IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 20, Issue 1 Ser. I (Jan. – Feb. 2023), PP 20-26
www.iosrjournals.org

Optimization and Numerical Investigation of flow behavior into Inlet guide vane (IGV) for Centrifugal Compressor.

Swanand Deshmukh[1], Viral Kumar Patel[2],Umesh Chavan[1]
[1](Mechanical Department, Vishwakarma Institute of Technology, Pune Maharashtra)
[2](Design & Development Department, Kirloskar Pneumatics Corporation limited (KPCL), Pune Maharashtra)

Abstract: *In a centrifugal compressor adjustable inlet guide vanes (IGV) in front of the impeller are used to regulate the pressure ratio and the mass flow. Aim of this paper to study the pressure drop and coefficient of discharge in plane geometry IGV blade and aero foil shape IGV blade.*

Background: *Currently available geometry of IGV is plane shape. Connection between blade of IGV and Pipe is maintained through lug. Lug diameter is more than thickness of blade that's why some part of lug is coming out of plane of blade, which results into the creation of turbulence. So avoid this problem plane shape geometry is replaced with aerofoil shape such that at maximum thickness lug aero foil blade lug will get cover. Hence streamline whill be maintained and pressure drop will get reduced.*

Materials and Methods *Flow calculating methods in ANSYS CFX were to create models of the internal volume (flow) for each IGV angle, converting the model geometry in ANSYS and formation of the computational grid.*

The boundary conditions for the calculation are: at the entrance of the computational domain (Inlet) there was full pressure; at the output (Outlet) of the transfer tube there was set mass flow; at all other surfaces there was defined a smooth wall without slip stream.

There were calculated 20 IGV rotation angles at the range from 30 to 50° from the initial position of IGV rotation. For each IGV angle flow for three different values of mass flow rate (0.2124 kg/s, 0.4248 kg/s, 0.8496 kg/s) was calculated. The total pressure and temperature on the inlet region are respectively 101.325 Pa and 20°C. The calculation model was used turbulence k-e. The criterion on the convergence of discrepancies was established on 10-6. The calculation was conducted in the CFX-Pre application. The model convergence for the rotation angles majority is achieved in the range of 500 iterations. The calculation results are processed in the program CFD-Post (Fig.5-8), where the velocity vectors were constructed and necessary parameters for loss factor calculating were identified.

Results: *pressure drop in plane shape IGV geometry is 6.25% whereas in aero foil shape geometry it is 3.04% for 0.4248kg/s.*

Conclusion: *Pressure drop in airfoil shape is reduced by 50% compared to pressure drop in plane shape geometry.*

Key Word: *centrifugal Compressor, Inlet guide vane, plane geometry of IGV blade, aero foil geometry of blade.*

Date of Submission: 02-01-2023 Date of Acceptance: 15-01-2023

I. Introduction

Inlet Guide Vanes (IGVs) and Inlet Butterfly Valves (IBVs) perform an important task—to regulate the airflow and pressure that enters a centrifugal compressor's first stage of compression. The IGV has multiple triangular-shaped vanes that allow the air to flow into the compressor in a swirl direction. The "pre-swirled" air reduces the amount of work needed from the main driver to spin the air entering the impeller.

Currently available geometry of IGV blade is plane geometry (fig no 1), which has had a lug to maintain the connection between blade and IGV body. After performing CFD analysis it is found out that turbulence is getting created nearby that lug and it is acting as hurdle for streamline flow (fig no 3). this also becoming major cause of pressure loss.

To solve that problem, current geometry is modified by replacing it with Airfoil shape (fig no2). Reason of choosing airfoil is at maximum thickness of airfoil lug can fitted such that it will come out of plane of blade as like in current geometry of IGV.

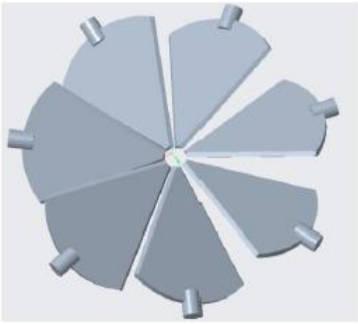

Fig.1 Current geometry of IGV

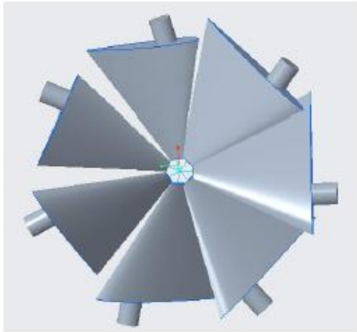

Fig.2 Aero foil shape IGV

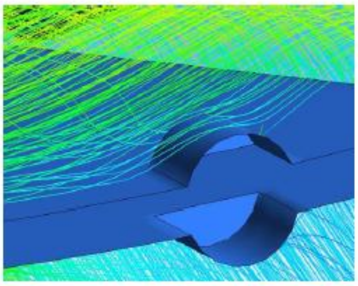

Fig.3 velocity streamline over current IGV blade

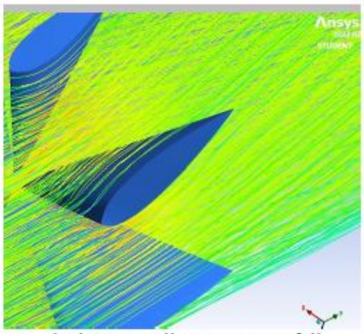

Fig.4 velocity streamline over Aero foil shape IGV blade

Plotting Aerofoil profile

The NACA airfoils are airfoil shapes for aircraft wings developed by the National Advisory Committeefor Aeronautics (NACA). The shape of the NACA airfoils is described using a series of digits following the word"NACA". The parameters in the numerical code can be entered into equations to precisely generate the cross-sectionof the airfoil and calculate its properties.

The NACA 0015 profile is selected, it is a Symmetric Aero foil Profile. The NACA 0015 airfoil is symmetrical,the 00 indicating that it has no camber. The 15 indicates that the airfoil has a 15% thickness-to-chord length ratio: itis 15% as thick as it is long.

where:

$$y_t = 5t\left[0.2969\sqrt{x} - 0.1260x - 0.3516x^2 + 0.2843x^3 - 0.1015x^4\right]$$

x is the position along the chord from 0 to 1.00 (0 to 100%),
$y_t$is the half thickness at a given value of x (centerline to surface),
t is the maximum thickness as a fraction of the chord (so t gives the last two digits in the NACA 4-digitdenomination divided by 100).

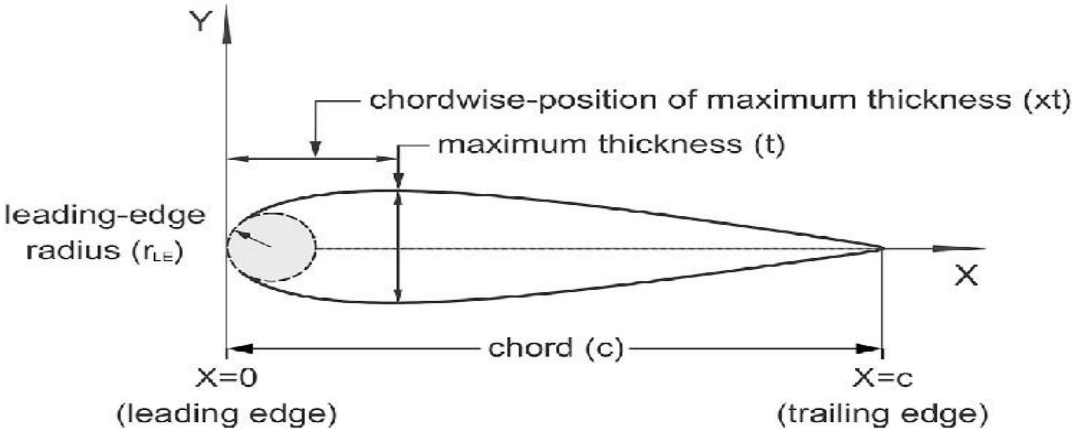

Fig.5 Aero foil terminologies

Section 2 Upper Aero foilSection1LowerAerofoil

Fig.6 Blending of upper and lower aero foil profile an

The following CAD model and Fluid volume for CFD are modeled in Creo parametric 7.0.8.0 student version.

II. Material And Methods

CFD – Computational fluid dynamics

Computational Fluid Dynamics (CFD) is a method for simulating behavior of the fluid flow or the system involving heat transfer, radiation, and so on. This method uses computers for solving special equations over a region of interest with known conditions on the boundaries. The first mentions about CFD were on around 1910; more interest in CFD began to show after the Second World War, however, more practical use came up with the expansion of computer technology at the end of the 80s and early 90s. Since these times CFD has been on a theoretical level. Nowadays, the CFD in one of the basic design tools helping to reduce the design time and to increase effectivity of engineering work. CFD is widely used on fields of power and energy, engine industry, aerodynamics, thermodynamic, aeronautics etc.

Solving The CFD problem

From the global point of view there are 5 steps to reach data and solution from CFD simulation:

1) Defining the geometry of region of interest
2) Setting and generating mesh
3) Solver setting
4) Solving the case
5) Visualizing the results

Fluid Volume Geometry

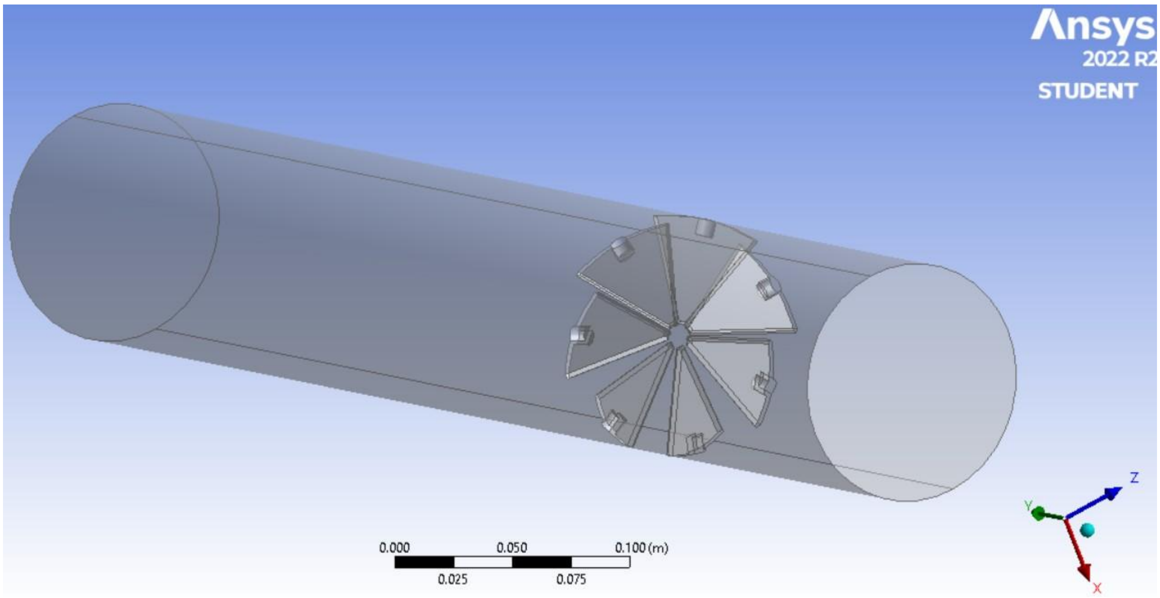

Fig.6 Fluid Volume for The total length of the pipe kept is 700 mm in fluid volume, out of which 500 mm is kept behind the IGV and 200 mm on the front side [4]

Mesh generation

Mesh is prepared is Ansys 2022 workbench student version and for each case mesh was generated separately.

- Face sizing and inflation layer criterion applied.
- For Overall mesh element size kept is 5mm.
- For face sizing element sizing kept is 2mm.
- For inflation layer transmission ratio kept 0.77 and maximum layer kept 5.
- All hexahedral with global quality criterion > 0.4
- First layer distance 0.0001 [m]
- Growth rate: 1.2

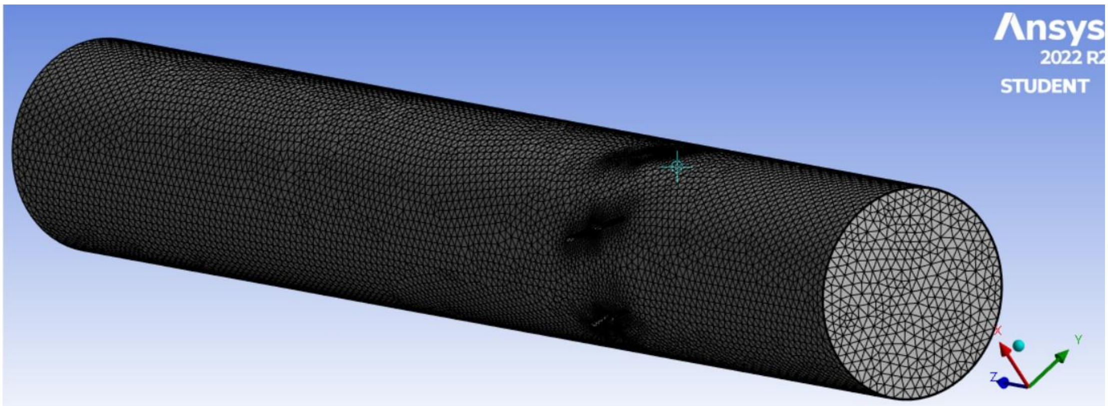

Fig.7 Meshed geometry of fluid volume with 4,92,355 nodes and 19,30,829 elements.

Boundary condition and calculation:

Turbulence model

The Shear Stress Transport turbulence model with an automatic wall function is very well fitted to modeling aboundary layer, which was very important in this case, especially because the “WALL” boundary condition was setto “SMOOTH.” This model gives a highly accurate flow separation prediction and free shear flow prediction farfrom the wall as well. This model is based on a k-ω model, and the proper transport behavior can be obtained by alimiter to the formulation of the eddy-viscosity.

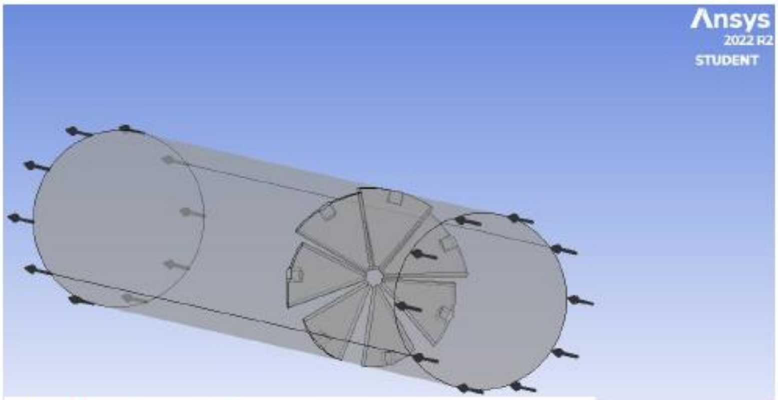

Fig.8 boundary conditions

Boundary conditions

- Inlet:
  - Subsonic flow regime
  - 0.2124-0.8496 [kg. s-1] mass flow rate was tested.
  - medium intensity turbulence (5%)
  - 288 [K] Static temperature
- Wall:
  - No slip and smooth wall
  - Adiabatic heat transfer
- Outlet:
  - Subsonic flow regime
  - Total pressure: – 1atm
- Solver:
  - High resolution advection scheme and first order turbulence numeric
  - 500 iterations per each case with the auto timescale
  - RSM 1e-6 and conservation target 0.000001 was reached

III. Result

Table 1 values of total pressure and pressure drop at outlet for different operating angle at mass flow rateequal to 0.2124kg/s.

Sr.no	IGV blade angle(degree)	P1[pa]	P2[pa]	Dp1(%)	Dp2(%)
1	**30**	100287	100527	1.02	0.78
2	**50**	100918	101154	0.40	0.16
3	**70**	101205	101266	0.11	0.05

Table 2 values of total pressure and pressure drop at outlet of IGV for different operating angle at mass flowrate equal to 0.4248kg/s.

Sr.no	IGV blade angle(degree)	P1[pa]	P2[pa]	Dp1(%)	Dp2(%)
1	**30**	94987.5	98239.2	6.25	3.04
2	**50**	99883.8	100733	1.42	0.58
3	**70**	100909	101111	0.41	0.21

Table 3 values of total pressure and pressure drop at outlet of IGV for different operating angle at mass flowrate equal to 0.8496kg/s.

Sr.no	IGV blade angle(degree)	P1[pa]	P2[pa]	Dp1(%)	Dp2(%)
1	**30**	86987.3	97699.8	14.15	3.57
2	**50**	95058.8	97916.3	6.18	3.36
3	**70**	100873	100556	0.44	0.31

Dp_1(%) = Percentage of Pressure loss in Current IGV.
Dp_2(%) = Percentage of Pressure loss in aero foil shape IGV.

P_1[pa] = Total Pressure at the outlet of Current IGV.
P_2[pa] = Total pressure at the Outlet of Airfoil shape IGV.

IV. DISCUSSION

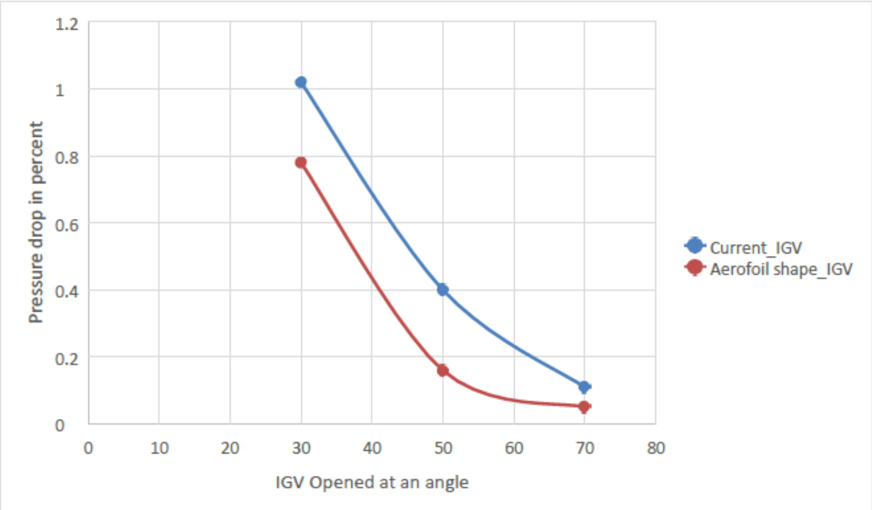

Fig.9 Pressure drop in percent vs IGV blade angle at mass flow rate equal to 0.2124kg/s

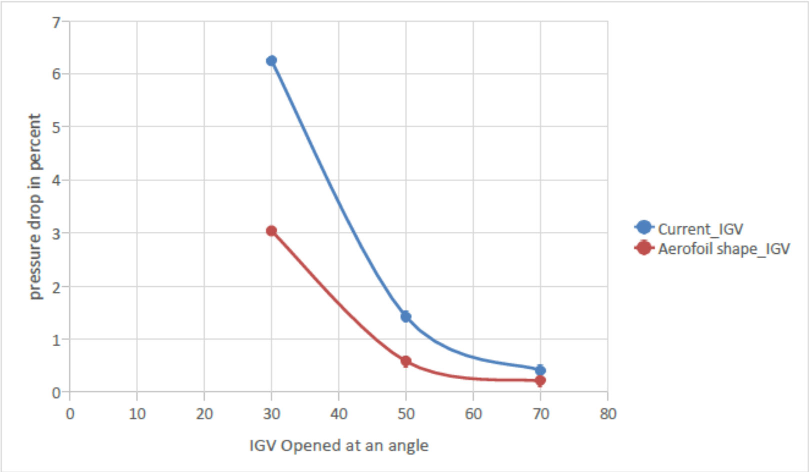

Fig.10 Pressure drop in percent vs IGV blade angle at mass flow rate equal to 0.4248kg/s

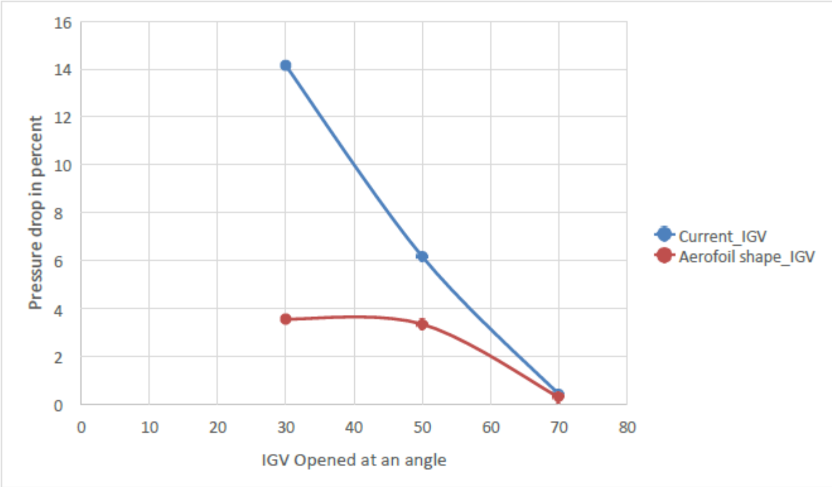

Fig.11 Pressure drop in percent vs IGV blade angle at mass flow rate equal to 0.8496kg/s

After Observing above 3 graph we found out that;
At **constant mass flow**, rate of decrease in pressure drop with increase in IGV angle, decreases for both the type ofblade geometry. For example, for mass flow rate 0.2124kg/s, In Current IGV geometry change in pressure drop inprecent from 30^0 to 50^0 is 0.62 but from 50^0 to $70^0$is 0.29. Similarly in Aero foil shape IGV geometry change inpressure drop in percent from $30^0$to 50^0 is 0.62 but from $50^0$to $70^0$is 0.11.

At **constant blade angle**, rate of decrease in pressure drops with increase in mass flow rate, Increases in CurrentIGV geometry and decreases in Aero foil shape IGV geometry. For example, IGV opened at 30^0, In Current IGVgeometry change in pressure drop in precent from 0.2124kg/s to 0.4248kg/s mass flow rate is 5.23 but from0.4248kg/s to 0.8496kg/s is 7.9. Similarly in Aero foil shape IGV geometry change in pressure drop in precent from0.2124kg/s to 0.4248kg/smass flow rate 2.26 is but from 0.4248kg/s to 0.8496kg/s is 0.53.

V. Conclusion

- Investigating Pressure At outlet of Pipe it found out that, in case of Aero foil shape IGV pressure loss isless than current plane shape IGV blade.
- As we Increase Operating angle of IGV from 30-degree to 70-degree, pressure drop decreases.
- As we Increase mass flow rate from 0.2124kg/s to 0.8496kg/s, pressure drop across IGV increases.
- The reason of Decrease in pressure drop is successful accommodation of lug inside the blade which is disturbing flow path and more smooth streamline flow over the blades of IGV.

References

[1] Hui Cao, Zhengyi Zhou, and Jinhua Zhan(16 May 2017) Aerodynamic design schemes of the inlet guide vane in a core-driven fan stage.
[2] Ming Liu, Lei Tan, and Shu Liang Cao (2018) A review of pre-whirl regulation by inlet guide vanes for compressor and pump.
[3] L Kessens and M. Rutenberg (1998) Flow Measurement Behind The inlet guide vane of a centrifugal compressor.
[4] B Gherman, M Gall, V A Popa and V-A Sterie(2018) IGV position optimization for centrifugal blower.

Swanand Deshmukh, et. al. "Optimization and Numerical Investigation of flow behavior into Inlet guide vane (IGV) for Centrifugal Compressor." *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, 20(1), 2023, pp. 20-26.