Original Paper

Design and Performance Analysis of a Low-speed, High Aspect Ratio Contra-rotating Fan Stage

Chetan S Mistry[1] and Pradeep A M[1]

csmistry@rediffmail.com, ampradeep@aero.iitb.ac.in
[1]Department of Aerospace Engineering, IIT Bombay, Mumbai-400076, India

Abstract

The present paper discusses the design methodology and computational analysis of a contra-rotating fan stage. The fan stages named as rotor 1 and rotor 2, were designed to develop pressure rise of 1100 Pa and 900 Pa, respectively. The design and analysis procedure included a preliminary mean line analysis using the fundamental governing equations. Subsequently, detailed computational analysis of the rotors was carried out using ANSYS CFX ®. Based on the results obtained from the CFX analysis, the baseline mean line analysis was modified to achieve the desired performance of the rotors. A detailed analysis of the various parameters influencing the rotor performance was carried out. The number of blades in each rotor and axial spacing between the rotors were observed to be the significant parameters influencing the performance characteristics. It was observed that increasing the number of blades, reduces the diffusion passage and this can lead to considerable pressure rise up to a certain extent. The pressure rise (as observed from the simulations) is still below the expected design pressure rise of 2000Pa by about 7%. This is probably due to the mixing plane approach between two rotors used for simulations in CFX. It is likely that the data from rotor-1 domain may not correctly be transferred to rotor-2. Secondly, the strong suction effect of rotor-2 may not be captured because of mixing plane approach.

Keywords: *contra rotating fan, aspect ratio, bypass ratio, speed ratio, axial spacing.*

1. Introduction

During last thirty years, the common trend to improve thermal and propulsive efficiency of an engine requires the use of higher turbine inlet temperature and high bypass ratio. This trend has been amplified in past decade by more and more challenging requirements in terms of noise emissions. The current trend is towards reduction of specific fuel consumption and resulting increase in bypass ratio. This leads to significant increase of engine weight as well as nacelle and installation drag. Thus, modern engines that are expected to have higher fuel efficiency, lower emissions, noise, drag, reliability and safety ask for development of new concepts which will take all the above requirements into account.

Both civil and military engines stand to benefit from the use of contra-rotation in terms of better efficiency and improved thrust-to-weight ratios. Higher bypass ratios significantly reduce the jet exhaust noise levels. However, higher bypass ratios also lead to higher nacelle drag and weight. Contra-rotating arrangement of the fans can lead to improved per stage pressure ratio with lower fan diameter. This therefore leads to lower fan tips speeds, besides lower nacelle drag [1].

Only few researches were reported in open literature so far as detailed design is concern [1, 2]. The trend for use of low aspect ratio blades in early 90's concentrated major research conducted on low aspect ratio blades. These research works have been devoted in design aspects and detailed flow field to improve the performance. The studies have demonstrated that the contra rotating fan stage performance is significantly affected by the number of parameters such as speed ratio of two rotors [3, 4, 5], axial gap between them [3, 4] and tip clearance [6, 7]. It was found that with increasing the speed ratio, there is an improvement of stage pressure rise, flow capacity (shifting stall point towards low flow coefficient) [3]. By decreasing the axial gap between two rotors the performance gets improve but it need to be sacrificed for noise reduction [3, 4]. Tip clearance study reported indicates the tip gap of front rotor is not as sensitive as rear rotor. No research has been reported in open literature till the date so far as design, selection of geometrical parameters and performance study of the higher aspect ratio contra rotating fan is concern. Higher aspect ratio leads to smaller chord length and which leads to smaller passage area for diffusion. Thus the design of high aspect ratio contra rotating fan is always challenging. It is believed that present work will provide necessary guidelines for selection of parameters for design and development of contra rotating fan.

Corresponding Author: ampradeep@aero.iitb.ac.in

2. Computational Domain

A commercial software package ANSYS CFX-12 is used for the computational analysis. The CFD simulation is done for one passage through the combinations of both rotors and is assumed to represent the flow situation for the entire rotors combination. Computational domain consisted of single blade flow passage of the axial fan rotor. Figure 1 shows the computational domain which have been used for the analysis. The domain is split into 2 parts, the rotating domain for rotor-1 and the rotating domain for rotor-2. The two domains are connected to each other through "*frozen rotor*" type of interface model. The length of the rotor-1 domain is based on the assumption that the disk on which the rotor will be placed having thickness of 10mm from leading edge of hub and for rotor-1 the inlet is placed at one chord upstream of this distance. Based on the axial spacing between the two rotors (in % of chord) half region is covered in domain-1 for the calculation for rotor-1 exit. Same way for rotating domain for rotor-2 generated using inlet of domain placed exactly at the exit of rotor-1 domain and accordingly modified co-ordinates are inserted for rotor-2 domain. The exit of domain-2 is placed 10mm disk thickness plus 2 chords downstream.

2.1 Grid Generation

Computational grid was generated using commercial code ANSYS Turbo Grid. This grid generator has provision for automated grid generation for a specified geometry. For maintaining the tip gap a feature is given so tip clearance also can be maintained as per requirement. Topology feature is a structure of blocks that acts as a framework for positioning mesh elements. Topology blocks represent sections of the mesh that contain a regular pattern of Hexahedral (Hex) elements. They are laid out adjacent to each other without overlaps or gap, with shared edges and corners between adjacent blocks, such that the entire domain is filled.

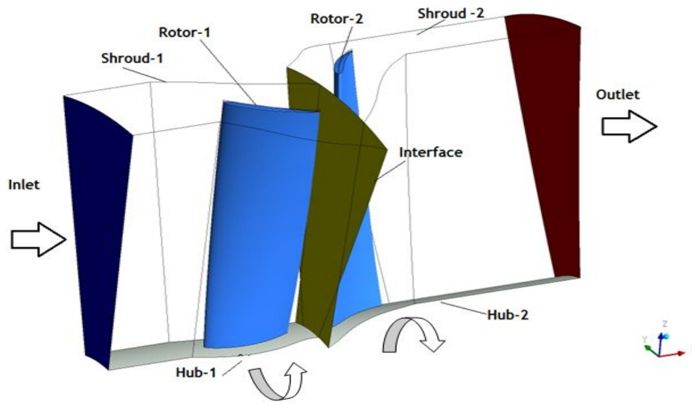

Fig.1 Flow domains with the given boundary conditions

By using topology blocks to control the placement of hex elements, a valid hex mesh can be generated to fill a domain of arbitrary shape. J-grid applies a topology of type J-grid with an optional embedded O-grid that surrounds the blade. The blade surface was surrounded by O-grid to capture the accurately gradients along the surface of the blades. In the span wise and the pitch wise density of the mesh was also maintained in proportion with the stream wise direction. The leading and the trailing edge region of the blades were captured with J grid in the circumferential plane. Both rotating flow domains was captured initially with approximately 3, 50,000 nodes for single domain with sum of 7,00,000 for both domains to get a converged solution.

2.2 Grid sensitivity

A mesh-sensitivity analysis has been performed giving final improved mesh, in which the cell size in the vicinity of the wall allows applying wall functions. The 20,00,000 elements gives y^+ value to be 8.27 and rise of 1850Pa, further increments of no. of elements gives no considerable rise in both values of y^+ and pressure rise.

2.3 Boundary Conditions

The boundary conditions for simulations are given from design data like mass flow, inlet pressure and temperature. There are 2 rotating domains out of which Inlet is specified for rotor-1's domain inlet and Outlet at rotor-2's domain exit. The boundary conditions are specified in the absolute frame of reference. For the rotor domain which is rotating with some angular velocity the boundary conditions are applied in the relative frame of reference that is the rotating frame. At the entry to the Inlet domain total inlet pressure and at the exit mass flow rate condition is specified. Considering the outlet static pressure determines operating status in actual application, so the static pressure outlet condition is preferred for computation but giving flow rate for outlet is recommended for the reason that it will eliminate the cascade incidence loss influence [7].

The k-ω based SST model accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients. One of the advantages of the SST model is its ability to simulate the additional anisotropy of the Reynolds stresses due to the Coriolis forces appearing in the rotating frame of reference [11].

3. Aerodynamic design:

The objective of the present work is to design and develop the test rig to study performance of high aspect ratio, low speed contra rotating fan looking to its feasible application of future military and commercial aircraft. The parametric constrain and chosen for design are as mentioned in table-1. For design the inlet guide vanes and stator at exit are omitted. Thus, the design of contra rotating fan is more complicated compare to conventional fan because to introduction of second rotor with its tip speed and work coefficient as an additional parameter. The design of contra rotating fan is such that rotor-1 is highly loaded compare to rotor-2 as it permits higher overall pressure rise and in addition while off design conditions the rotor-1 remains always stall free [3].

Table 1. Parametric constrains and chosen for design.

Parametric constrain for design		Parameters chosen for design	
Maximum diameter of casing	406 mm	Aspect ratio	3
Power available	15 kW	Maximum thickness of the airfoils	10% Chord
Fan speed (Maximum design)	2400 rpm	Efficiency of 1st rotor	85 %
Mass flow rate	6 kg/s	Efficiency of 2nd rotor	85 %
Pressure rise required from first runner	1100 pa	Mechanical efficiency	75 %
Pressure rise required from second runner	900 pa	Tip diameter	405 mm

The preliminary design of rotor-1 was attempted at constant mean diameter; design parameters like flow angles, velocities, diffusion factor, degree of reaction, power required etc were calculated using conventional design approach for single stage compressor/fan. The parameters at other radial locations were then determined from this mean diameter values. The design attempted for rotor-1 based on variable work loadings at different 20 blade sections to get desired pressure rise. The assumption was made that rotor-2 receives air at an absolute velocity and assumed to be discharge axially. The whirl component from rotor-1 will be transfer to rotor-2 which is in opposite direction to rotation which forces rotor-2 to do additional work and thereby giving the advantage in terms of higher relative velocity at rotor-2 inlet as shown in Fig. 2. The axial velocity and peripheral speed of rotor-2 are same as rotor-1. Using same design approach the rotor-2 was designed for constant mean diameter for rotor-2. The preliminary design parameters like flow angles, camber angles, stagger angles, degree of reaction , diffusion factor, power required etc derived at hub, mean and tip sections are as mentioned in table-2.

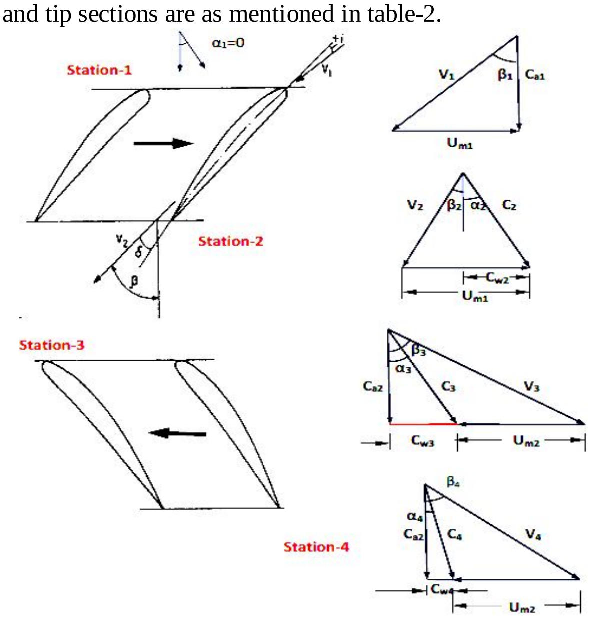

Fig 2 Velocity triangle

Table 2. Geometrical parameters at hub, mean and tip section for rotor-1 and rotor-2

Rotor-1			
Station	Hub	Mean	Tip
β_1	20.90	37.37	48.89
β_2	-19.66	1.5	23.35
DF	0.18	0.5	0.63
DOR	0.032	0.517	0.688
Incidence angle	2	0	-2
Chamber angle	41.35	39.85	31.53
Deviation angle δ	17.85	24.46	21
Corr. δ	2.8	4	4
Stagger angle ζ	-1.7	17.43	35.11

Rotor-2			
Station	Hub	Mean	tip
β_3	48.27	56.34	61.73
β_4	20.57	39.82	50.65
DF	0.427	0.483	0.505
DOR	0.02	0.56	0.72
Incidence angle	2	0	-2
Chamber angle	28.69	24.51	20.75
Deviation angle δ	3	7.5	7.67
Corr. Δ	3	7.5	7.67
Stagger angle ζ	30	44.08	53.36

3.1 Selection of number of blades:

Generally for design of axial compressor or fan, a prime number is selected for number of blades. Initially for preliminary design the number of blades for rotor-1 and rotor-2 was assumed to be 11 and 12 respectively and based on available literature axial spacing was assumed to be 50% of chord [3, 4]. The computational study was done initially for total pressure rise of 1750 pa for different number of blades combination is as shown in Fig. 3.Higher aspect ratio blade gives lower chord length. The overall length available for diffusion will therefore be less. For a combination of higher aspect ratio blades and lower number of blades, a comparatively limited flow passage area for diffusion for both rotors flow domain will be available and thereby giving lower pressure rise than the expected which can be clearly visible from pressure plots as shown in Fig. 4(a) and (b). As the number of blade increases; the flow angles and thereby domain shape changes which give higher diffusion of flow and pressure rise. With lower number of blades for rotor-1, pressure rise achieved was lower and in combination of both rotors thereby giving lower overall pressure rise as contra rotating fan acts like two fans that are connected in series for pressure rise [3]. By increasing number of blades for rotor-1 to 19 resulted in a pressure rise close to the desired pressure rise. Still higher number of blade gave marginal improvement in performance and also it was constrain by the diameter of rotor disk. After doing number of such iterations for different blade number combinations, the final blade combination was found to be 19-17 respectively. Computational study thus gives an idea to study flow field, to shape desired flow passage and accordingly to make choice for number of blade selection.

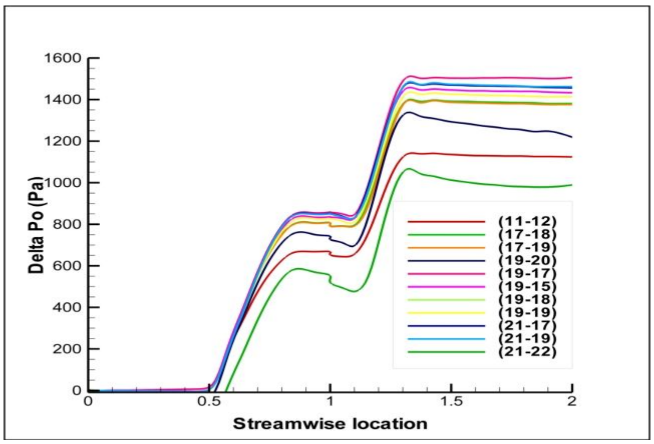

Fig 3 Effect of number of blades on pressure rise

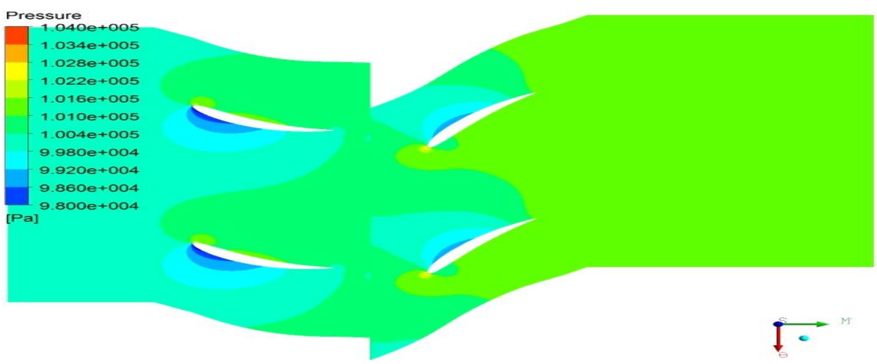

Fig 4(a) Pressure contours plot at 50% span for 11-12 combination

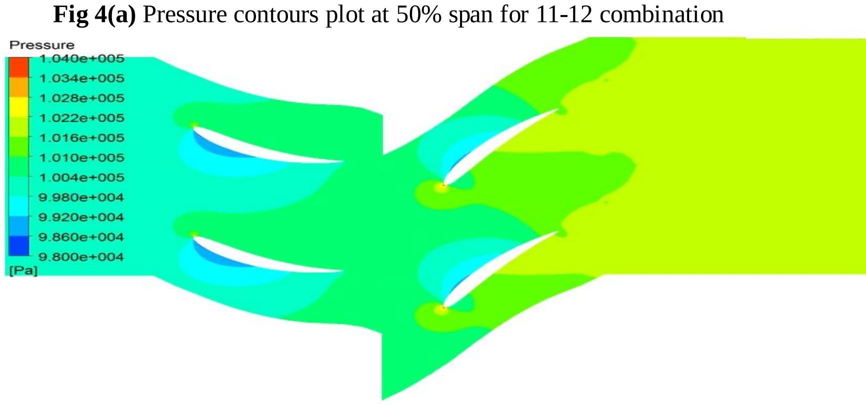

Fig 4(b) Pressure contours plot at 50% span for 19-17 combination

3.2 *Incidence and deviation angle:*

Incidence angles for both the rotors were assumed to be varied from $+2^0$ at the hub,-2^0 at the tip and zero at the mean radius. This is to avoid the risk of tip stall during throttling due to large incidence occurring at tip. This effect is less prominent at the hub due to smaller value of inlet blade flow angle and lower rotational speed.

The flow deviates from the design outlet blade angle as the flow needs to turn through the full angle required by the shape of the blades. For the conventional fan/compressor designs the blade deflection angle depends on blade chamber angle, pitch/chord ratio, shape of the chamber line and air outlet angle.

For contra rotating fan, the fluid coming out from rotor-1 is directly transferred to rotor-2 by strong suction generated by rotor-2. In this case, conventional design approach cannot be used for calculation of the deviation angle for rotor-1. Then the question arises how to choose the deviation angle such that the flow will smoothly be transfer to rotor-2 without separation? From the computational flow field study for the entire flow domain of rotor-1, it was found that the presence of low momentum fluid near trailing edge of rotor-1 changes the deviation angle, thereby changing the incidence to rotor-2 and deteriorating the performance of rotor-2.

Computations suggests that the deviation angle of 4^0 for the whole span avoids the flow separation problem and improve the performance of rotor-1 considerably but by a smaller value for rotor-2 as shown in Fig. 5(a), (b), (c) and (d). Cp plots indicate that the modification in deviation angle of rotor-1 has direct influence on the overall performance of rotor combinations. Due to deviation angle modifications the flow gets accelerated for a small extent near the leading edge of rotor-1 thereby giving more deceleration along the whole blade length, improves diffusion and the loading capacity of rotor-1. The incidence angle of rotor-2 will get modified and the flow behavior improves for rotor-2 flow passage, as it gets decelerated more compared to the earlier case. This also resolves the problem of flow separation in rotor-2. The overall improvement in performance clearly reflected from Cp plots. Lower region of rotor-2 nearly at 30% span shows the deterioration in performance because of the separation problem near hub which was rectified using the change of deviation angle of rotor-1.

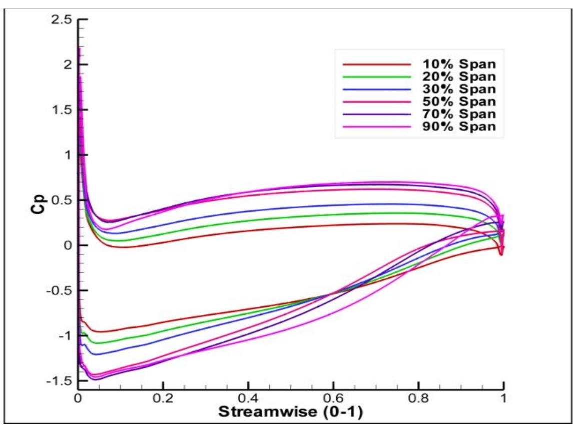

Fig 5(a) Cp plot along of rotor-1 with calculated values

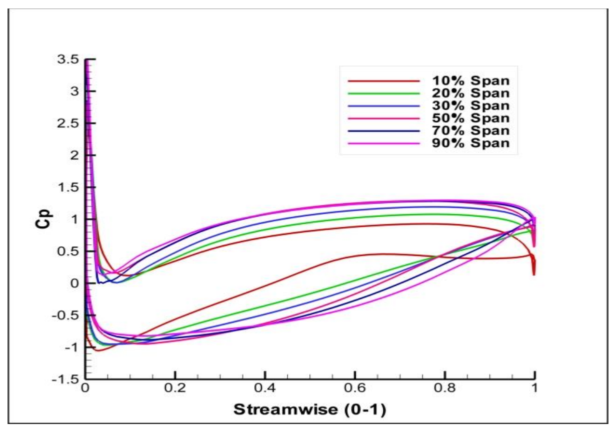

Fig 5(b) Cp plot along of rotor-2 with calculated values

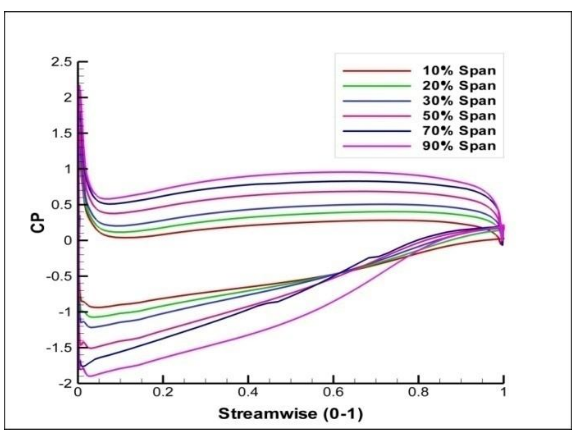

Fig 5(c) Cp plot along of rotor-1 with Deviation change

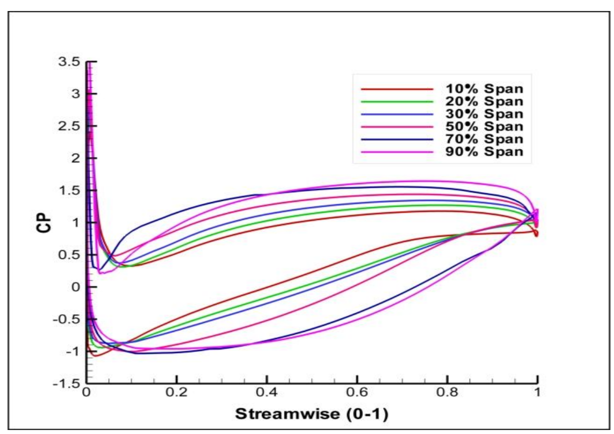

Fig 5(d) Cp plot along of rotor-2 with Deviation change

Fig 5. Comparison of Cp plots for rotor-1 and rotor-2 without and with deviation angle change.

3.3 Blade profile:

The airfoils selected for both rotors belong to C4 family and are fairly suitable for low speed applications. For C4 airfoil, the maximum thickness occurs at 30% of chord and maximum thickness selected for current application is 10% of chord and maximum chamber at 40% chord. The airfoils were stacked in such a way that the CG of each airfoils lie along the same axis.

The flow analysis for combination of 19-17 blades and with modification of deviation angle was still not giving the designed pressure rise. Velocity contour plots indicate the presence of lower momentum fluid presence in rotor-1 flow domain which needs to be eliminated without changing the chamber and stagger angles. The blade profiles were modified accordingly and the best combination was found. It was found that by changing the overall blade profile dimensions; there is a significant improvement in performance. The modified profiles for rotor-1 and rotor-2 are as shown in Fig. 6(a) and (b).

The important observation from this blade profile modification is that the blade profile must maintain a proper acceleration and deceleration of flow along the flow path. The reduced dimension at leading edge of rotor-1 clearly indicates the slight acceleration of flow up to certain extent say 5 to 8 % of chord and subsequent deceleration leads to a smooth diffusion process. At mid region of flow needs to take care of smooth deceleration of flow up to trailing edge. The smooth exit of flow was already incorporated by proper selection of deviation angle at exit.

Since rotor-2 suction surface moves in the opposite direction to rotor-1 suction surface, the suction effect of rotor-2 pulls the rotor-1 wake along with it in a direction opposite to that of rotor-1 and thus tends to suppress separation and lower the deviation angle. Flow behavior in between rotors and towards rotor-2 complicated and difficult to compute. Flow coming out of rotor-1 gets accelerated because of additional swirl component and thus subjected to higher relative velocity at entry of rotor-2; by guiding the flow in good manner it is possible to get good diffusion in rotor-2 flow passage. The blade profile of rotor-2 was modified by changing the thickness of profile and radius at leading edge to accommodated highly accelerated flow to be diffused in controlled manner such that the flow needs to be accelerated marginally at leading edge and direct the flow to downstream throughout the blade passage without separation at trailing edge. Thus, by proper profiling of both the rotors, the flow passages gets modified and allows more diffusion to be done and thus overall improvement in performance of combined rotor stage as shown in Fig. 7(a) and (b) as compared to Fig. 5(a) and (b).

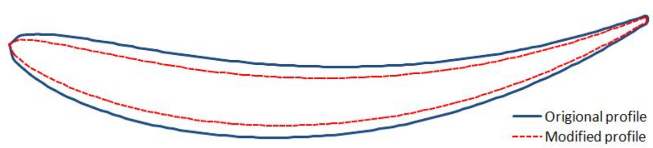

Fig 6(a) Modified profile for rotor-1at hub

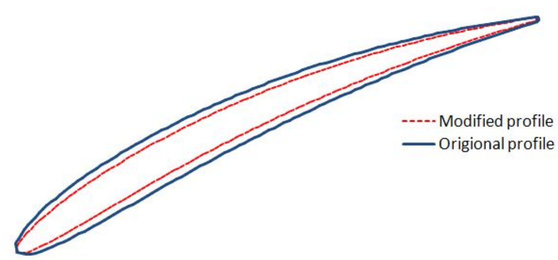

Fig 6(b) Modified profile for rotor-2at hub

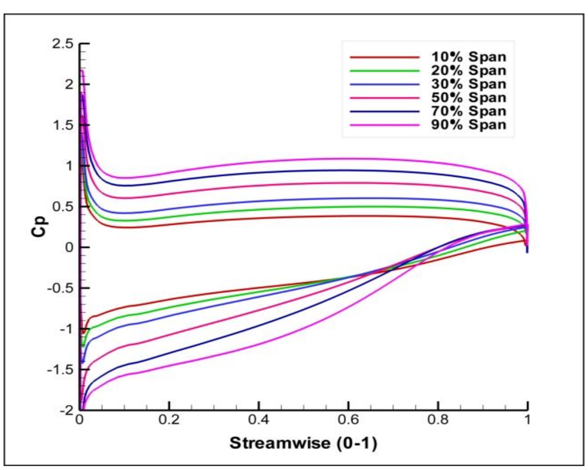

Fig 7(a) Cp plot along of rotor-1 with modified profile

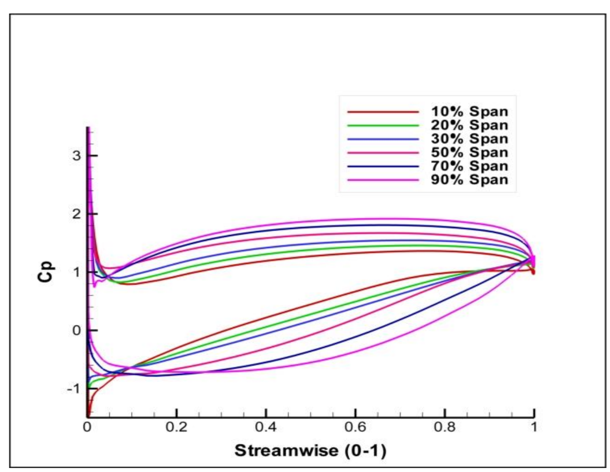

Fig 7(b) Cp plot along of rotor-2 with modified profile

Fig 7 Cp plots after profile modification for rotor-1 and rotor-2.

4. Results and discussion:

For counter rotating fans the wake of the first rotor basically is characterized by a deficit in the absolute velocity in the wake region. When impinging on the leading edge of the second rotor, this causes the inflow to the second rotor to be at positive incidence for the short period of time. A small region of high pressure develops at the pressure surface of the second rotor migrating downstream in time with higher velocity than wake itself. At the suction surface of second rotor one can observed a very similar effect. A region of low pressure develops at the time when rotor-1 wake impinges on the leading edge because of additional expansion on the suction surface caused by the second rotor and migrates downstream in time with a higher velocity near suction side rather than the pressure side. Rotor-1 wakes are not as obvious in rotor-2 outflow as in inflow. A reason for this is that the wake is being transported with a higher velocity near rotor-2 suction than on pressure side as rotor-2 blade suction as it moves in opposite direction to the rotor-1 suction surface, and thus the suction effect of rotor-2 pulls the rotor-1 wake along with in the direction opposite to that of rotor-1 and thus tend to suppress separation and lowers the deviation angle. This causes the rotor-1 wake being split apart in two parts, one arriving earlier at an axial position behind the second rotor as the other one. This can be observed by following plots.

The plots for various velocity components at 5mm upstream of Rotor-1 Leading edge, 5mm downstream of rotor-2 trailing edge, 5 mm upstream of rotor-2 Leading edge and 5mm downstream of rotor-2 trailing edge were plotted to compare the values with designed parameters which are as shown in Figures 8, 9 and 10 at design mass flow rate and speeds combination,

4.1 Axial Velocity:

As shown in Fig. 8 the axial velocity at rotor-1 inlet was found to be decrease due to flow separation and hub vortex by rotation of the front rotor near the hub. At mid section, the axial velocity increases. Near the tip region, the axial velocity decrease due to the boundary layer effect of casing and tip leakage flow.

Axial velocity plot at the exit of the rotor-1 shows velocity deficit due to losses can be observed. At mid stream the losses are dominated by blade profile losses. Close to the outer casing, additional losses due to leakage flow of blade-tip gap get added.

At the entry to the rotor-2, the axial velocity reduces because of the presence of the hub vortex. At the tip section, the axial velocity is decreased in the form of reverse flow by the tip leakage flow and tip vortex of rotor-1.

At the exit of rotor-2, the axial velocity decreases due to lower work transfer because of lower rotational speed. This leads to separation of flow up to 20% of span, which is the probable reason which brings the reduction in performance in terms of pressure rise.

4.2 Absolute and Relative velocities:

Comparison of the designed and computational velocity plot indicates that the rotor-1 inlet conditions exactly matches. That can also be seen from flow angle plots as shown in Fig. 9(a) and (b) at design mass flow rate. Relative velocity at the exit of rotor-1 computationally found higher because of counter rotation of rotor-2 higher swirl will be generated and that transferred toward rotor-1 will give rise in absolute and relative velocity rise. Fig. 9(a) and (b) shows greater variation flow angles from the design value because of the presence of rotor-1 exit wake and along with suction effect of rotor-2 thereby changing both deviation angle of rotor-1 and incidence angle of rotor-2 and this effect is found to be significant from mid section to tip region.

As mentioned earlier, the change of incidence angle to rotor-2 will bring change in velocity plots. The lower axial velocity near the hub region directly affects the exit relative and absolute velocities. Because of lower relative velocity at exit near hub region results into less flow turning thereby give rise separation near that region.

4.3 Circumferential velocity:

As shown in Fig. 10 the circumferential component has an exit swirl in the direction of rotation which is slightly decreased close to the casing at rotor-1 exit. The velocity is increased in magnitude due to swirl by the rotation of front rotor but it is having opposite direction so it shows different color plot as shown in Fig. 10. The lower value of swirl at the exit which indicates good design of rotor so far as the exit is concern, it shows axial exit.

4.4 Radial velocity:

Radial velocity components are relatively low in magnitude both upstream and downstream of rotor-1 and rotor-2.

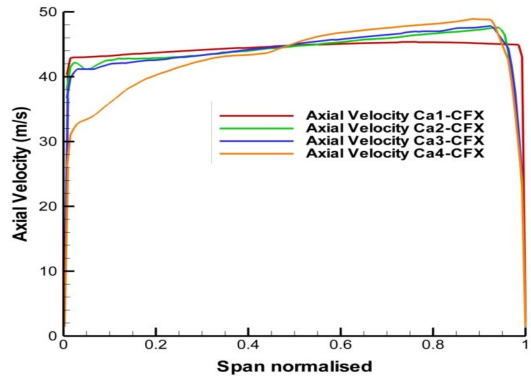

Fig 8 Computational Axial Velocity at upstream and downstream of rotor-1 and rotor-2 at 6 kg/sec

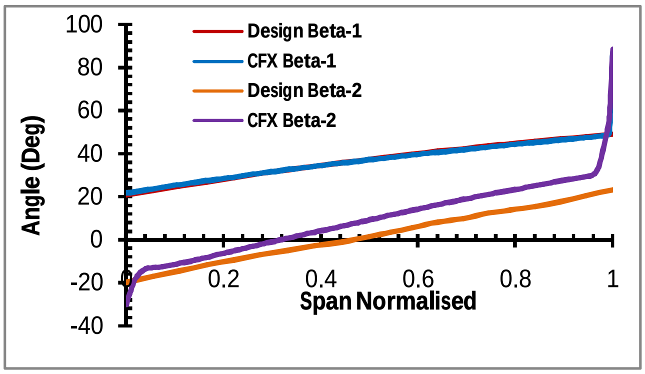

Fig 9 (a) Velocity flow angle at rotor-1 inlet and outlet

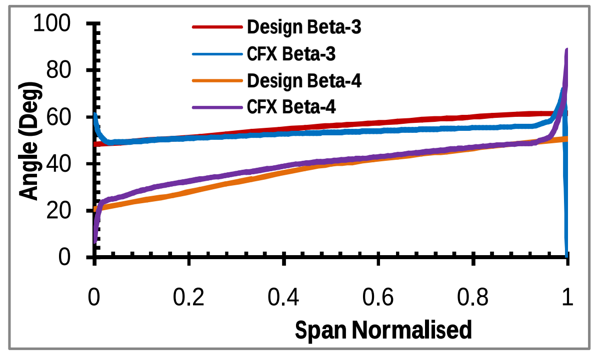

Fig 9 (b) Velocity flow angle at rotor-2 inlet and outlet

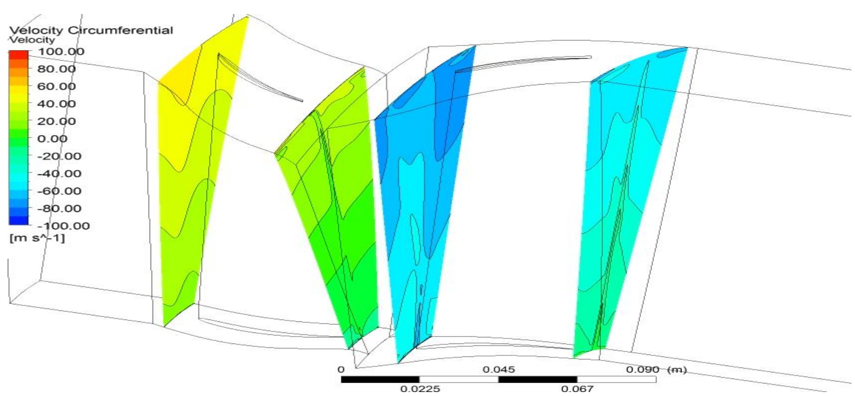

Fig 10 Circumferential Velocity contours upstream and downstream of rotor-1 and rotor-2

The performance of the contra rotating fan is similar to conventional compressor/fan. The advantage of contra rotating fan is it gives wider operating range under off design conditions. While designing it was ensured that rotor-1 will always be highly loaded compared to rotor-2 such that for lower mass flows, the rotor-1 will always remains stall free. The performance of contra rotating fan depends on several parameters like axial spacing between rotors, speed ratio combination, tip gap combination etc. For this paper performance study was done for design speeds and constant axial spacing of 50% of chord as initial base for development of contra rotating fan rig. The performance study was conducted using variable mass flow conditions at exit of rotor-2.

For off design flow conditions such as higher mass flow conditions the total pressure rise is lower for both the rotors which is because of higher velocity components as shown in Fig. 12(c). These higher velocities change the flow angle at the exit of rotor-1.The change in flow angle changes the upstream flow conditions for rotor-2. For higher mass flow condition as shown in Fig. 11 the total pressure was found to droop drastically at rotor-1 exit and then again rises by opposite rotation effect of rotor-2 and as an overall effect the performance deteriorates. Slightly higher mass flow rate from design condition changes whole flow structure for contra rotating fan and thereby the overall performance was affected.

However, for lower mass flow rates the, situation is somewhat different. Near the design mass flow condition, the flow strictly follows the design flow conditions. For lower mass flow rates, the important observation was found that for wider range of flow conditions the total pressure rise was found to be constant which gives more flat operating characteristic at different lower mass flow conditions.

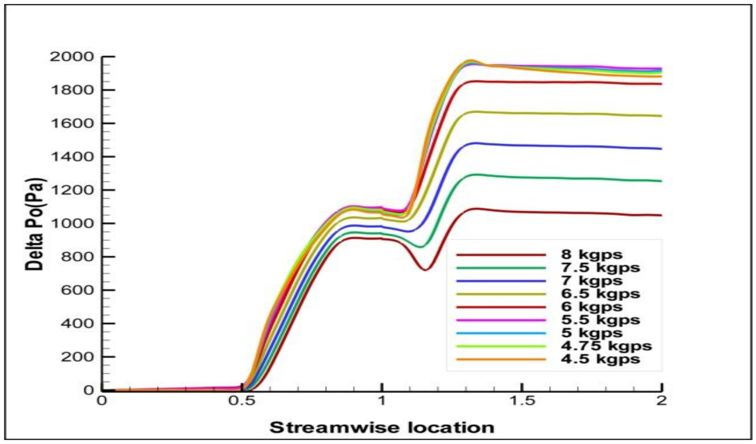

Fig 11 Total pressure rise across the rotors

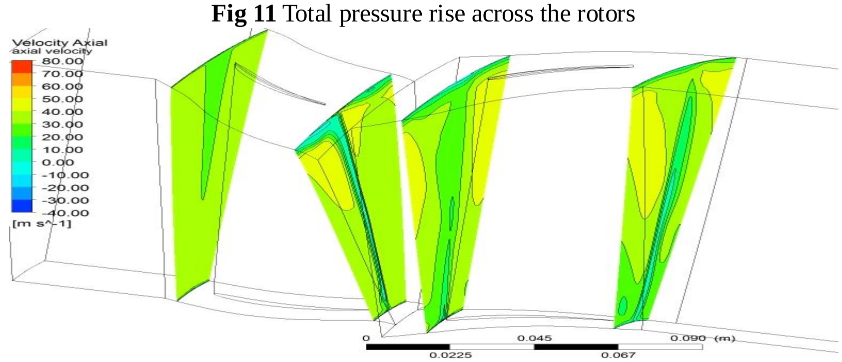

Fig 12 (a) Axial Velocity contours upstream and downstream of rotor-1 and rotor-2 at 4.5 kg/sec

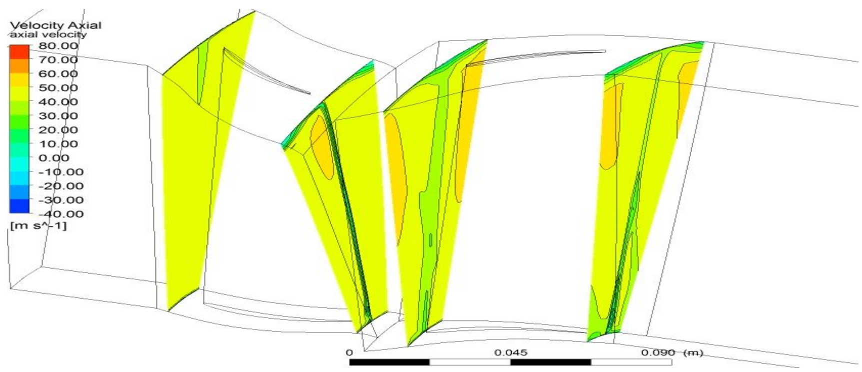

Fig 12 (b) Axial Velocity contours upstream and downstream of rotor-1 and rotor-2 at 6 kg/sec

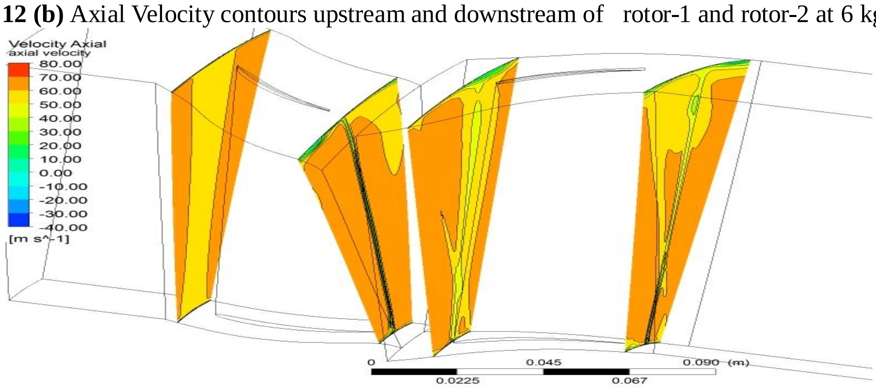

Fig 12 (c) Axial Velocity contours upstream and downstream of rotor-1 and rotor-2 at 8 kg/sec

The comparatively wider size flow wake coming out from rotor-1 will get sucked by suction of rotor-2 at nearly design incidence and higher relative velocity as a net effect the performance of rotor-2 improves which is as shown in Fig. 11.

Conclusion

The present paper discusses the design methodology and computational analysis of a contra-rotating fan stage. The fan stages were designed to develop pressure rise of 1100 Pa and 900 Pa, respectively. The design and analysis procedure included a preliminary mean line analysis using the fundamental governing equations. Subsequently, detailed computational analysis of the rotors was carried out using ANSYS CFX ®.

Some of the important conclusions derived based on present computational work are summarized as below,

1. Computational study for number of blades on both rotors suggests that the selection of this parameter has considerable influence on performance (Pressure rise) of high aspect ratio blades. Based on this study, the number of blades combination was found to be 19-17.
2. Axial spacing studies show that the stagger angle and the camber angle have a direct effect on the axial spacing. The selection of deviation angle of rotor-1 has a direct effect on the performance of contra rotating fan and this agrees well with the B Roy flow model [3]. The present studies for baseline designs have been conducted by considering 50% of chord axial spacing. Comparison of velocity plots from computational study and design values shows great variation in axial velocity near the hub region for both the rotors because of lower work transfer which lead to flow separation and hub vortex by rotation of rotors.
3. The expected pressure rise is still below the design pressure rise of 2000Pa by 7%, which may be because of mixing plane approach between two rotors and it is presumed that the data from rotor-1 domain may not correctly be transfer to rotor-2. Secondly the strong suction effect of rotor-2 may not be captured because of mixing plane approach.
4. Performance plot suggest the pressure rise characteristic of contra rotating fan is similar to conventional compressor/fan. The important result that was found that over wide range of lower mass flow rates the contra rotating fan gives more flat operating characteristic and rotor-1 remains stall free.

It is believed that present work will provide necessary guidelines for selection of parameters for design and development of low speed high aspect ratio contra rotating fan. The computational results need validation from experimental data, which is currently underway at IIT Bombay.

Nomenclature

C	Absolute Velocity	α	Absolute flow angle
C	Chord	β	Blade angle
C_a	Axial Velocity	δ	Deviation angle
DF	Diffusion Factor	θ	Camber angle
DOR	Degree of Reaction	ξ	Stagger angle
i	Incidence Angle	Subscript	
P_0	Total pressure	1	Station 1 (Inlet of first rotor), Rotor-1
U	Peripheral speed	2	Station 2 (Exit of first rotor), Rotor-2
V	Relative Velocity of flow	3	Station 3 (Inlet of Second rotor)
Delta P_0	Total pressure rise across both rotors (P_{0x}- P_{01})	4	Station 4 (Exit of Second rotor)
R	Rotor	0x	Span wise location
Cp1,2	(Pressure- mass flow average (Pressure)@ R1Inlet)/ (mass flow average (Total Pressure in stationary Ref. frame)@ R1Inlet)-mass flow average (Pressure)@ R1Inlet		

Reference

[1] Sébastien DRON, Snecma SAFRAN Group, 2008, “Toward ACARE 2020: innovative engine architectures to achieve the environmental goals?” 26th International Congress of the Aeronautical Sciences.
[2] Petre Scimming, 2003, "Counterrotating fans-An aircraft propulsion for the future", Journal of Thermal Science, Vol-12, No-2, pp 97-103.
[3] B Roy,K Ravibabu, P Shrinivasa Rao, S Babu, A Raju and P N Murthy,1992, " Flow studies in ducted twin-rotor Contra rotating axial flow fans.", International Gas Turbine and Aero engine congress and exposition, Germany, June 1-4,92-GT-390.
[4] D S Pundir & P B Sharma, 1992 "A study of aerodynamic performance of a contra-rotating axial compressor stage", Defense Science Journal, Vol-42, no-3, July 1992, pp 191-199.
[5] Y-Y Chen,B Liu, Y Xuan and X-R Xiang ,2008, "A Study of speed ratio affecting the performance of a contra-rotating axial compressor", Proc. I MechE, Vol-222, Part G: Journal Aerospace Engineering.
[6] S Kumar, Bhaskar Roy, 2000, "End wall flow development across a contra-rotating fan unit", International Gas Turbine and Aeroengine congress and exposition, Germany, May 8-11, 2000, 2000-GT-502.
[7] Jian Xu, Chunqig Tan, Haisheng Chen,Yagnli Zhu and Dongyang Zhang, " Influence of tip clearance on performance of a contra rotating fan", Journal of Thermal Science,Vol-18,No-3,pp 207-214.
[8] Robert Meyer, Karsten Knobloch, Johannes Linden, 2010 " Hot wire measurements in a high speed counter rotating turbofan rig", Proc. Of ASME Turbo Expo 2010: Power for Land, Sea and Air, June 14-18, 2010, Glasgow, UK
[9] J Kerrebrok, Elstein, A Merchant, 2008, "Design and test of an aspirated counter-rotating fan", Journal of Turbomachinery, April 2008, Vol-130, pp 021004-(1-8).
[10] Lee-Sang CHO, Bong-Jun CHA, Jin-Soo-Cho,2009, " Experimental study on the three dimensional unsteady flow characteristics of the Counter-Rotating Axial Flow fan", Journal of Fluid Science and Technology,Vol-4,No-1,2009,200-209.
[11] CFX solver theory guide.