

Do fan blades require cyclic pitch when in push configuration inside telecommunication chassis ?

Farzam Roknaldin
Applied Thermal Technologies
3255 Kifer Road
Santa Clara, CA 95051 USA
E-mail: roknaldi@applied.fluent.com

ABSTRACT

It has been realized that fan tray does not function in push configuration in telecommunication chassis when inlet height is smaller than certain value. Originally this was associated with excessive pressure drop due to flow turn inside inlet plenum or basically bottle neck pressure drop through inlet opening itself, however, the problem became worst when higher pressure capacity fans were used. In latter, total CFM increased, as expected, but slot to slot as well as front to back airflow distribution became very uneven, leaving few slots starving. CFD simulations were used to understand this flow behavior, however, they were not capable of duplicating the measured airflow distribution. In this work, this problem is presented and analyzed by means of aerodynamic theory of fan blades. It is shown that high speed flow through small inlet do not turn completely, but rather, enter the fan tray at shallow angle. This causes one side of fans to be ineffective leaving slots over those portions starving. Even the effective sides of fans produce directional jets leaving rear portion of those boards directly above them starving. By bringing knowledge from helicopter aerodynamics to fan blade design, the cause for this mysterious behavior was discovered. Consequently, the concept of fan blade *cyclic pitch* [3] is introduced as possible solution to this problem. Next generation fans might be designed to have variable blade angle within each revolution. Other implication of this work is to lay out building blocks for better CFD model for fan that include some blade aerodynamic effect. This will be the subject for future work.

INTRODUCTION

There are two common ways of placing fan tray inside telecommunication chassis including routers and switches: Pull system is referred to the configuration where the fan tray is located above the card cage holding the boards. Push system is referred to the configuration where the fan tray is below the card cage (pushing the air through). In terms of system level design, both configurations seem to be well understood. In both cases flow has undergo 90 degree turn from the inlet into the card cage and another 90 degree turn toward the exhaust. These turns causes most of the pressure drop excluding pressure drop through filter that can be significant if dirty. For push systems, short inlet height is usually desirable due to overall height limit on chassis. Common thermal management knowledge relates reduction in inlet height to the bulk pressure drop inside the chassis and suggests utilizing high-pressure capacity fans, available in the market, to overcome this additional pressure drop. Computational Fluid Dynamic (CFD) models are also support this line of thinking in that CFD tools predict accurate CFM, system pressure drop as well as reasonable velocity distributions. However, what they fail to predict are local jets induced directly from fan blades. In most CFD codes, used in thermal management industry, fan and its blades are modeled as a plane object with specified fan curve [1,2]. More advance CFD packages provide options of modeling the fan hub as well as induced flow swirl over the fan plane. But, as experiment shows, this is not always enough to capture right flow behavior. Small inlet problem is among those exceptions where flow enters the fan tray in a shallow angle that might cause flow stall. Manufacturing fan curves, used in the modeling, are obtained from wind tunnel testing where the flow approaches the fan normal to the fan plane. Large variation from this assumption results either degradation from fan curves or complete stall over the fan blades.

DESCRIPTION OF THE PROBLEM

In order to describe the problem associated with fan tray, air velocity data from two different tests conducted at Applied Thermal Technologies Laboratory are presented. These tests belong to two different chassis. One has relatively large inlet and inlet plenum beneath the card cage. Other suffers from small inlet but fans are chosen to have higher pressure capacity.

Test 1:

In this test opening is large such that fan blades are in their ideal suction condition. By that one means flow turn 90 degrees before enter fan blades.

Figure 1 shows schematic front view picture of this system. As can be seen fan tray is below the card cage. Fan blade lines are shown below the card cage and fan hubs are marked with black rectangles. Figure 2 shows the slot-to-slot velocity distribution measured by velocity probes located at center of blank boards (one probe per board) that fill all fourteen slots. Vertical positions of probes are indicated as “probe line” in figure 1. For quick analysis check, fan hubs are shown at the bottom of figure 2 as well.

As can be seen from measure symbols, there is relatively velocity shift moving from the fan hub toward blade tips. This is expected since blade tips with higher velocities provide jet flows that carry most of the push, whereas, there is no push over fan hubs. In addition to this measurement, probe line was moved from front (faceplate side) to back (connector side) to spot front to back variations or shifts in flow pattern. However no major change was observed, therefore, plotted data are good representation of flow pattern within slot.

For comparison purpose, simulated results from Icepak/Fluent CFD code is plotted in the same figure as a solid line. Fan hubs as well as flow swirl per each fan were added in computational model. However, one can see that only smooth (average like values) are captured and actual variations are mainly missed or somehow damped in the CFD simulation. At the first glance, it was suspected that inability to resolve variations is associated with employing combination of first order numerical scheme and algebraic turbulence model. However, when second order numerical scheme was used plus turbulence model was turned off, solution did not improve. This pointed toward inadequate fan model [1] rather than overall numerical scheme.

Although finding ways to improve the fan model is an interesting topic by itself, it might be presented in another article. The main focus of this works is to understand true fan or fan tray behavior inside chassis. Finding here can be used either to design suitable fan blades or to enhance fan models within simulation

packages. Saying that, let’s discuss findings in “Test 2” which is the main focus of this paper.

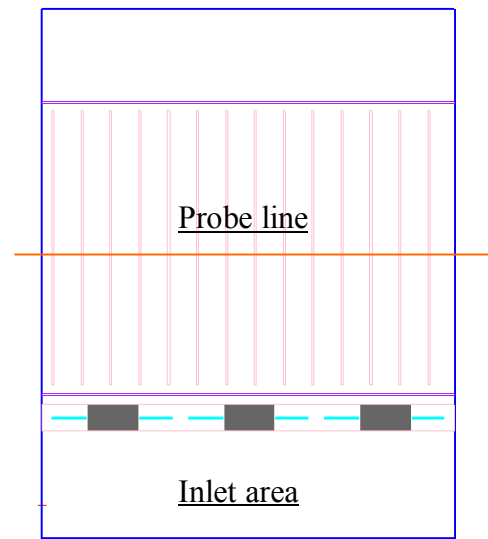


Figure 1. Schematic front view of the push system used in test 1

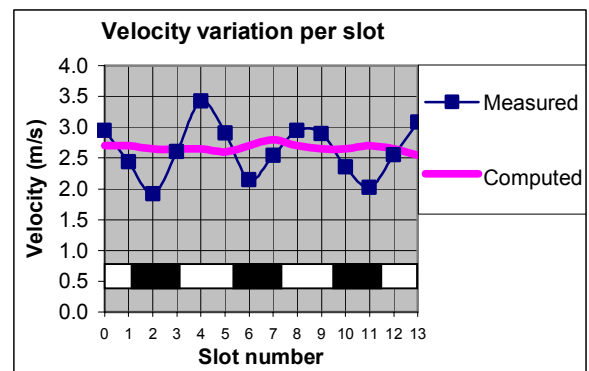


Figure 2. Slot to slot velocity distribution at the center of blank cards in test 1

Test 2:

Schematic picture of the system in this test is shown in figure 3. Again the fan blades and fan hubs are also marked. Inlet opening is in the front bottom and exhaust opening is the rear top. Here the inlet opening is relatively shorter than the one in other chassis measured in test 1. Such a short opening, despite of

causing huge pressure drop is widely seen in existing designs. As mentioned before, common knowledge revolves around balancing the pressure drop and leads to a design that uses more powerful fans run at high rpm. Figure 4 shows the right side of the actual chassis used in this test. Fan tray consists of two arrays of five fans each. Second array is behind the first array in Figure 4. What is shown in this picture is right portion of each array. Relative locations of fans with respect to slots are clearly seen. Chassis is filled with blank boards. Each board has three velocity probes that attached to it at the vertical center of the board constituting an array to measure front position near faceplates, middle position and rear position near connectors. Reason for choosing three probes per board is because of measured variations in depth as well and will be clear later when analyzing the data. Figure 5 shows the velocity distribution regarding this case. Simulation results were not shown since existing CFD models are incapable of duplicating this phenomenon. For simplicity only the results for slots 11-17 are shown. Four fans under those slots are marked from 1 to 4 as it is shown in figure 4. Two tests are conducted at two different times with different probes and sealed faceplates to ensure reliability of the data. Test results were almost identical. Only one set is shown here.

Focusing on probes for the “front” portion in figure 5 (diamond shape data) and shifting from right to left above fan 2 and fan 1, one observes high velocity jets above right blade tips of fan 2 (slots 17 and 16), low velocity above hub of fan 2, (slot 15), but also very low velocity above left blade tips of fan 2 (slot 14). How can it be? Recovery starts at slot 13 with the help of jet flow above right blade tips of fan 1. Moving further to the left, same trend continues.

Here airflow toward front portion of slot 14 is lost or, in another terms, jet flow over left blade tips of fan 2 is lost, or in a simpler term, only 1/3 of fan 2 is effective. More disastrous than this is the airflows through “middle” and “back” portion of the chassis are dead for slots 16-17 and pick up gradually and reaches maximum at slot 13. What happened to the CFM produced by fan 4? One might think fan 2 stealing most of the airflow entering the system and leave fan 4 choked, but as will be seen later, this is not the case and strengths of these fans are measured to be identical.

From thermal management point of view this design is troublesome no matter how powerful the fans are. System that is shown in test 1 (figure 2) suffered from expected velocity variation but the root mean square (rms) values of the variation is within the design range. This is certainly not true for system with smaller opening shown in test 2.

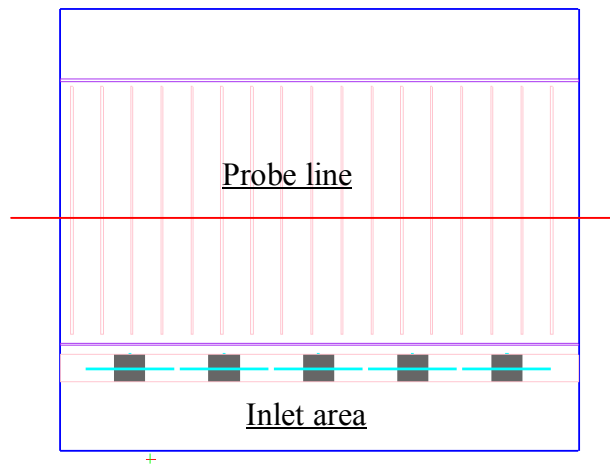


Figure 3. Schematic front view of the push system used in test 2

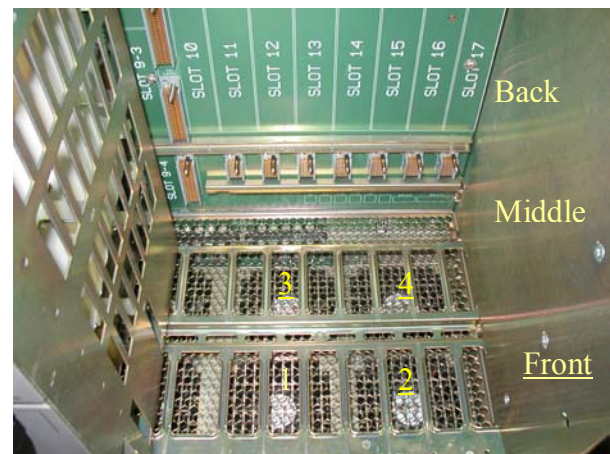


Figure 4. Relative position of fans with respect to slots in the actual chassis used in test 2

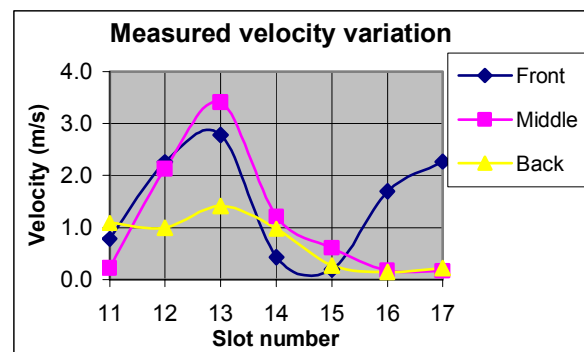


Figure 5. Slot to slot velocity distribution at the center of blank cards in test 2

Explanation through aerodynamics of fan blade

Highlights of above problem sparked recognizing the similar problem faced in aerospace industry regarding designing of helicopter blades. Design of helicopter blade is more complicated than regular propeller for fixed wing plane in that the blades are rotating in a plane tangent to the helicopter's forward motion [3]. This is quite different from propellers that blades are rotating in a plane normal to planes motion. Fan designs resemble propeller designs. All manufacturing fan curves and blade designs are based on a single factor that flow enters the fan at a normal angle. But what makes helicopter blade effective to provide a balanced lift force in forward motion, where main-stream flow in tangent to blade rotating blade, is what is so called *cyclic pitch*. This means change of angle of attack of the blade within each revolution. Consider left and right blades in a top view of helicopter's blade plane shown in figure 6. Approach velocity per blade is the sum of free stream velocity and blade's rotational velocity ($V=wr$) where w is the angular velocity and r is the radius from the center. One can easily see that approach velocity is different for left and the right blade. Right blade faces larger approach velocity therefore produces more lift, whereas, left blade faces smaller approach velocity and produces less lift. By maintaining this configuration, helicopter will tip over from right to left in a forward motion. To avoid this situation, cyclic pitch is introduced which reduces the angle of attack of any of blades enter the right side. Hence the blade in the right side, although faces higher approach velocity, is less angled, therefore produces the same amount of lift force as the one in the left side. It can also be done the other way around by increasing angle of attack for the left side. Mechanism to implement cyclic pitch is simple by introducing a lever bar in one side at the joint between the blade and the center rotor.

This background can be used to understand flow variation problem that is faced in small inlet push system. Because of small size inlet, air speeds up tremendously (in excess of 10 m/s) when passing through inlet in a direction tangent to the fan plane. Due to lack of plenum, this high-speed flow does not turn normal to the fan tray, rather approaches the fan blades in a shallow angle similar to the flow over helicopters blades. Since fan blades does not support these angles and don't have adjusted cyclic pitch, they stall completely [4]. Figure 7 shows blade orientation for a typical fan pushing the flow normal up. Note that fan is rotating in clock-wise direction. Stall happens in two different ways in following manners:

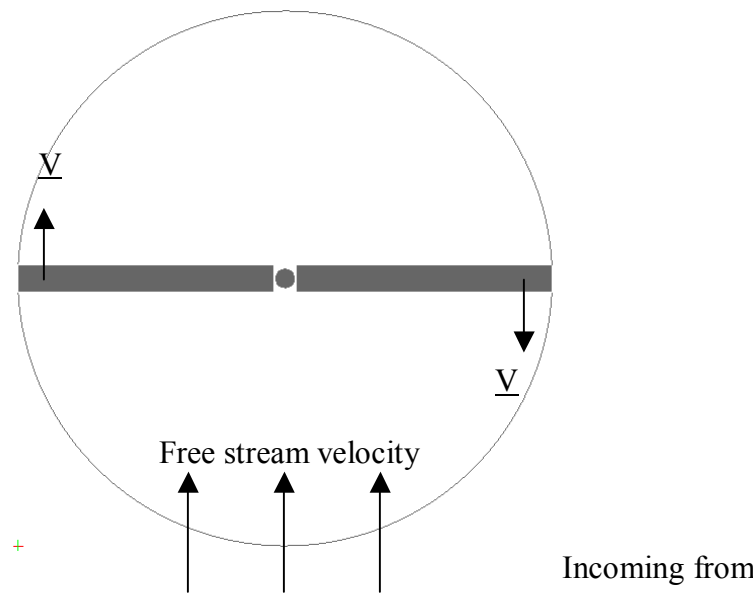


Figure 6. Schematic front view of the push system used in test 2

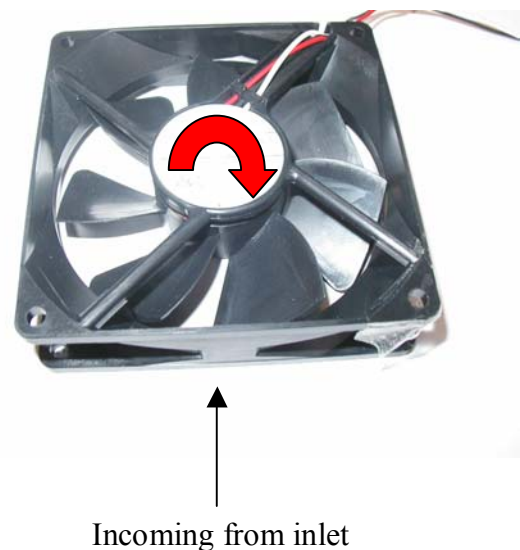


Figure 7. Push fan in clock-wise rotation

Right side of fan: The incoming air from inlet and the blade itself collide head to head. Fan blade here does not act as aerodynamic body, but rather, scoops the air normal to its surface. This is shown in figure 8.

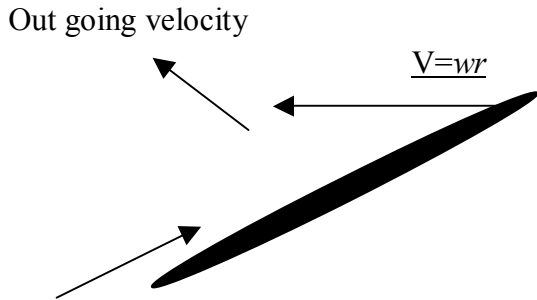


Figure 8. Velocity triangles for a blade in the right side of fan shown in figure 7

Left side of fan: This side is basically ineffective. Airflow from inlet hits back of the blade. Blade basically blocks the incoming airflow from inlet to pass through. It only induces some flow swirl due to blades angular velocity.

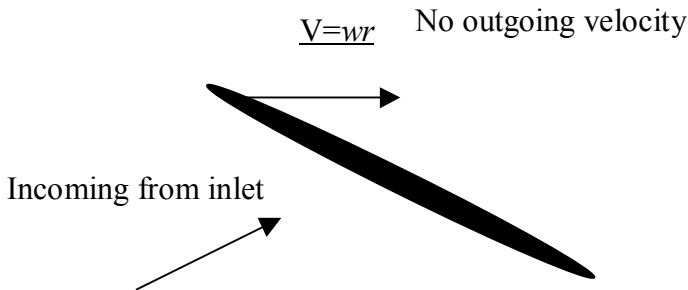


Figure 9. Velocity triangles for a blade in the left side of fan shown in figure 7

With above schematic in mind lets refer back to the chassis in used in test 2 and try to explain the data

presented in figure 5. Figure 10 illustrate the side view of the chassis. Incoming flow from the inlet enters the fan tray at shallow angle. Fan numbers 2 and 4 in figure 4 are marked here. As mentioned, 2/3 of each fan is ineffective: (right 1/3 as explained in figure 9 and middle 1/3 that is the hub portion). Remaining 1/3 (left side) is effective but produces angled jets as marked by arrows in figure 10.

Notice that incoming airflow is redirected by pushing it forward toward faceplates. This leaves the rear portion of boards in void as agrees with velocity data seen in figure 5. The middle portion of board probe should also receive the jet flow from the rear fan 4. This explains the high -speed data point seen in slot # 13 for the middle probe. However this is not seen for the middle probe in slot # 17. There might be an explanation for that by looking at figure 11.

Figure 11 shows the front view of the chassis. As seen in the laboratory, jet flows from effective portion of fan, also tend to lean toward right. This behavior is observed when boards were removed. Presence of board, however, might slice the flow in this direction and make them straight parallel to boards. Again large void regions are seen. Here the jets from fan 2 and 4 hit the right wall of the chassis. This wall might re-direct the jet flow from fan 4 to the front leaving middle portion, in addition to rear portion, of slot # 17 as void.

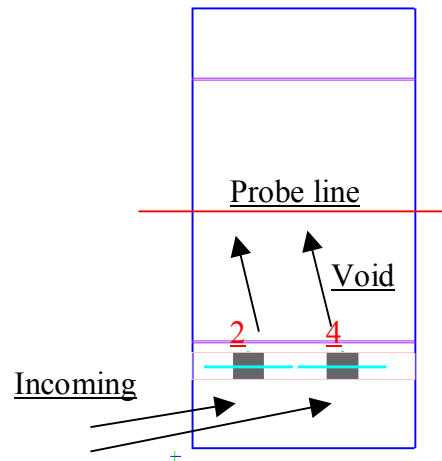


Figure 10. Directions of velocity vectors in chassis side view

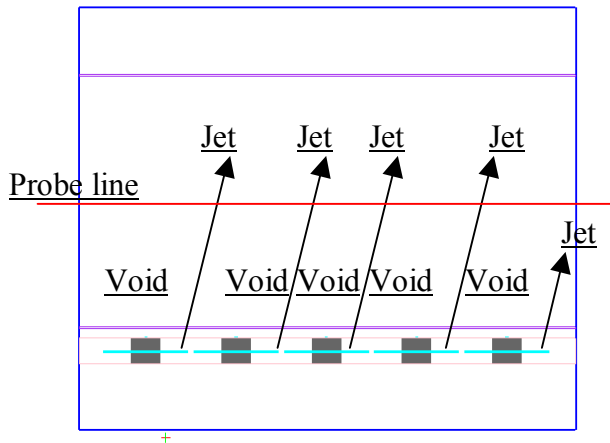


Figure 11. Directions of velocity vectors in chassis front

There are several fix to this airflow stall problem. The most obvious one is to enlarge inlet area and provide enough inlet plenum for complete flow turn. However with a trend to use smaller inlets in the future designs below are proposed:

- 1) Design new fans with cyclic pitch. Blade pitch can be adjusted prior to usage depend on chassis characteristics.
- 2) Include guide vanes before fan blades to redirect the airflow normal to fan plane. Many high performance fans have guide vanes after fan blades. This does not solve the problem.

A quick fix may also be possible by adding a layer of honey-comb structure below the fan tray to act as guide vane.

One area that did not discussed in this work is acoustic noise increase due to mentioned problem. As tested for a single fan, acoustic noise is higher when fan blades are at stall, therefore any effort to maximize the aerodynamic efficiency will reduce the acoustic noise.

CONCLUSION

In this work a common problem was introduced regarding un-symmetric airflow distribution that appears in telecommunication chassis when the fan tray is below the card cage in push configuration. This un-symmetric airflow distribution that is only observed during flow testing occurs when inlet is small and leave many boards starving of air no matter how powerful the fans are underneath. This problem remained as mystery for some time since it was beyond the common knowledge and CFD modeling was incapable of capturing it. Roots of this problem is explained and analyzed through aerodynamic theory of fan blades. It is shown that small inlet will cause flow stall over the whole fan area. This is an unusual stall such that it makes 2/3 of the fan area ineffective and the remaining 1/3 of the fan produces directional jets that leave rear portion of the board (near back plane) starving. New fan designs were proposed that incorporate cyclic-pitch (as for the helicopter blades) to adjust fan blade angles in accordance with approaching flow direction coming from the small inlet.

ACKNOWLEDGEMENTS

Author would like to thank Applied Thermal Technology for the support and the access to airflow test laboratory.

REFERENCES

- [1] F. Roknaldin, R.A. Sahan and X.H. Sun "A simplified CFD model for radial blower" Itherm conference 2002
- [2] Icepak user's guide, Fluent Inc.
- [3] J. Gordon Leishman , Michael J. Rycroft , Wei Shyy " Principle of Helicopter Aerodynamoics"
- [4] T. Cebeci , F. Roknaldin,. and L.W. Carr, "Predication of Stall and Post-Stall behavior of airfoils at Low and High Reynolds Numbers" AIAA Paper No. 93-3502, January 1993.