Grid methods given by Turbogrid.

O grid is used with width factor of 0.4 for the prediction of boundary layer on the end wall region. The details of the mesh generated for the configurations are shown in Table 2 and Figure 3 shows details of the mesh at the impeller. Similar structured mesh is generated for diffuser and OGV.

Components	Total no. of nodes	Total no. of elements	Total Number of Hexahedrons	Total no of faces
Impeller	747992	703575	703575	99690
Diffuser	291005	260100	260100	60920
OGV	416130	392512	392512	53610

Table 2: Grid details

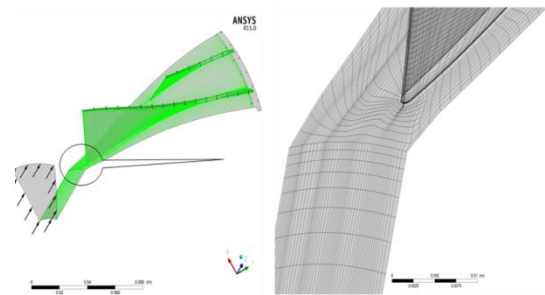


Figure 3: Meshing schematic for the impeller

Boundary conditions

The boundary conditions were given in the CFX-Pre^(c) module in which the initial setting was chosen as centrifugal

compressor with z axis as rotation. The numerical analyses were carried over for steady state ranging from 75% to 105% of design speed in steps of 5%. The design speed is 50500 rpm. The impeller is given with anticlockwise rotation, the stator and OGV are taken as stationary components. The boundary conditions are provided in the Table 3 and is shown in the Figure 4. The blades, hub and shroud surfaces are taken as wall with no slip condition.

Physical Definition	
Fluid	Air Ideal Gas
Analysis Type	Steady state
Model Data	
Reference Pressure	0 Pa
Turbulence Model	Shear stress transport model
Inlet/ Outlet boundary condition	
Inlet	Total Pressure = 101325 Pa
	Total Temperature = 293K
Outlet	Static pressure varying from choke to stall condition

Table 3: Inlet boundary conditions

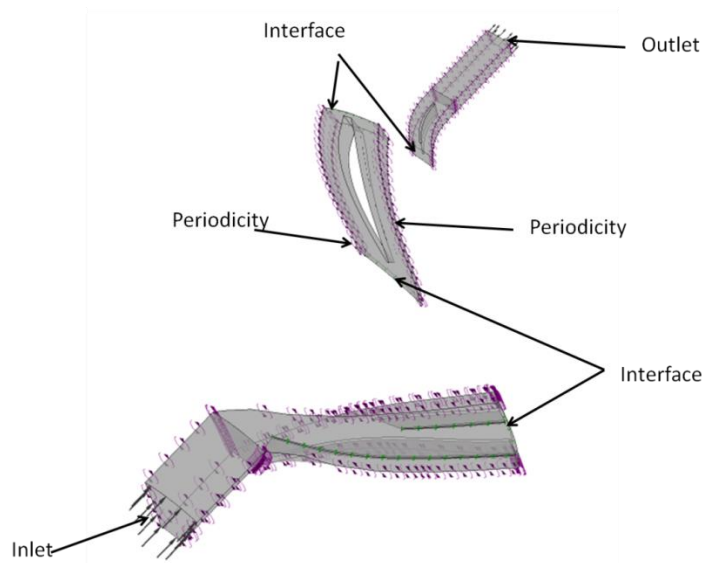


Figure 4: Boundary Conditions for fluid domain

K- ω SST turbulence model was opted with high resolution advection scheme and an auto time scale factor of 1. The residual target for proper convergence was increased from 10^{-4} to 10^{-6} . Another check on the mass flow rate imbalance and efficiency was monitored. The solution is considered to be converged when the residuals are below 10^{-6} or the successive difference between mass flow rate and efficiency is found to be near zero for more than 1000 iterations.

Post Processing

The post processing was carried out in CFD –Post^(c) and Tecplot^(c). The performance parameters such as total pressure ratio across the stage and efficiency is evaluated by creating the planes as shown in the figure 5.

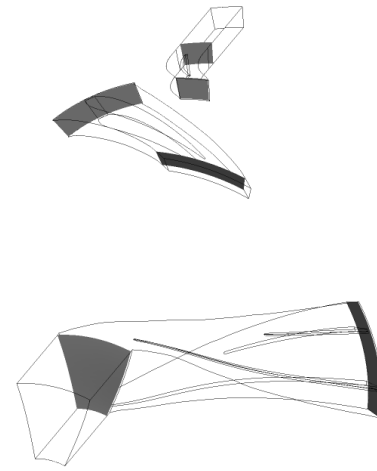


Figure 5: Planes considered for Post Processing of stage

3. Results and discussions:

Stage Performance

The total pressure ratio curves were obtained at various speeds and plotted against the corrected mass flow rates and are shown in the Figure 6. It is observed that stage pressure ratio obtained at the design condition is 4.3 against the desired value of 5. The stall margin obtained at the designed speed is about 4.57 %. The stall margin obtained at different speed is evaluated. It is observed that as the speed comes down the stall margin is increased.

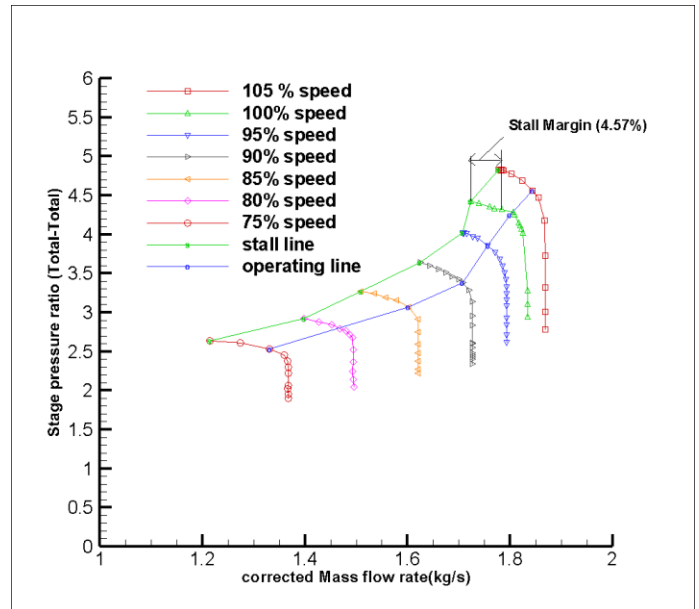


Figure 6: Stage Pressure ratio vs mass flow rate

The isentropic efficiency (T-T) of the compressor stage are evaluated and shown in Figure 7. It is observed that the isentropic efficiency obtained at the designed speed is 73.85 % against the desired efficiency of the 80%. The loss in the isentropic efficiency is due to the loss in the stagnation pressure in diffuser and outlet guide vane. The stagnation pressure loss is related to the separation caused due to high curvature in the hub surface as shown in the Figure 14(b) and (d).

It is observed that the impeller had a better performance at 95% speed and below. The reason for drop in

the efficiency at design speed may be due to the high Mach number at the inlet of the diffuser which is causing separation at the L bend. The other reason for drop in efficiency is due to the high relative Mach number at the impeller inlet, which would produce shock and it is observed that high relative Mach number covers 1/3 of the blade height at inlet.

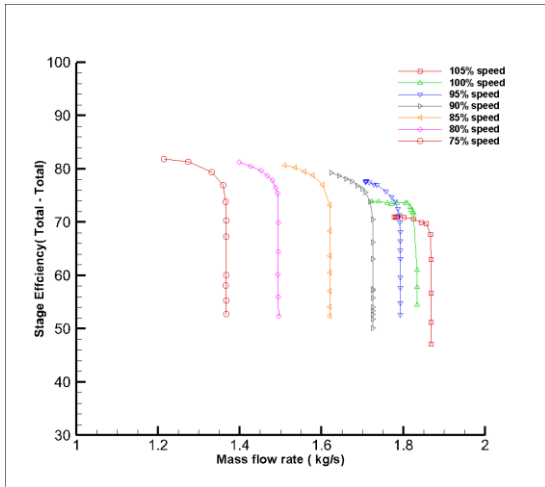


Figure7: Stage Efficiency Vs Mass flow rate

Flow angle:

The energy transferred from the impeller to the fluid to results in the increased absolute velocity. The absolute velocity exits from the impeller is at an angle. The angle between the absolute velocity components and the radial velocity components is called absolute flow angle. The diffuser design depends upon the exit flow angle from the impeller. As the impeller is designed for 70° of absolute flow angle. Against which the obtained absolute flow angle is 71° at mean. Figure (8) shows the absolute flow angle variation from hub tot tip. It is observed that the flow angle is almost uniform up to 80% of the blade height whereasit increases to 90° at tip section. The increase in the flow angle at the tip is due to the interaction of the tip clearance flow to the primary flow.

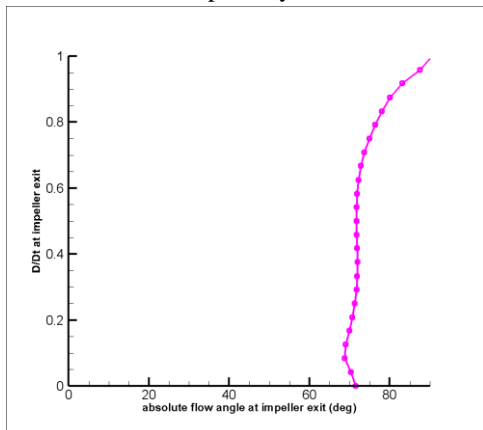


Figure8 : Absolute flow angle at impeller exit

The diffuser was designed to turn the flow from 70° at impeller exit to 45°. The flow angle at the diffuser exit is

plotted from hub to tip as shown in Figure 9. It is observed that the flow angle at the diffuser exit is not uniform as compared to the impeller exit flow angle. The flow angle is varying from 40° to 90°. This indicates that the flow in the diffuser channel is not uniform and is being separated from 50% of diffuser hub surface. From figure 1 it could be observed that the diffuser leading edge is positioned near to the bend. Because of the geometrical constraint in the diameter an improper diffusion is happening which indicates that there is a locally high Mach number at the diffuser inlet near to hub. And as there is bend near to leading edge it is causing the flow to separate from L-bend. This effect is causing higher variation of flow angle from hub to 50% of span, from 50% to the tip flow is almost uniform. Average flow angle at diffuser exit is 59°.

Flow angle obtained is at higher end than the requirement it might be because of the separated flow. The separation of the flow on the diffuser could be reduced by shifting the diffuser leading edge such that the diffuser will face a subsonic flow or by relaxing the hub curvature variation on hub.

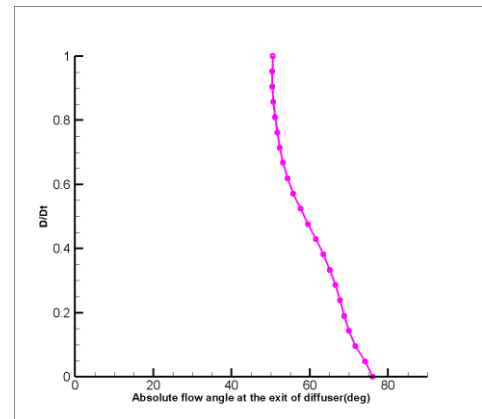


Figure 9: Absolute flow angle at diffuser exit

The OGV is designed for turning the flow from 50 deg. to 30 deg. The flow leaving the diffuser is highly non uniform (Figure 9). The non-uniform flow when interacted with the OGV the flow separation occurs due to mismatch in the blade angle and flow angle causing high negative incidence. This causes separation of the flow on the OGV blade. Even from static pressure and relative Mach number contours it is evident that OGV is operating at fully separated zone. Figure 14 (d) shows that major flow separation in the entire compressor stage is happening at OGV portion only. Average flow angle at OGV exit is 25 deg. From this study it is suggested that OGV has to be redesigned to avoid flow separation but the requirement may not met.

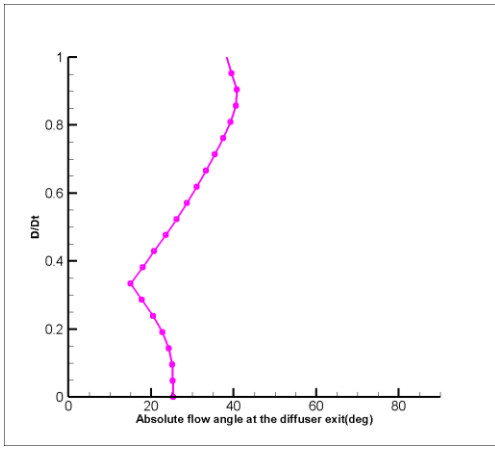


Figure 10: Absolute flow angle at OGV exit

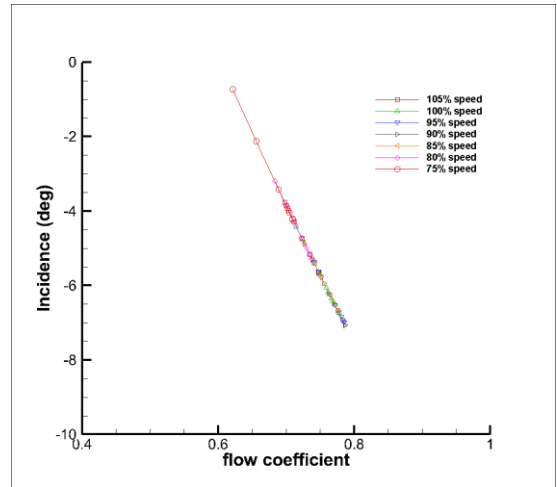


Figure 12 : Incidence to impeller at various speed

Incidence:

The incidence angle is defined as the difference between the absolute flow angle and tangential line at inlet blade.

$$i_{impeller} = \beta_1 - \beta'_1$$

$$i_{diffuser} = \alpha_2 - \alpha'_2$$

The incidence variation at design speed for various flow coefficients is shown in Figure 11 at the design flow coefficient of 0.734 the impeller is having a negative incidence of -5° where as the diffuser has -0.5° . This shows a good matching of flow between impeller and diffuser. It can be observed that at lower flow coefficients for each speed the incidence variation is linear which indicates that the flow angle is varying w.r.t to the axial velocity component.

The impeller incidence variation is shown in Figure 12 for various speeds. A linear variation is observed with respect to flow coefficient and as the speed is reduced the incidence angle is approaching towards zero, because of this phenomena the impeller is performing better with better efficiency at off design speeds.

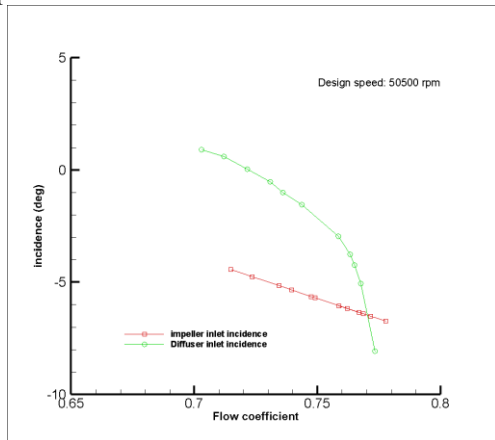


Figure 11: Incidence to impeller and diffuser at design speed

Incidence of the impeller at the design speed is plotted from hub to tip is shown in Figure 13. The variation of the incidence shows that as the back pressure is increased the incidence is varying towards positive. It is observed that as the back pressure is increased the positive incidence near the hub surface is increased. This indicates a separated flow on the hub surface.

Incidence to diffuser is plotted and shown in Figure 14. It is observed that upto a back pressure of 415 kpa the diffuser is having a negative incidence up to the blade height of 60%. As the back pressure in numerical analysis is increased, the incidence shifts to positive and it is observed that near to the tip zone the diffuser has very high positive incidence. This high positive incidence is due to the high flow angle as shown in Figure 8. From this observation it can be concluded that for this stage the stall is initiated at the diffuser.

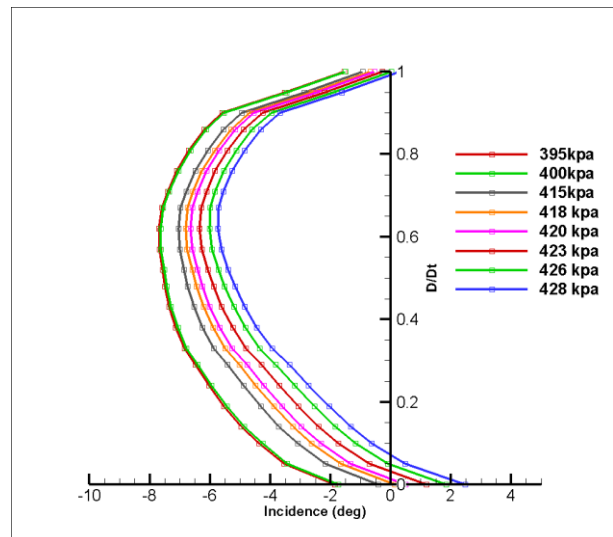


Figure 13 : Incidence at impeller inlet

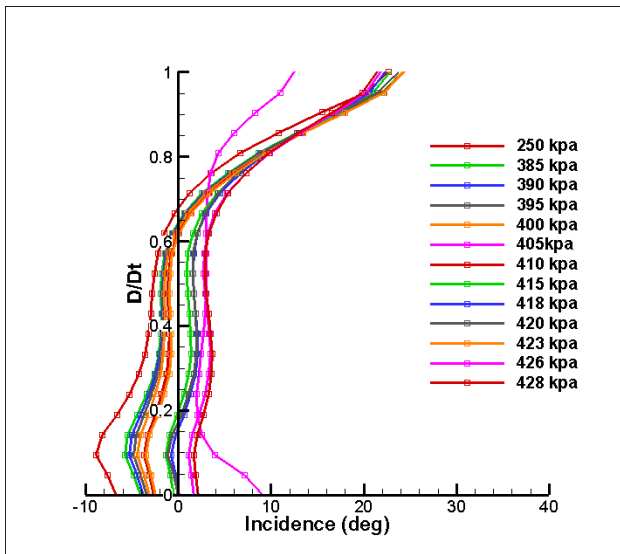


Figure 14: incidence at diffuser inlet

Blade loading:

The normalized static pressure variation on the blade surface at hub, mean and tip is plotted in Figure 11,12 and 13. It is observed that the designed mixed flow impeller is loaded properly without any major flow separation on the pressure and suction surfaces at design point. Loading diagram shows that there is uniform loading from inlet to exit at design point but there is a cross over at leading edge section. Cross over is dominant at the choking point and moves towards the leading edge as the compressor goes near to stall. This is due to the change of incidence. Static pressure plots shows that there is proper diffusion in the impeller flow field which results in the static pressure rise across the impeller.

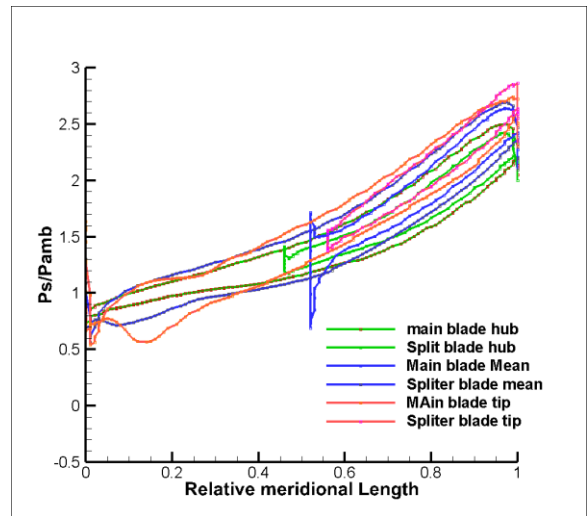


Figure 12: Blade loading at design point

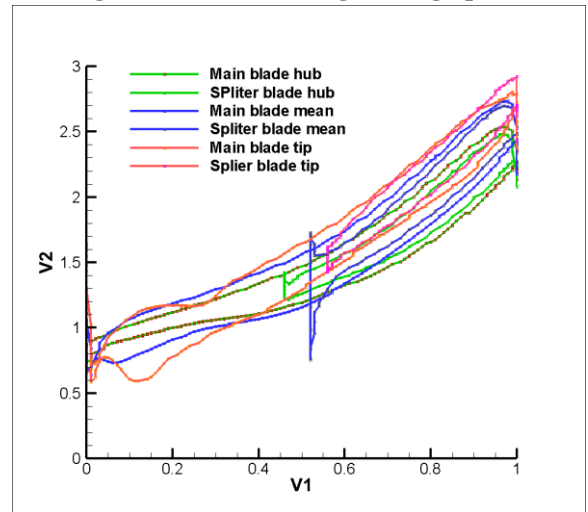


Figure 13: Blade loading at near stall point

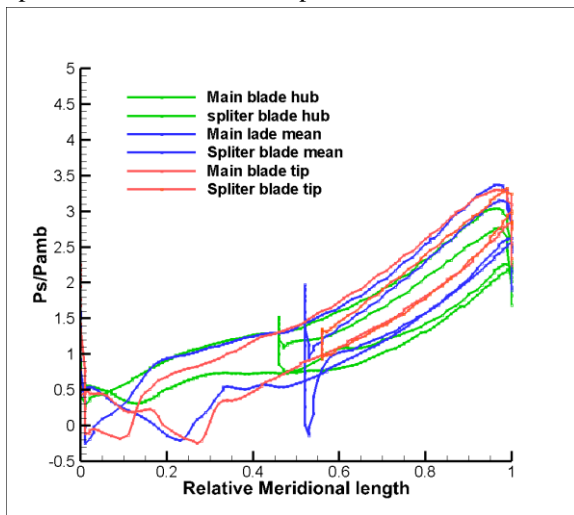


Figure 11: Blade loading at choke point

Diffuser-Coefficient of pressure recovery:

Performance of diffuser is based on the pressure recovery coefficient (CPR). Diagonal diffuser for the compressor stage is designed to perform CPR of 0.6. Numerical results gives the satisfactory CPR of 0.5 at design mass flow and 0.59 at highly loaded condition of stall mass flow at design speed. Diffuser recovering the static pressure with the 9% of total pressure loss. Hence diffuser outlet is providing total – total pressure ratio of 4.75 with absolute mach number of 0.46 and absolute flow angle of 58 deg at the design point.

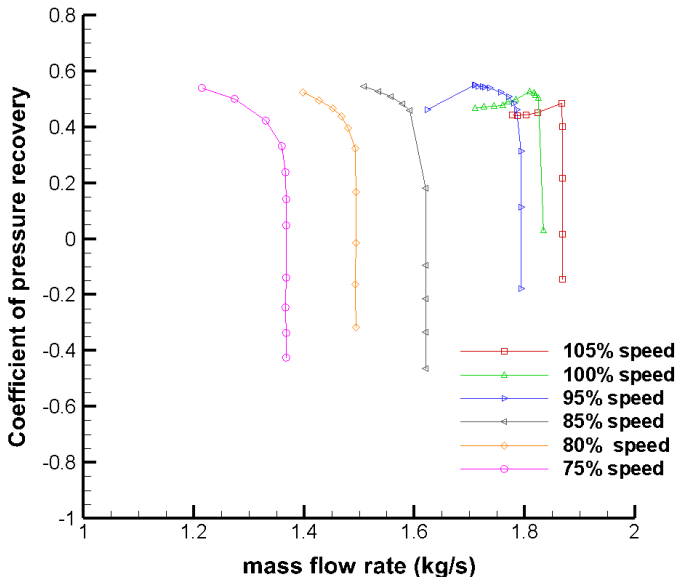


Figure 14: Coefficient of pressure recovery of diffuser

Outlet guide vane – Total pressure loss:

To meet the stage design requirement OGV was designed with high turning angle. Numerical simulation results with average CPR of 0.29 and the total pressure loss of 32%. Pressure loss coefficient of OGV is 0.13. Due to higher losses in the OGV compressor stage results with total – total pressure ratio of 4.42 with absolute mach number of 0.3 and flow angle of 18 deg.

4. Flow Studies at the design mass flow

The effect of incidence on the flow of impeller, stator and outlet guide vane is shown in Figure 16 (a), (b),(c) and (d) at design flow. The following salient point were observed

- Figure 16 (a) discusses about the relative Mach number across the rotor. It is observed that the relative velocity across the impeller is decreasing which shows that there is diffusion across the impeller which contributes to the static pressure rise in the diffusion ratio (w_2/w_1) across the rotor at design mass flow rate is 1.74. It is observed from the Figure 16 (a) that a sharp drop in the relative Mach number at hub is due to the increase in the channel area variation upto 0.4 from design.

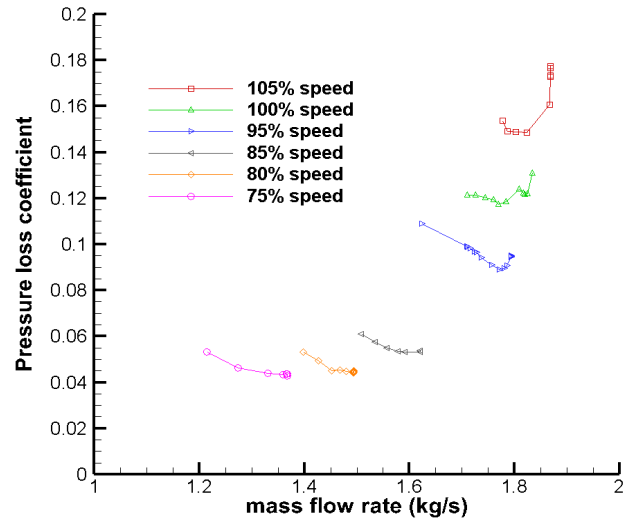


Figure 15: Pressure loss coefficient of OGV

- The relative Mach number at mean and at the tip near to leading edge is found to be greater than 1. The high relative Mach number is causing the shocks.
- The high relative Mach number prevailing at the inlet asks for “unique incidence theory”. If unique incidence theory is applied it will cause a loss in the operating range as the impeller will be very sensitive to the incidence and flow condition at inlet.
- From Figure 16(b) the absolute Mach number for the impeller is observed to be increasing from inlet to exit. The increase in the absolute Mach number is due to the Coriolis force majority of which contributes to the increase in the total pressure across the rotor.
- Figure 16(c) discusses about the diffuser stage performance in which the absolute velocity is decreased which increases the static pressure rise.
- It is observed that due to high curvature in the hub surface of the stage there is separation at hub surface. This flow structure affects the performance of the OGV.
- At OGV it observed that it always operates in the wake of the diffuser separated flow there causing the high total pressure loss which affect the stage performance.

Figure 17 (a) and (b) discusses about the flow in the impeller for the design speed at choke, design mass flow and near to stall. The following points are observed:

- At the inlet plane of the compressor has a high relative tip Mach number of 1.2 and it gradually reduced to 0.5. The absolute Mach number at plane 1 is found to be uniform at near to 0.5.

avoided by providing backward lean to the blade at the tip.

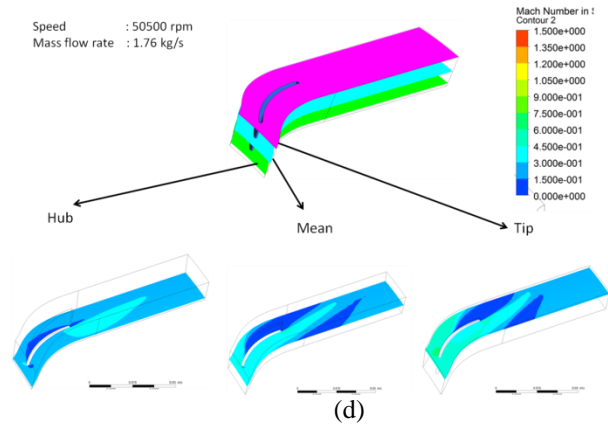
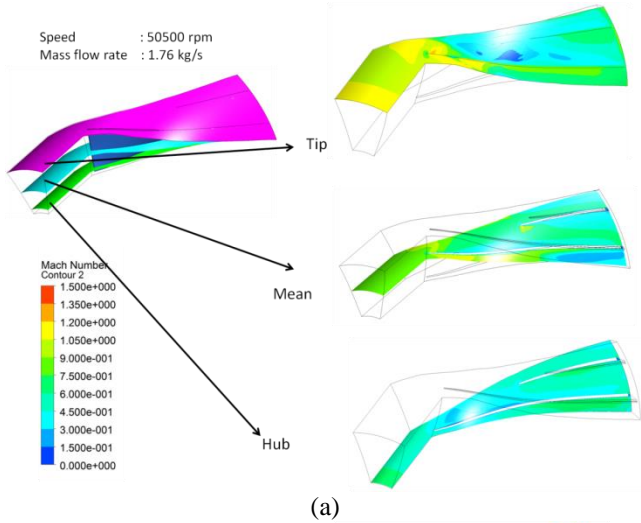
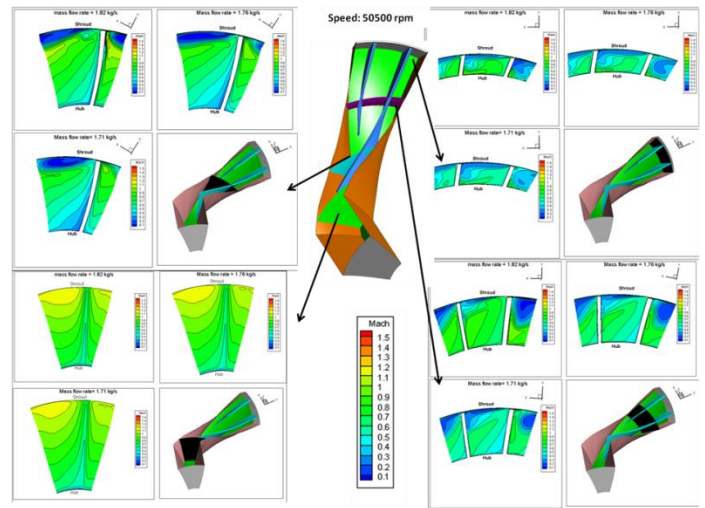
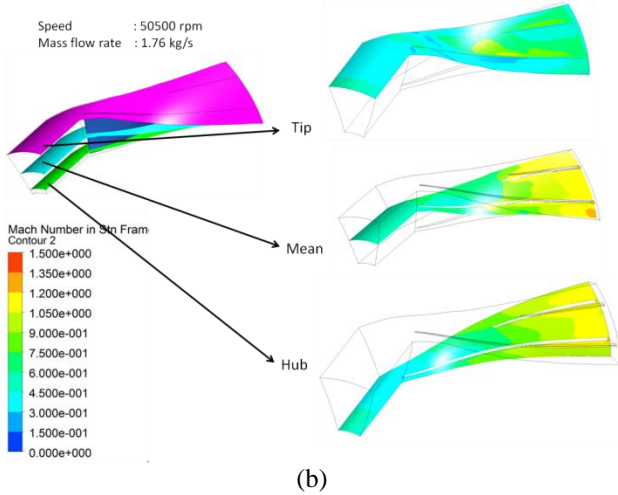
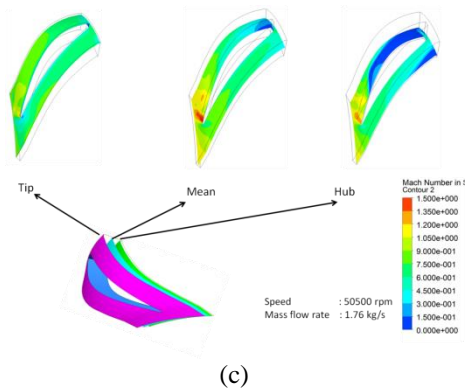


Figure 16: Flow at the design mass flow rate and design



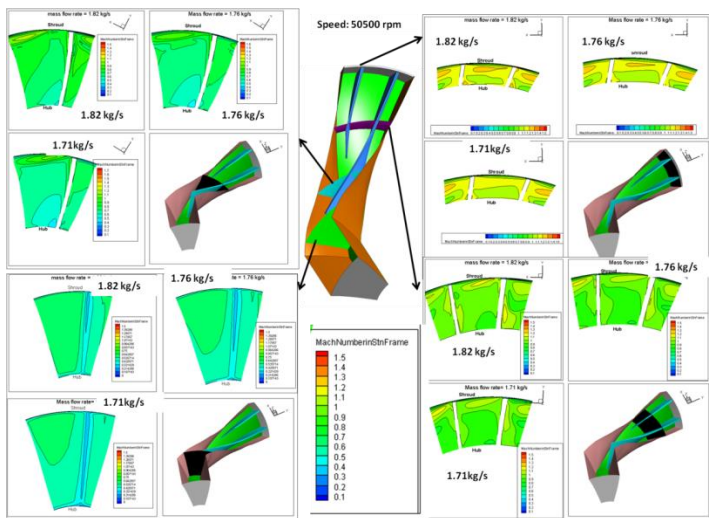
(a) Relative Mach contour at different stations at choke , design and near to stall mass flow rate



- At plane 2 it is observed that due to the pressure difference between the suction and pressure surface, a vortex is generated in the relative frame. This vortex interacts with the secondary flow in the channel thereby affects the performance of the stage.
- In plane 2 it can be observed that the area covered by the vortex is minimum at the design condition compared to the choke and stall. The vortex generated when travels to downstream is split by the splitter blade and it covers one full region of blade.
- The increase in the absolute Mach number is observed in plane 2,3 and 4. At plane4 a localized jet and wake phenomena is observed. These jet and wake can be

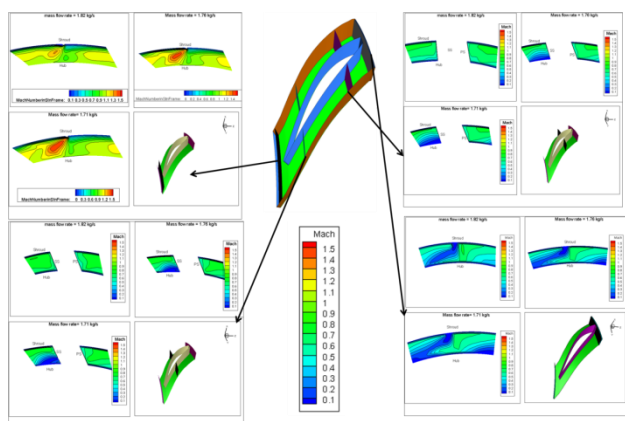
The flow in the diffuser blade and OGV at three different flow conditions is discussed and is shown in Figure 18 (a) and (b).

- There is a decrease in the absolute Mach number observed from plane 1 to plane 4.
- It can be observed that at plane 2 and plane 3 for mass flow rate of 1.76 kg/s and 1.71 kg/s there is a localised separated flow near to the hub. The same is not observed for near choke flow of 1.81 kg/s.

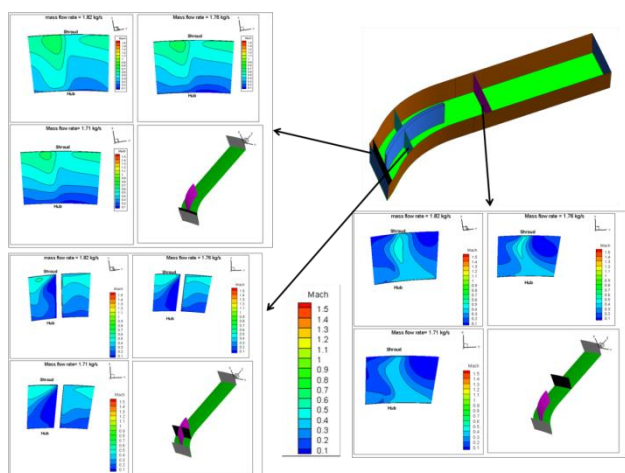


(b) Absolute Mach contour at different stations at choke , design and near to stall mass flow rate

Figure17: Relative Mach number and Absolute Mach number across impeller



(a) Diffuser Flow at three mass flow rate 1.81, 1.76 and 1.71 kg/s



(b) Outlet guide vane flow at three mass flow rate 1.81, 1.76 and 1.71 kg/s

5. Conclusion:

A detailed numerical analysis on the designed mixed flow compressor stage has been carried out. Designed compressor is able to produce total pressure ratio of 4.42 over the targeted pressure ratio of 5 and isentropic stage efficiency can be obtained 73.88% over the targeted value of 80%. In this analysis the detailed reason for drop in performance of the mixed flow compressor is discussed. Due to the non uniform flow and high Mach number at the impeller exit there is a separation seen at the hub surface of the diffuser. Due to this the OGV operates in the wake zone which causes the total pressure ratio loss.

To improve the efficiency following suggestions are made from analysis. By improving efficiency targeted pressure ratio will be achieved with out any maor flow separation zone.

1. Increase the blade angle at hub to bring the incidence at negative zone for better operating range.
2. Relaxing the hub curvature [with increased diameter] of diagonal diffuser will make the flow uniform from hub to tip at diffuser exit. Which also improve the stall margin.
3. OGV redesign should be carried out for uniform flow angle at compressor exit.

6. Reference:

1. CFD analysis of flow through mixed flow compressor under various operating conditions, by D. Ramesh Rajakumar, M.Govardhan, S.Ramamurthy.
2. An Assessment of Computational Fluid Dynamic Techniques in the Analysis and Design of Turbomachinery - B.Lashminarayana
3. Centrifugal compressor: A Strategy for aerodynamic design and analysis By Ronald H.Aungier
4. Centrifugal compressor: A Strategy for aerodynamic design and analysis By Ronald H.Aungier
5. A transonic mixed flow compressor for an extreme duty, Hamiz hazby and Michael Casey, PCA engineers, UK. "Journal of turbomachinery"
6. Design optimization of a mixed flow compressor impeller for a small turbojet engine, Mert cevik and Oguz uzol, "Aircraft engineering and aerospace technology: An international journal".

Figure 18: Flow in Diffuser and OGV at mass flow rates of 1.81, 1.76 and 1.71 kg/s