Proceedings of the International Conference on Aerospace Science and Technology
26-28 June2008, Bangalore, India

INCAST 2008-001

THEORETICAL EVALUATION OF FLOW THROUGH CENTRIFUGAL COMPRESSOR STAGE

S.Ramamurthy[1], R.Rajendran[1], R. S. Dileep Kumar[2]
[1]Scientist, Propulsion Division, National Aerospace Laboratories, Bangalore-560017,ramamurthy_srm@yahoo.com
[1]Scientist, Propulsion Division, National Aerospace Laboratories, Bangalore-560017,raghavanrajendran@gmail.com
[2]Engineer, Quest Pvt. Limited, Bangalore-560 017.rsdileep@yahoo.com

ABSTARCT: In this paper a three dimensional viscous flow field in a Centrifugal Impeller with backswept and lean has been numerically analyzed using a commercial code (FLUENT) to understand the physics of complex real flow phenomena. The study was taken to investigate the flow interaction between impeller and vane diffuser for three different setting angles. It was shown at design point the flow at the exit of the impeller is uniform and the classical jet wake pattern is absent. At the intermediate plane between impeller inlet and exit and within vane diffuser channel the flow behaves like a potential flow. The stage analysis carried out using mixing plane formulation for three diffuser settings provided the optimum diffuser setting angle for maximum stage efficiency.

1. INTRODUCTION

The flow through the centrifugal impeller is complex due to three-dimensional nature of flow channel. There exists tip clearance flow due to clearance between impeller and stationary shroud and secondary flow due to blade curvature. In addition to this jet-wake flow arises due to turbulence and flow stratification associated with blade and shroud curvatures. Computational fluid dynamics has become a powerful tool in the investigation of a flow field inside centrifugal compressor impeller. Numerical flow simulations can throw some light on complex flow phenomenon inside the flow passage.

Using laser velocity-meter Eckardt [(1)] showed a jet-wake pattern at the impeller exit. It was shown by Dean [(2)], the wake formation with high losses affects the operating range of both the rotating impeller as well as stage. Krain [(3,4,5)] showed Jet/wake pattern could be avoided at impeller outlet by proper shaping of the shroud and blade curvature. Numerical flow predictions carried out by Krain [(6)] are very close to the experimental results.

In this paper a three dimensional viscous flow field in a Centrifugal Impeller with backswept and lean has been numerically analyzed using a commercial code (FLUENT) to understand the physics of complex real flow phenomena within the flow channel. The study was also taken up to study the interaction between impeller and vane diffuser for three different setting angles.

2. MODELLING AND ANALYSIS

Figure-1 shows the CAD model of compressor stage generated in unigraphics. In this CAD model the diffuser setting angle is 70° with reference to radial direction. The co-ordinates required for generating the CAD model were obtained from an aerodynamic design of compressor stage. For the analysis a segregated solver was used with steady state, compressible flow. Standard k-e turbulence model was used for computation. Grid independence study was carried out to get the optimum grid for numerical solution to be independent of grid. The grid was generated using Gambit. Figure-2 shows the single flow channel consisting of one impeller channel and one diffuser channel considered for analysis with appropriate boundary conditions. At the inlet boundary total temperature, total pressure and absolute flow angle were specified. At the outlet boundary static pressure is specified. At the periodic boundaries, the flow properties are computed by averaging the properties on either side of the periodic boundaries. In the present study since the model is axi-symmetric the flow through a single blade passage and diffuser passage has been studied, assuming periodicity. For the upstream, vane less space sidewalls periodic boundary condition is used. At the impeller solid walls, no slip boundary condition is used and all the solid walls are assumed to be adiabatic. The hub wall of the impeller is assumed to be moving

Figure-1 CAD MODEL

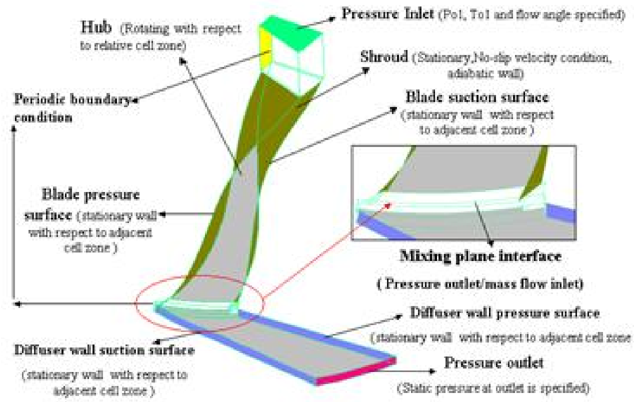


Figure-2 Flow Channel &Boundary conditions

with the rotor blade, while the upstream and downstream hub is made stationary. The whole shroud is stationary with respect to absolute reference frame. At the mixing plane interface pressure outlet and mass flow inlet boundary conditions were imposed. The conservation of mass flow rate across the mixing plane interface was satisfied. Diffuser hub, shroud, vane walls are kept stationary with respect to the relative zone cell zone. Compressor stage analysis using mixing plane was also carried out. Table-I gives the details of CFD analysis carried out for full model and axisymmetric model.

Table -1Comparison of full model analysis with mixing plane model

Parameter	**Full Model**	**Mixing plane model**
Stage Pressure ratio	1.370	1.372
Mass flow rate (kg/s)	0.5320	0.5324
Computational time (hrs)	24	6
Number of grid cells	1200000	240000

3. RESULTS AND DISCUSSIONS

Figure-3 shows the comparison of impeller performance obtained by CFD with experimental results. The total to static pressure ratio and total to total pressure ratio of the impeller was plotted against non-dimensional mass flow parameter for three different speeds. It is observed from this figure

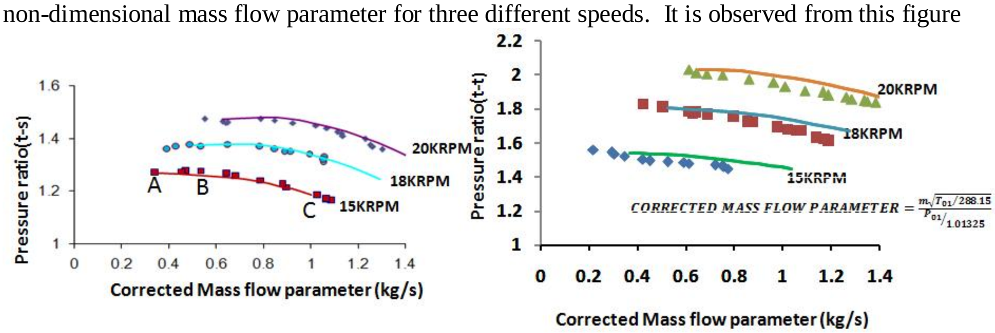


Figure-3 Impeller performance

theoretically estimated value agrees with experimental values. There is large difference in the measured surge point as compared to theory. Three points on the lower speed characteristics namely A, Band C were considered for detailed flow studies. Figure-4 shows the compressor stage performance map plotted using the ratio of diffuser outlet static pressure to inlet total pressure against corrected mass flow parameter for three different diffuser setting angles namely 60°, 65° and 70° at 15000 rpm. It can be seen from this figure that the stage pressure rise achieved is higher for 70° diffuser setting when compared to other two configurations as it matches closely with impeller exit flow angle. However the operating range is lowest as compared to other setting angles of the diffuser.

The maximum mass flow rate passing through the stage reduces with increase in diffuser setting angle due reduction in diffuser throat area. It is also observed from the analysis the stage adiabatic efficiency is higher for largest setting angle of the diffuser. In this diffuser setting both pressure rise and efficiency is higher as compared to smaller setting angle because the fluid leaving the impeller is

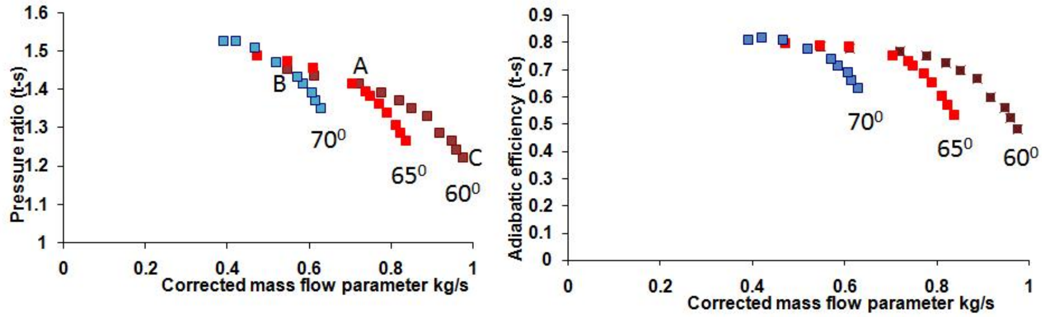


Figure-4 Stage Performance

approaching the diffuser vane at almost same angle with respect to radial direction keeping the losses minimum. The diffuser diverging angle is also more in case of larger setting angle, this help in higher static pressure recovery in the diffuser channel. Operating points near choke, design and near surge were considered on the operating curve to study detailed flow in the compressor stage.

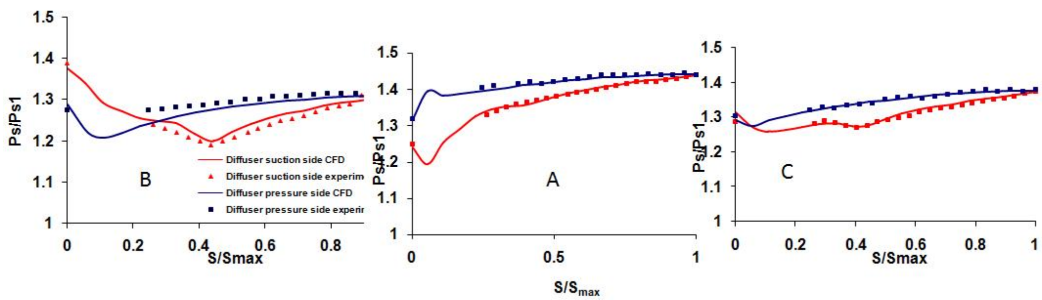


Figure-5 Static pressure distribution on vane diffuser

Figure-5 shows the static pressure distribution along the vane diffuser at three operating points at 15K rpm. In each figure the suction and pressure surface static pressure being plotted against non-dimensional surface distance and compared with experimental points indicated by square symbols. It is observed that at the design operating point the experimental values agrees with the CFD values, whereas close to choke and surge there are small deviations indicating inaccuracy in predictions. There is a cross over in pressure distribution at lower flow rates as the incidence to the diffuser is large positive and a tendency for the flow to separate from the pressure surface. The flow variation within the impeller and diffuser channel is shown in Figure-6. The relative velocity distribution along the meridional planes from impeller inlet to exit and absolute velocity distribution in the diffuser channel from impeller exit to diffuser exit at design point is shown in this figure. It is observed that the flow at the exit of the impeller is uniform and the classical jet wake pattern is absent. This uniform flow has been achieved by suitable shaping of the shroud and providing lean to the blade at impeller exit. This uniform flow entering into the diffuser has minimum distortion, hence improves the flow structure and performance of the diffuser in terms of pressure recovery. In the intermediate plane between impeller inlet and exit the flow behaves like a potential flow. It is observed that at plane 8 which is at semi vane-less portion of the diffuser the flow is uniform. The plane 9 is just after the vane diffuser throat the pressure gradient between suction and pressure surface starts building and the pressure gradient increases as the flow proceeds towards the vane diffuser exit. It is also observed that the pressure gradient is larger at the shroud wall than on the hub wall due to the influence higher blade loading at the impeller shroud.

4. CONCLUSIONS

Stage analysis carried out using mixing plane formulation for three diffuser settings provide the optimum diffuser setting angle for maximum stage efficiency. At low flow rates the flow reversal has been noticed at the inlet tip. Uniform flow is observed at the impeller exit instead of jet/wake phenomenon satisfying the design approach adopted for generating the impeller vane shape

ACKNOWLEDGEMENT

The authors wish to acknowledge Mr. Daniel Manoharan and Mr. S. Thennavarajan, Propulsion Division for assisting the Scientists in the experimental programme. The authors also wish to acknowledge Director, NAL and Head, Propulsion Division for permitting the authors to publish the research work in the proceedings.

REFERENCES

[1] Eckardt D, ñDetail flow investigations within a high speed centrifugal compressorò, Journal of Fluids Engineering, Trans. of ASME, pp 390-402, 1976.

[2] Dean R.C. and Senoo Y., ñRotating wakes in vane-less diffusersò Journal of Basic Engg., Trans. ASME, Sept. 1960, PP. 563-574

[3] Krain H,òSecondary flow measurements with L2F Technique in Centrifugal Compressorsò Agard-68 Symposium, Advanced Technology foe Aero gas turbine components. May 1987.

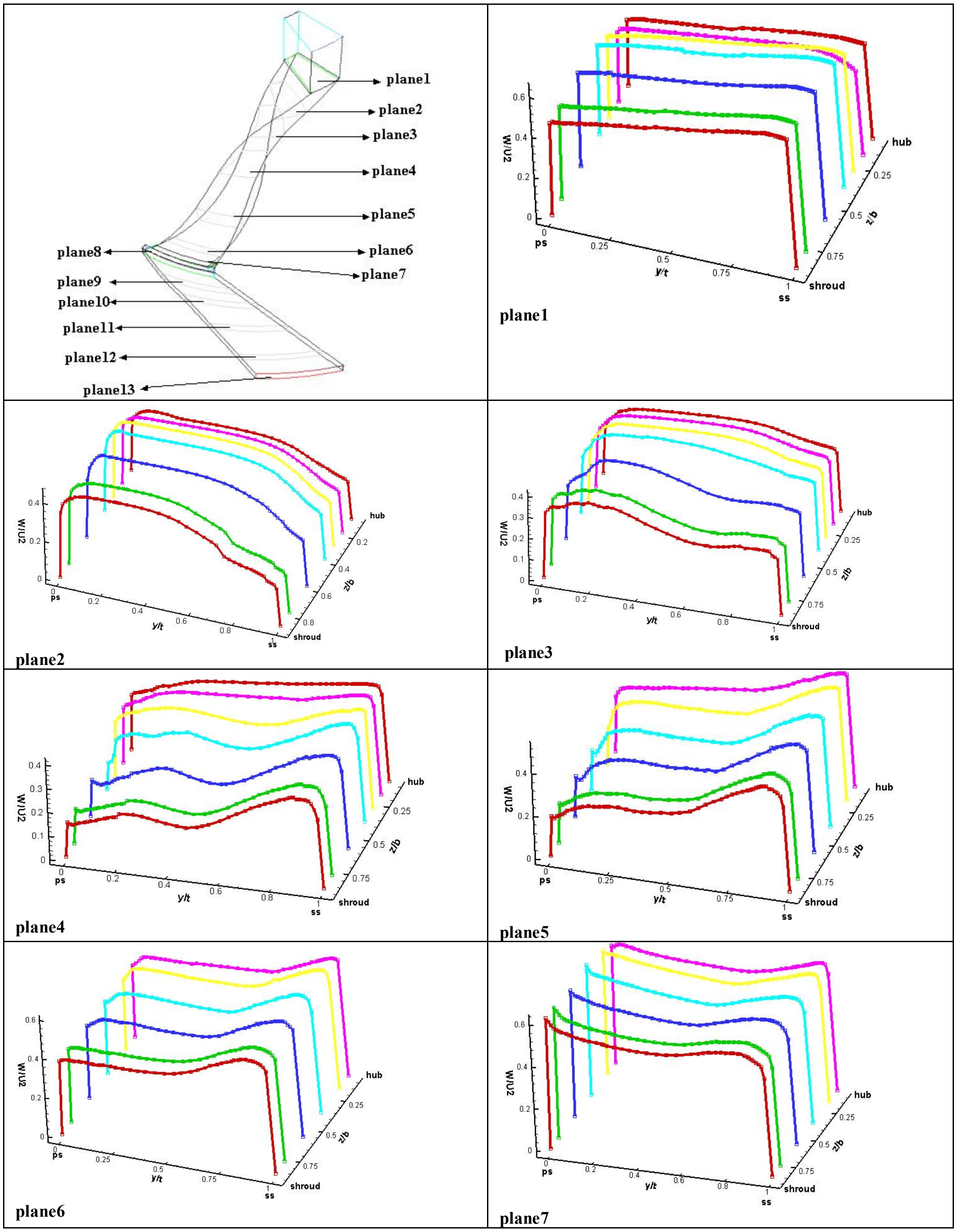
[4] Krain H, ñA study on centrifugal impeller and diffuser flowò, Journal of Engineering for power, Trans of ASME, Vol. 103, pp 688 to 697,1981.

[5] Krain H, ñSwirling impeller flowò, Journal of Turbomachinary, Trans of ASME, Vol 110, pp 122-128, 1988.

[6] Krain H, ñExperimental and theoretical analysis of centrifugal compressor impeller flowò Turbomachinery efficiency prediction and improvement, Proc, Instn. Mech. Engg., 1987.

NOMENCLATURE

b	Impeller axial width
c	Absolute velocity (m/s)
P_{s3}	Diffuser exit static pressure (N/m2)
P_{o1}	Impeller inlet total pressure (N/m2)
ps	Pressure surface
ss	Suction surface
t	Circumferential pitch between blades at exit (m)
U_2	Impeller tip speed (m/s)
w	Relative velocity (m/s)
y	Circumferential distance from the pressure surface (m)
z	Axial distance from shroud (m)



Velocity distribution (W/U2) at operating point B in Impeller Channel

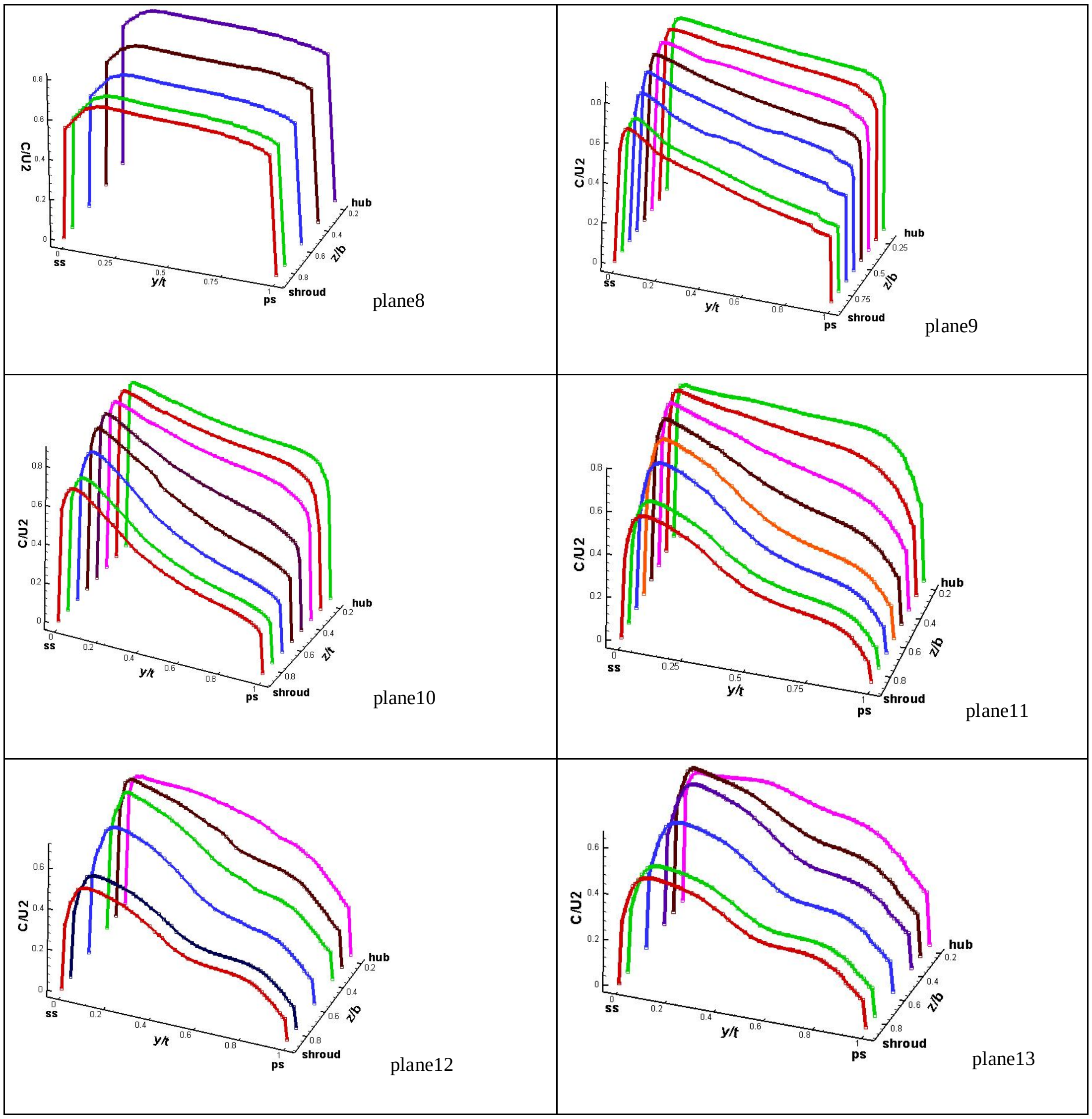


Figure-6 Velocity distribution at design point for compressor stage