Numerical and Experimental Investigation of a Serpentine Inlet Constructed with a Polynomial

Changjie Ge¹, Bin Chen², Yinhui Shang³ and Rui Han⁴
¹,²,³,⁴Chengdu Aircraft Industrial (Group) Co., Ltd, Chengdu, China, 610031

Abstract. The investigation of the serpentine inlet is particularly important due to the high stealth requirements of the next generation unmanned aerial vehicle (UAV). The paper presents a design method of generating a serpentine inlet controlled by a center line which is a polynomial. The serpentine inlet is designed considering the positions of the airborne equipment, the stealth and aerodynamic performances. The study includes numerical simulations based on the Reynolds-averaged Navier-Stokes simulation approach and experiments in wind tunnel. The aerodynamic performances of the serpentine inlet at different inflow Mach numbers, mass flow ratios and angles of attack are obtained. Moreover, the flow structures are studied at several critical conditions and the performance and distortion of the designed serpentine inlet are acceptable without any flow control.

Keywords: Serpentine inlet; UAV; Circumferential distortion index; Numerical and experimental investigation.

1. Introduction

UAV has received extensive attention in recent 20 years due to its advantages of low cost, no casualties, high survivability and convenient tests[1]. For the UAV aircraft, the engines are usually installed in the back and into the fuselage. The flying wing UAV with top-mounted inlet is widely used because of its higher lift-drag ratio and better stealth[2]. The inlet is one of the main sources of the scattering of radar wave, which mainly comes from the inner surface of multiple reflection and diffraction. Besides, compressor blade is also a source of strong reflection, and therefore the stealth design of the inlet is particularly important. There are currently several design methods for the inlet considering the stealth. One of them is the submerged inlet and it’s fully imported into the fuselage, which not only can effectively reduce the radar cross section (RCS), but also greatly cut down the structural weight and windward area[3]. The second one is the inlet with grille intake which can reduce the effect of RCS through the upward reflection of reflected radar waves and the conductivity of the grille can also improve the stability of the engine[4]. Last but not least, the serpentine inlet is used to completely shield the compressor blade so that the incoming wave can be reflected repeatedly in the inlet to reduce the reflection energy. The parameterized design of S-shaped inlet include two aspects: one is the parameterized design of center line and area change law, the other is the parameterized design of smooth transition of inlet profile. Based on years of experience in inlet design, McDonald Corporation of America summarizes three kinds of regularity curves of center line and area of S-shaped inlet in engineering design, and gives three dimensional parameterized equations[5]. In recent years, many S-shaped inlet design and optimization method have been developed, such as B-spline curve, polynomial curve, NURBS equations gradient-based optimization techniques and so on[6]. Polynomial curve can be used to parameterize the design of S-shaped inlet, which is easy to adjust and optimized, so it is a popular direction of inlet design. As for numerical calculation and aerodynamic analysis, different turbulence models are used for numerical calculation and comparison with experiment. The results
show that k-w SST model has the highest simulation accuracy for separation\textsuperscript{[3]}. The serpentine inlet generally consists of two conventional S-shaped inlet. The pressure gradient of the flow in the inlet duct is ordinarily large due to the increase of the wall curvature, and the strength of the secondary flow increases according to the enhanced inlet duct bending\textsuperscript{[4]}. What’s more, larger flow separation will occur, and it will lead to the deterioration of the aerodynamic performance of the inlet, which will not only reduce the total pressure recovery, but also increase the distortion of the serpentine inlet\textsuperscript{[5]}. Without any flow control, the worsening flow can easily affect the stability of the engine after entering the compressor and the situation is not allowed. Previous researches indicate that there are three main methods available to solve the serpentine inlet/engine compatible issue. The first technique is the passive control technology which is typically represented by mechanical vortex generators. The mechanical vortex generators produce vortex flow in the opposite rotating direction to counteract the naturally vortices generated in the inlet duct, which will reduce the aggregation of the low energy flow and inhibit the flow separation. Many scholars have conducted in-depth research on passive flow control by optimizing the layout of passive flow control and the flow field at the inlet port is effectively controlled\textsuperscript{[6]}. The second method is the active control technology which mainly includes flow injection and bleeding. The basic idea of flow injection is to power the boundary layer before it separates, while flow bleeding is to remove the lowest energy part in the boundary layer which maybe detached from the surface of the wall later\textsuperscript{[7]}. The last method is the inlet design control technology, which utilizes the surface shape transition to inhibit the flow separations and does not require additional control technology.

In the paper, generating a serpentine inlet controlled by a center line is discussed. The center line is a polynomial and the graph of the polynomial is studied. Considering the positions of the airborne equipment, the stealth and aerodynamic performances, the final serpentine inlet is given. The aerodynamic performances of the serpentine inlet at different free stream Mach numbers, mass flow ratios, and angles of attack are obtained. Moreover, the flow structures are studied at several critical conditions and the performance and distortion of the designed serpentine inlet are acceptable without any flow control.

2. Serpentine Inlet Design
The serpentine inlet is controlled by a center line and the formula of the center line is a polynomial given by equation (1)

\[ y / \Delta y = a(x / \Delta x)^2 + b(x / \Delta x)^3 + c(x / \Delta x)^4 + d(x / \Delta x)^5. \]  

Dimensionless formula is discussed and given as equation (2). The coefficient \( a, b, c \) and \( d \) in the equation have the restricted condition as shown in the equations (3). It can be seen that the graph of the equation (2) goes through the point \((x_0, y_0)\), and if point \((x_0, y_0)\) is given, there will be 3 valid equations with 4 unknowns in the equations (3). As a result, the equations will have one degree of freedom to adjust the shape of the graph.

\[ y = ax^2 + bx^3 + cx^4 + dx^5 \]  
\[ y\big|_{x=1} = a + b + c + d = 1 \]  
\[ \frac{dy}{dx}\big|_{x=0} = 2a + 3b + 4c + 5d = 0 \]  
\[ y\big|_{x=x_0} = y_0 \rightarrow ax_0^2 + bx_0^3 + cx_0^4 + dx_0^5 = y_0 \]  

In this paper, the value of \( a \) and the coordinates of the point \((x_0, y_0)\) are discussed. To take one example, the value of \( a \) is given to be 0.9, and the coordinates of the point \((x_0, y_0)\) are set as \((0.968, 0.98)\) to improve the stealth performance and loading capacity of the UAV. As shown in figure 1, the blue curve is the graph of the equation \((a=0.9, (x_0, y_0) = (0.968, 0.98))\), the red curve is obtained by the symmetry point \((0.5, 0.5)\) of the blue curve and is used to be the serpentine inlet dimensionless center line. The offset of the serpentine inlet duct is -0.62D and the length is 7D, the diameter of the...
outlet is fixed at D=750mm (Black curve in Fig 1). Four different centerlines which will be discussed and the graphs are shown in figure 2.

Area distribution is governed by the equation

\[ \frac{A}{A_1} = \left( \frac{A_1}{A_1} - 1 \right) [3(x/\Delta x)^2 - 2(x/\Delta x)^3] + 1 \]  

(4)

Figure 3 shows the fore-body of a flying-wing UAV and the geometry of the serpentine inlet. Considering the overall layout and the structure of the engine, the exit equivalent section is 200mm. The CFD extension is set to be twice the diameter of the outlet.
From the results in Table 1, the inlet ducts constructed by the center line c or center line d will interfere with the airborne equipment while the inlet ducts constructed by center line a or b will not interfere. Moreover, considering the total pressure recovery and distortion index DC60, the center line a is finally chosen.

### 3. Methodology

#### 3.1. Numerical Approach

Fluent 17 software is used for the simulation of the flow. The flow is solved by using the Reynolds-averaged Navier-Stokes equations. The k-ω SST turbulence model is used to simulate the turbulence effects. The governing equations are solved by a second order upwind finite-volume scheme. No-slip adiabatic wall boundary conditions are imposed on the solid walls. The backflow turbulent kinetic energy is assumed to be 1m²/s², backflow specific dissipation rate 1/s and the wall roughness height 0. The pressure is given at the section of the CFD extension.

Multi-block structure mesh is generated by ICEM CFD 17 software. Figure 4 shows the local surface mesh and the flow field of the computational model. The cell number of the half model is about 7.5 million and 15 million for the whole model. The mesh size along the walls is specified such that the y+ number is less than 3.0, and it results the first cell minimum height as 0.02mm. The growth rate of the cell height in the direction to the near wall is 1.2.

![Figure.4. Multi-block structure mesh for the serpentine inlet.](image)

#### 3.2. Experimental Setup

Model tests are conducted to obtain the flow characteristics in the high-speed wind tunnel. The test section of the wind tunnel is 1.2 meters in height and 1.2 meters in width, and the available test Mach number ranges from 0.4 to 2.021. The experimental model scale is 9.04, and it’s set by the appropriate test rake with a diameter of 82.9mm at AIP. The model consists of the fuselage, the serpentine inlet, and the pressure measurement system (Fig.5). The pressure measurement system contains 40 total pressure probes, 6 dynamic pressure transducers and 8 static wall pressure taps at AIP (Fig.6). During the experimental tests, the free stream Mach number is varied from 0.4 to 0.8. The blockage of the model to the test section of the wind tunnel is about 3.4%.
4. Results and Discussion

In this section, the aerodynamic performance of the serpentine inlet is discussed through the CFD method and the wind tunnel tests. Static pressure distributions along the top and bottom surfaces are studied. The effects of free stream Mach number $Ma_0$ and mass flow ratio on the performance of the serpentine inlet are presented. The design point of the serpentine inlet is: cruising height $H=12\text{km}$, free stream Mach number $Ma_0=0.75$, angle of attack $\alpha=0^\circ$, side-slip angle $\beta=0^\circ$ and Mach number at the AIP $Ma_{AIP}=0.46$.

4.1. Static Pressure Along the Top and Bottom Surfaces

The static pressure distributions along the wall of the serpentine inlet at the design point is presented in figure 7. The location of the throat position is set to be $x=0$, and the AIP $x=1$. As shown in the picture, before the air flow enters the serpentine inlet, the static pressure of the bottom surface decreases rapidly from $x=-0.43$ to $x=-0.2$ because of the local flow acceleration. Subsequently, the mass flow ratio $\phi$ of the inlet at the design point is about $0.76<1$, as a result, the flow decelerates and the static pressure rises quickly from $x=-0.2$ to $x=-0.05$. The static pressure drops sharply near the inlet lip on account of the local Mach number dramatic changes. When the flow passes through the throat and enters the serpentine inlet, the static pressure distributions of the top and bottom surfaces show two typical $X$-form distributions. Furthermore, there is obviously a bulges near the AIP due to the existence of the equivalent section. Figure 8 shows the total pressure contour of the symmetric plane at the design point, which indicates that there is no significant flow separation in the serpentine inlet.
Figure 7. The static pressure distributions along the wall of the serpentine inlet (Ma0=0.75, α=0°, β=0°).

Figure 8. The total pressure contour of the symmetric plane (Ma0=0.75, α=0°, β=0°).

4.2 Effects of Free Stream Mach Number Ma0
Figure 9 and figure 10 show the effect of free stream Mach number Ma0 on the total pressure recovery σ and distortion index DC60 at the AIP. It can be seen that the total pressure recovery σ and distortion index DC60 vary little with the increase of free stream Mach number.

Figure 9. Total pressure recovery versus free stream Mach number Ma0.
5. Effects of Mass Flow Ratio

Figure 11 presents the effect of mass flow ratio $\varphi$ on the total pressure recovery $\sigma$ at the AIP. Figure 12 shows the distortion index DC60 with mass flow ratio $\varphi$. It is shown that the total pressure recovery $\sigma$ with mass flow ratio $\varphi$ from the CFD result is in a good agreement with the experiment result from the wind tunnel, while the distortion index DC60 with mass flow ratio $\varphi$ has some differences but not much. Furthermore, when $\varphi<0.58$, the total pressure recovery $\sigma$ increases with mass flow ratio $\varphi$, but decreases from $\varphi=0.58$ to $\varphi=0.85$. In the experimentally studied range of AIP Mach number from 0.253 to 0.555, the value of DC60 varied from $-0.30$ to $-0.13$, which meets the general requirement of the aircraft engine.

6. Effects of Angle of Attack

Figure 13 and figure 14 represent the total pressure recovery contours of the CFD and experimental results at the AIP for different angles of attack, respectively. It can be seen that there exists a little
difference between the total pressure recovery contours of the CFD results and the experimental results, but the locations of the low pressure regions are quite similar. Moreover, both of the total pressure recovery distributions of the CFD results and the experimental results are almost the same from $\alpha=-2^\circ$ to $\alpha=5^\circ$, however it’s different while $\alpha=10^\circ$.

![Figure 13](image1.png)

**Figure 13.** Total pressure recovery contours at the AIP for different angles of attack (CFD, $Ma_0=0.75$, $\beta=0^\circ$, MAIP=0.46).

![Figure 14](image2.png)

**Fig.14.** Total pressure recovery contours at the AIP for different angles of attack (Experiment, $Ma_0=0.75$, $\beta=0^\circ$, MAIP=0.46).

# 7. Flow Structures

In this section, numerical approach is used to investigate the flow structures of the serpentine inlet. As there are adverse pressure gradient and aggressive surface changing in the serpentine inlet, the flow of the low pressure region in the serpentine inlet is mainly dominated by vortexes, especially at the non-design points. In order to find out the physical reason of the low pressure region at the AIP and further investigate the vortex structure, the inlet is cut to several diagnostic cross sections along the center line and the streamlines of the low total pressure recovery regions and the secondary flows at the AIP are studied. Figure 15 shows the total pressure recovery contours of the cross sections, the streamlines of the low pressure regions and secondary flow at AIP with different angles of attack at the design point. Figure 15 (a) presents the result at $\alpha=0^\circ$, which reveals that there are three pairs of low total pressure recovery regions at the AIP caused by two pairs of counter-rotating vortices. The strong pair of vortexes is originated from the corner of the inlet due to the existence of the aggressive surface changing of cross sections which transform the triangle to the circle, and develops downward to the exit of the inlet under the effects of self-rotating, adverse pressure gradient and cross pressure gradient. Their final locations are the low pressure regions at the AIP. Moreover, the top and bottom low pressure regions are mainly caused by the growth of the boundary layer and the swirling action of S bending. As the adverse pressure gradient and cross pressure gradient rise, the secondary flow and the vortex will be occurred. Furthermore, it is possible that if the vortexes are large enough, they will interact with each other. Figure 15 (b) shows the result at $\alpha=5^\circ$, which indicates that the pressure distributions are similar to the condition at $\alpha=0^\circ$ and the corner vortexes are lifted higher than the position at $\alpha=0^\circ$ due to the raised cross pressure gradient. As shown in Figure 15 (c), while $\alpha=10^\circ$, the cross pressure gradient becomes larger on account of the bigger local turning angle of the inlet. Under the circumstances, the corner vortexes move up further, until they are coupled with the top vortex and form a larger vortex. Meanwhile, thick boundary layer along the fuselage is generated because of large angle of attack and the more low energy flow enters the inlet. However, the low energy regions at the
top surface of the AIP are much bigger due to the severe pressure gradient and the high energy flow is pushed.

![Secondary flow at AIP](image1)

**Figure 15.** Total pressure recovery contours, the streamlines of the low pressure regions and secondary flow at AIP (Ma0=0.75, β=0°, MAIP=0.46).

![Secondary flow at AIP](image2)

**Figure 16.** Total pressure recovery contours and the streamlines of the low pressure regions (Ma0=0.75, α=15°, β=6°, MAIP=0.46). It can be seen that under large angle of attack and side-slip angle, large pressure gradient is generated and so the flow separation occurs in the inlet duct. But in this case, the synthesis distortion index W=5.2% (experiment data) is still acceptable according to the requirements of the aircraft engine.
8. Conclusion
In this paper, the investigation of the serpentine inlet constructed with a polynomial is given. The total pressure recovery and distortion index DC60 change little with free stream Mach number Ma0=0.4~0.75. Moreover, the total pressure recovery is greater than 0.97 and the value of distortion index DC60 is less than 0.2 with Ma0=0.75. At the same time, the flow structure dominated by vortexes is analyzed and the synthesis distortion index W=5.2% at the harsh condition α=15°, β=6°. The serpentine inlet has excellent performance and is acceptable without any active or passive flow control techniques.

References
[1] AUSTIN R. Unmanned aircraft systems [M]. New York: John Wiley and Sons Incorporation, 2010.
[2] Torralm M. Design and analysis of compact UAV ducts [R]. AIAA 2006-2828, 2006.
[3] Knott E F. Radar cross section measurements [J]. Proceedings of the IEEE, 1993, 27(9):197.
[4] DAS N. C., KHASTGIR D, CHAKI T K, et al. Electromagnetic interference shielding effectiveness of carbon black and carbon fibre filled EVA and NR based composites [J]. Composites: Part A Applied Science and Manufacturing. 2000.31(10):1069-1081.
[5] Lee C. Subsonic diffuser design and performance for advanced fighter aircraft [J]. Aiaa Journal, 1985.
[6] Lee, B. J., Kim, C., and Rho, O.H., “Optimal Shape Design of the S-shaped Subsonic Intake Using NURBS,” 43th AIAA Aerospace Science Meeting and Exhibit, AIAA-2005-455, 2005.
[7] Lee B J, Kim C. Automated design methodology of turbulent internal flow using discrete adjoint formulation[J]. Aerospace Science and Technology, 2007, 11(2):163-173.
[8] Anthony M. Ferrar, Walter F. O’Brien, Progress in Boundary Layer Ingesting Embedded Engine Research[R], AIAA 2012-4283.
[9] Lee K, Lee B, Kang S, et al. Inlet distortion test with gas turbine engine in the altitude engine test facility [R]. AIAA-2010-4337, 2010.
[10] Chima, R. V, “Computation Modeling of Vortex Generators for Turbomachinery”. NASA/TM-2002-211551, NASA, April 2002. 43rd AIAA Aerospace Science Meeting and Exhibit, Reno, Nevada, Jan. 2005.
[11] D. L. Daggett, R. Kawai, D. Friedman. Blended Wing Body Systems Studies: Boundary Layer Ingesting Inlet with Active Flow Control. NASA CR 2003-212670, 2003.