Numerical study on adjustment of the main flow field with guide vanes in transition zone for an S-duct

Shuili Ren and Peiqing Liu

Abstract
A 30°/30° S-duct is often employed by a modern civil and military aircraft. Due to the vortex generators have poor control effect on the flow separation in the large curvature S-duct, so a new method adjusting the direction of the main flow in the transition zone for S-duct is proposed in the present study. These studies are carried out by solving Navier-Stokes equations with SST k-ω turbulence model. In order to explore the control mechanism of guide vanes on the flow field for S-duct, the internal flow field of S-duct with/without guide vanes is analyzed and compared in detail, including the velocity distribution on the symmetry plane, the secondary vortex intensity at different cross sections, turbulent kinetic energy on different cross sections, total pressure distribution on different cross sections, three-dimensional streamline, and so on. The results show that guide vanes can not only effectively adjust the direction of the main flow, but also restrain the flow separation, reduce the total pressure loss, and improve the uniformity of flow field.

Keywords
S-duct, guide vanes, the direction of the main flow, secondary flow

Date received: 24 August 2021; accepted: 23 December 2021

Handling Editor: Chenhui Liang

Introduction
The S-duct has several advantages: (1) its structure is compact; (2) can reduce the resistance of the propulsion system; (3) can reduce the sweeping area of radar and improve the stealth performance of the aircraft. In view of the above advantages, it has been widely applied in modern commercial aircraft and military aircraft, such as Boeing 727, General Dynamics F-16, and McDonal-Douglas F-18.

In order to decrease the velocity of airflow and increase the static pressure of airflow effectively, and provide a uniform airflow for an engine compressor, the geometric characteristics of an ideal S-duct are that the centerline has curvature and the cross section increases along the streamwise direction, which is a bent diffused duct. The curvature of the centerline leads to the formation of a pair of counter-rotating vortices, and increase of cross section produces the streamwise adverse pressure gradient, which leads to the flow separation. All these cause the decrease of total pressure recovery coefficient and the increase of distortion coefficient on the aerodynamic interface plane (AIP), which not only drop the performance of the engine, but also lead to diversified troubles.

Numerous researchers have studied the flow-field of S-duct. Wellborn et al. studied the subsonic compressible flow through a diffused S-duct. They observed the streamwise flow separation in the S-duct and the
counter-rotating vortices on the AIP, and provided standard verification data for computational fluid dynamics (CFD). Wendt and Reichert\(^8\) have conducted an experimental study on vortex ingestion in a diffused S-duct. They reported that the appropriate vortex ingestion can improve the quality of the flow-field on the AIP. Fiola and Agarwal\(^13\) have studied numerically the diffused S-duct, employing different turbulence models in their research. It is found that the SST-k-\(\omega\) turbulence model has a better ability to capture the flow-field. Gil-Prieto et al.\(^14\) have carried out a delayed detached-eddy simulation on the flow-field in S-duct. They demonstrated that DES simulation has good ability to capture the unsteady characteristics for the highly turbulent flow-field in the S-duct. All the above studies show that the flow separation and secondary flow in S-duct will cause the non-uniformity of the flow field and the total pressure loss, which will affect the performance of the engine.

In order to reduce the total pressure loss and make the flow field uniform for S-duct, many researchers use flow control methods. The commonly used passive control method is the vortex generator (VG). Researchers mainly controlled the local boundary layer separation near wall in S-duct by vortex generators in the past. The vortex generator can play a good role in controlling the local flow separation near wall region. Through the generated vortices by VGs, the high-energy fluid in the core zone is transported to the low-energy fluid in the boundary layer, so that the boundary layer separation can be prevented. The vortex generator also introduces secondary flow, which can prevent the formation of a pair of counter-rotating vortices.\(^15\) Vakili et al.\(^16\) arranged wing-type vortex generators upstream of the transition zone for a 30°/30° S-duct and successfully eliminated the flow separation. Sullerey et al.\(^17\) investigated the effect various vortex generator configurations in reducing the flow total pressure loss and distortion in S-duct (16°/16° and 24°/24°). Their results show that it is possible to use vortex generators to increase static pressure and improve flow field quality. Tanguy et al.\(^18\) carried out a vortex generator flow control study in an S-duct by using stereo particle image velocimetry. Their results show that the vortex generator is helpful to reduce the flow total pressure loss, but the control effect of the distortion of the flow field is poor. The results also show that vortex generators flow control method is effective in inhibiting the boundary layer separation near the wall for a small curvature S-duct, but it has little effect on inhibiting the flow separation for a large curvature S-duct.

Guide vanes are often employed to adjust the main flow direction. Tan et al.\(^19–21\) employed guide vanes to adjust the main flow direction in the flow field induced by force-vortex and established appropriate theoretical model to predict the flow field. For the diffused S-duct, due to the main flow is affected by the adverse pressure gradient and centrifugal inertial force, the direction change of the main flow along the centerline of S-duct is easy to cause the boundary layer separation near the outer wall of the exit zone and secondary vortex generated by the flow through bend. In order to better inhibit the flow separation near outer wall of the exit zone and the non-uniformity of velocity distribution in the main flow zone, a new method is proposed to improve the flow quality on the AIP by adjusting the main flow direction in the transition zone. Compared with the local flow control technology of vortex generator, the present paper attempts to set guide vanes in the transition zone of diffused S-duct to adjust the main flow direction so that the main flow can not only be close to the outer wall of the exit zone as much as possible to inhibit the local separation near the outer wall, but also adjust the non-uniformity of velocity distribution in the main flow zone to improve the flow field quality and uniformity on the AIP. The method of setting guide vanes in the transition zone of S-duct to improve flow field quality is rarely used in the past literatures.

In the present paper, by use of numerical simulation, the mechanism of adjustment of the direction of the main flow and improvement the main flow field through the S-duct are explored for a diffused S-duct with guide vanes in transition zone.

**Computational setup**

**Model in study**

As shown in Figure 1, the geometric model studied in the present paper is a 30°/30° S-duct. Delot et al.\(^22\) have studied the flow field in the duct experimentally. Reza et al.\(^23\) have also carried out a numerical study on internal flow field of the S-duct. The main geometric dimensions are given in Table 1. The center line of the S-duct is composed of two arcs, and the center angle of each arc is 30°. The diameter of the inlet of the S-duct \(D_{in}\) is 133.15 mm, the diameter of the outlet \(D_{out}\) is 164 mm, the curvature radius of the central line \(r_0\) is 665.4 mm, the arc length of the central line \(s_{arc}\) is 5.32\(D_{in}\), the length of the s-duct \(L\) is 5\(D_{in}\), the offset of the model \(H\) is 1.35\(D_{in}\) and the detailed information of the diameter of

![Figure 1. Schematic of 30°/30° S-duct geometry model.](image-url)
each cross section along the s coordinate can be given by

\[
\frac{D}{D_{in}} = 1 + 3 \left( \frac{D_{out}}{D_{in}} - 1 \right) \left( \frac{s}{s_m} \right)^2 - 2 \left( \frac{D_{out}}{D_{in}} - 1 \right) \left( \frac{s}{s_m} \right)^3
\]

in which, \(s_m\) is the length of the central line, \(s = 0\) at the center of the inlet for S-duct. In such a diffused S-duct, the flow undergoes a process of deceleration and pressurization.

**Numerical method and validation**

The double precision solver of ANSYS FLUENT 16.0 is used to perform a numerical simulation for the flow-field in the S-duct. In the simulation, the Reynolds-averaged Navier-Stokes (RANS) equations are solved with a finite volume method. The SST-k-\(\omega\) turbulence model is employed for the numerical simulation. The SIMPLE algorithm is used to deal with pressure-velocity coupling. The convection terms are discretized by second order upwind scheme, and the diffusion terms are discretized by second order central difference scheme.

The mesh for the simulation is generated in the ANSYS ICEM16.0 and the number of hexahedral cells is about 7.8 million for S-duct. The whole domain mesh is shown in Figure 2(a), the distance between the inlet of the whole domain and the inlet of S-duct is 6\(D_{in}\), the distance between the outlet of the whole domain and the outlet of S-duct is 4\(D_{in}\). The mesh for the outlet of the domain is shown in Figure 2(b). The number of nodes in S-duct cross section is about 13,000, and the number of nodes in the centerline of S-duct is about 600. The first layer grid height next to the wall is enough small to ensure the \(y^+\) smaller than 1 and the expansion ratio off the wall is 1.05, so the mesh has better quality and could capture the main flow field details.

Appropriate boundary conditions are very important for numerical simulation. The pressure inlet boundary condition was chosen for the inlet of computational domain and the pressure outlet boundary condition was chosen for the outlet of computational domain. According to the atmospheric environment when the S-shaped intake works at altitude of 1100 m, total pressure (\(P_{0,\text{inlet}} = 88,744\) Pa) and total temperature (\(T_0 = 281\) K) were set at the pressure inlet. The static pressure (\(P_{\text{outlet}} = 79,700\) Pa) was set at the pressure outlet to meet the outlet mass flow of 2.427 kg/s.

To verify the numerical method, the simulation results are compared with the results in the literatures, as shown in Figure 3. The surface pressure coefficient \(C_p\) for L-up and L-down are compared with literature results. The pressure coefficient \(C_p\) is calculated according to the following formula,

\[
C_p = \frac{P - P_{\text{ref}}}{\frac{1}{2} \rho U_{\text{ref}}^2}
\]

In which \(P_{\text{ref}}\) (\(P_{\text{ref}} = 72000\) Pa) and \(U_{\text{ref}}\) (\(U_{\text{ref}} = 180\) m/s) is the average pressure and velocity on the cross section (\(s/s_m = 0\)). It can be seen that there are some small deviations between the computed results and the experimental results, but the main trends are in good agreement. For the S-down, the numerical results are in good agreement with the experimental data (Figure 3(a)). For the S-up, there is 5% deviation between computed and experimental results (0 < \(s/s_m < 0.2\) and 0.6 < \(s/s_m < 0.8\), and there is a constant pressure region (0.35 < \(s/s_m < 0.6\), indicating the streamwise flow separation phenomenon (Figure 3(b)). Due to the curvature effect, a pair of streamwise counter-rotating vortices is presented on the AIP (Figure 3(c)).

**The mean flow field characteristics in the diffused S-duct**

**Computational results**

The streamwise velocity (\(U_s/U_{\text{ref}}\)) distributions on the six cross sections of S-duct are shown in Figure 4. The

| Parameter | Value |
|-----------|-------|
| \(D_{in}\), mm | 133.15 |
| \(D_{out}\), mm | 164 |
| \(L\), mm | 665.4 |
| \(H\), mm | 179.75 |
| \(r_0\), mm | 665.4 |

**Table 1.** Geometric dimensions of S-duct.
velocity distributions on six lines (L1–L6) are shown in Figure 5. In Figure 5, dimensionless coordinate \( \eta \) is given by

\[
\eta = \frac{r - r_0}{R}
\]  

(3)

The whole flow field of S-duct is divided into three parts: the entrance zone \((0 < s/s_m < 0.417)\), the transition zone \((0.417 < s/s_m < 0.583)\), and the exit zone \((0.583 < s/s_m < 1)\). In the entrance section, the fluid velocity near the upper wall is higher than that of the lower wall, and the pressure is lower. In the exit zone, the fluid velocity near the upper wall is lower than that of the lower wall, and the pressure is higher. In the transition zone, the streamwise velocity profile changes.

Figure 3. Comparisons between simulated results and literature results.\(^{22,23}\): (a) the surface pressure coefficient for S-down, (b) the surface pressure coefficient for S-up, and (c) the total pressure recovery coefficient on the AIP.

Figure 4. Streamwise velocity distributions \((U/U_{ref})\) on six cross sections.

Figure 5. Variation of dimensionless velocity \((U/U_{ref})\) with \( \eta \).
Due to the direction of the main flow in the transition zone change, it will cause the direction of the main flow not parallel to the centerline, and thus lead to the flow separation. It is helpful to reduce the separation area and improve the quality of the flow field that adjusting the direction of the main flow the transition zone. In order to better adjust the direction of the main in the transition zone, we first analyze the characteristics of main flow from the Navier-Stokes equations.

Establishment of theoretical model

The flow in the S-duct is mainly affected by streamwise adverse pressure gradient, the transverse pressure gradient, and the inertial centrifugal force. There is not only streamwise flow separation and secondary flow, but also the direction change of the main flow in the S-duct. The flow characteristics will affect the flow quality in the S-duct, and then affect the performance of the engine. In the present study, the flow is steady ideal incompressible flow and the incoming flow is uniform flow. In the entrance zone, the velocity distribution on the symmetry plane of the duct is that the velocity of the main flow near the inner wall is higher, the pressure is lower, and the velocity of the main flow near the outer wall is lower, the pressure is higher. In the exit zone, the velocity distribution is that the velocity of the main flow near the inner wall is higher, the pressure is lower, and the velocity of the main flow near the outer wall is lower, and the pressure is higher. The transition zone belongs to the adjustment zone of the direction of the main flow.

Without considering the boundary layer flow, we make some simplification and consider a curved duct with constant cross section. Ignore streamwise pressure gradient and gravity and adopt the coordinate system as shown in Figure 6. $U_r$ is radial velocity, $U_s$ is streamwise velocity, $U_\theta$ is velocity in $\theta$ direction. According to the flow characteristics, $U_r = 0$, $U_s = (r, s)$, $U_\theta = 0$

The continuity equation is,

$$\frac{\partial U_r}{\partial r} + \frac{1}{r} \frac{\partial U_s}{\partial s} + \frac{\partial U_\theta}{\partial \theta} + \frac{U_r}{r} = 0$$  \hspace{1cm} (4)

After simplification, we get,

$$\frac{\partial U_s}{\partial s} = 0$$  \hspace{1cm} (5)

The $r$-momentum equation,

$$U_r \frac{\partial U_r}{\partial r} + U_s \frac{\partial U_r}{\partial s} + U_\theta \frac{\partial U_r}{\partial \theta} + \frac{U_r^2}{r} = \frac{1}{\rho} \frac{\partial p}{\partial r}$$

$$+ \frac{\mu}{\rho} \left( \frac{\partial^2 U_r}{\partial r^2} + \frac{1}{r} \frac{\partial U_r}{\partial r} + \frac{1}{r^2} \frac{\partial^2 U_r}{\partial s^2} + \frac{\partial^2 U_r}{\partial \theta^2} + \frac{2}{r^2} \frac{\partial U_r}{\partial s} - \frac{U_r}{r^2} \right)$$  \hspace{1cm} (6)

After simplification, we get,

$$\frac{1}{\rho} \frac{\partial p}{\partial r} = \frac{U_r^2}{r}$$  \hspace{1cm} (7)

The $s$-momentum equation,

$$U_r \frac{\partial U_s}{\partial r} + U_s \frac{\partial U_s}{\partial s} + U_\theta \frac{\partial U_s}{\partial \theta} - \frac{U_r U_s}{r} = \frac{1}{\rho} \frac{\partial p}{\partial s}$$

$$+ \frac{\mu}{\rho} \left( \frac{\partial^2 U_s}{\partial r^2} + \frac{1}{r} \frac{\partial U_s}{\partial r} + \frac{1}{r^2} \frac{\partial^2 U_s}{\partial s^2} + \frac{\partial^2 U_s}{\partial \theta^2} + \frac{2}{r^2} \frac{\partial U_r}{\partial s} - \frac{U_s}{r^2} \right)$$  \hspace{1cm} (8)

After simplification, we get,

$$\frac{\partial^2 U_s}{\partial r^2} + \frac{1}{r} \frac{\partial U_s}{\partial r} - \frac{U_s}{r^2} = 0$$  \hspace{1cm} (9)

Considering $U_s = U_s (r)$, the equation (9) can be rearranged,

$$\frac{d^2 U_s}{dr^2} + \frac{1}{r} \frac{d U_s}{dr} - \frac{U_s}{r^2} = 0$$  \hspace{1cm} (10)

Rearrange equation (10),

$$\frac{d^2 U_s}{dr^2} + \frac{d}{dr} \left( \frac{U_s}{r} \right) = 0$$  \hspace{1cm} (11)

Integrating of both sides of equation (11),

$$\frac{1}{r} \frac{d}{dr} \left( r U_s \right) = C$$  \hspace{1cm} (12)

Integrating of both sides of equation (12),

$$U_s = \frac{C}{r} + \frac{K}{r} \quad (C, K \text{ are integral constants})$$  \hspace{1cm} (13)

It can be seen from equation (13) that the flow velocity is induced by the combination of forced vortex and free vortex. $\frac{C}{r}$ is induced by forced-vortex and $\frac{K}{r}$ is induced by free vortex. The flow field in the volute of centrifugal fan is a typical flow field induced by the combination of forced vortex (\$C/r\$) and free vortex (\$K/r\$). In the present study, the flow is steady ideal incompressible flow and the incoming flow is uniform flow. According to Helmholtz's vortex theorem, there is no
forced vortex in the flow field, so we only consider the free vortex induction, and $\frac{Q}{r}$ is omitted. Therefore,

$$U_s = \frac{K}{r}$$  \hspace{1cm} (14)

The volume flow through the duct is

$$Q = K \int_{r_0}^{r_0+R} \int \frac{2R^* \sin \varphi}{r_0-R} dr$$  \hspace{1cm} (15)

The following equations (16) and (17) can be derived from the geometric relationship

$$r = r_0 - R \cos \varphi$$  \hspace{1cm} (16)

$$dr = R \sin \varphi d\varphi$$  \hspace{1cm} (17)

By substituting equations (16) and (17) into equation (15), $K$ can be obtained

$$K = \frac{Q}{\int_{0}^{2\pi} \int_{r_0}^{r_0+R} \frac{2R^* \sin^2 \varphi}{r_0-R \cos \varphi} d\varphi}$$  \hspace{1cm} (18)

The comparison between the theoretical and numerical results is shown in Figure 7. In the numerical simulation, $K$ is obtained by the following formula

$$K = \frac{\int \int U_s^* rdA}{\int \int dA}$$  \hspace{1cm} (19)

Figure 7(a) shows the change of cross section radius with $s/s_m$. As the flow progressed downstream, the change of $(K^*D_m)/Q$ for numerical simulation and theoretical model are shown in Figure 7(b). It can be seen from Figure 7(a) that the cross section radius of the S-duct gradually increases in streamwise direction. Combined with equation (15), it can be concluded that when the volume flow rate of the duct is constant, the $K$ should gradually decrease in the streamwise direction, which can also be reflected in Figure 7(b). For the entrance zone of the S-duct ($0 < s/s_m < 0.417$), the results of numerical simulation are in good agreement with those of theoretical model, which indicates that the flow field in this region mainly presents potential flow behavior. For the transition zone and the exit zone of the S-duct ($0.417 < s/s_m < 1$), due to the existence of flow separation and secondary vortex flow, the numerical simulation results deviate greatly from the theoretical results, which shows that the flow field doesn’t purely presents potential flow behavior. In the transition zone, if the direction of the main flow can be adjusted to make the flow match the theoretical solution better, the flow separation area and secondary flow can be effectively reduced, and the uniformity of flow can be improved.

Adjusting the direction of the main flow by using guide vanes in the transition zone

The problem studied in this paper is that the direction change of the main flow in the case of a diffused S-duct with adverse pressure gradient. The main flow is affected by adverse pressure gradient, centrifugal force, viscous force, and inertial force. In this paper, the uniformity of flow field is improved by adjusting the direction of the main flow in the transition zone. The goal of adjusting the direction of the main flow in the transition zone is to reduce the flow field distortion and make the flow field more uniform without increasing the total pressure loss.

In order to better adjust the direction of the main flow, two guide vanes (Figure 8(a) and (b)) are set in the transition zone. The function of the guide vanes is to adjust the direction of the main flow, restrain the streamwise flow separation and secondary vortex flow in the S-duct, and improve the quality of the flow field in the cross section of the S-duct (reducing the total pressure loss and improving the uniformity of the velocity distribution on the cross section). The design of the guide vanes is based on the following principles: (1) due to guide vanes control the direction of the main flow, guide vane should be located in the main flow zone; 2) in order to avoid the corner vortex flow located in the interface between the guide vanes and the S-duct, guide vanes should not be completely connected to the wall of S-duct; (3) in order to control the flow field to a large extent and have a certain distance from the wall, the spacing between the deflectors is 0.4 times diameter of the cross section ($s/s_m = 0.5$).

According to the above design principles, the geometric dimensions of the guide vanes are as follows. The width and length of the guide vane (scheme1) are $L_a = 65$ mm and $L_b = 68$ mm, respectively. The length of connecting rod $L_c$ is 138 mm. The distance between the guide vanes (scheme 1) $L_e$ is 58 mm. The cross section edge length of the connecting rod $d$ is 5 mm. The thickness of the guide vane (scheme 1) is also 5 mm (Figure 8(d)). The guide vanes for the scheme1 are connected with the S-duct wall through A, B, C, D, E, F, G, and H (Figure 8(d)). The difference between scheme 2 and scheme 1 is that the guide vanes (scheme 2) are completely connected to the wall of S-duct (Figure 8(d)). The edges of the guide vane are obtained by translating the center line ENF of the S-duct (0.482 < $s/s_m < 0.564$, $r = 665.4$ mm) (Figure 8(c)). The center line EN (0.482 < $s/s_m < 0.5$, $r = 665.4$ mm) has opposite curvature to the center line NF (0.5 < $s/s_m < 0.564$, $r = 665.4$ mm) and is tangent to it at $N$ ($s/s_m = 0.5$) (Figure 8(c)). The placement direction of the guide vanes is also parallel to the center line (0.482 < $s/s_m < 0.564$) (Figure 8(c)). The main reason why the guide vanes are not connected to the wall of S-duct is...
to avoid the corner vortex flow between the guide vanes and the S-duct (Figure 8(e)). Therefore, we finally choose the guide vanes (scheme 1).

The dimensionless streamwise velocity \( \frac{U_s}{U_{ref}} \) distributions on six cross sections for S-duct with guide vanes are shown in Figure 9. The velocity profiles on six lines for the S-duct with guide vanes are shown in Figure 10. Compared with Figures 4 and 5, it can be seen that due to the presence of guide vanes, the low-speed area near the upper wall surface of the exit zone is reduced.

Figure 11 shows the comparison of \( \frac{K^*D_m}{Q} \) for S-duct with/without guide vanes. For the entrance zone of the S-duct \((0 < s/s_m < 0.417)\), the \( \frac{K^*D_m}{Q} \) for S-duct with/without guide vanes are in good agreement with those of theoretical model. For the transition zone of the S-duct \((0.417 < s/s_m < 0.583)\), the results show that the flow field of S-duct without guide vanes deviates greatly from the theoretical model, while the flow field of S-duct with guide vanes is basically in good agreement with theoretical model. Because the curvature of the entrance zone is opposite to that of the exit zone, the direction of the main flow is changed in the transition zone, guide vanes set in the transition zone can well adjust the direction of the main flow. Here we can see that the adjustment effect of guide vanes on the

![Figure 7](image_url)

**Figure 7.** Comparison between theoretical results and numerical results: (a) the change of cross section radius, (b) theoretical and numerical results, (c) velocity profile on L1, (d) velocity profile on L2, and (e) velocity profile on L3.
the exit zone of the S-duct (0.583 < s/sn < 1), due to the existence of flow separation and secondary flow, both of them deviate from the theoretical model, which shows that the main flow field does not have potential flow characteristics.

Figure 12 shows the streamline on the symmetry plane for S-duct with/without guide vanes. As the cross-sectional area of S-duct gradually increases in the transition zone: (a) the position of guide vanes, (b) the mesh for the S-duct with guide vanes, (c) the symmetry plane of guide vanes, (d) the geometric dimensions of guide vanes, and (e) secondary flow streamlines on cross section (s/sn = 0.5).

Figure 9. Streamwise velocity distributions (U/Uref) on six cross sections for S-duct with guide vanes.

Figure 10. Velocity profile on six lines for S-duct with guide vanes.
streamwise direction, streamwise adverse pressure gradient decelerates the flow velocity. At the same time, due to the curvature of the central line for the entrance zone and the exit zone are opposite, the direction of main flow in the transition zone is changed and there is the large streamwise adverse pressure gradient near the upper wall, so the flow separation area appears near the upper wall for the exit zone. By comparing Figure 12(a) and (b), we find that the guide vanes have an obvious adjustment effect on the direction of the main flow in the transition zone for S-duct, reducing the size of the separation zone and the total pressure loss.

Figure 13 shows the surface pressure coefficient for S-duct with/without guide vanes. It can also be seen from Figure 13(b) that there is a constant pressure area on the L-up line ($0.4 < s/s_m < 0.6$), which implies the phenomenon of flow separation. In the downstream of the transition zone ($0.6 < s/s_m < 1$), the pressure coefficient for S-duct with guide vanes is higher.

Due to the curvature of the center line of the S-duct, there will be secondary flow in the cross section of the duct under the combined action of centrifugal force and radial pressure gradient. A pair of vortices will appear in the cross section and gradually strengthen downstream (Figure 14). It can be seen Figure 14(b) that guide vanes can weaken the secondary flow, which helps to make the flow field uniform and reduce the distortion coefficient of the flow field. Figure 15 shows the dimensionless vorticity ($\omega^*D_in$) contour maps of different sections for S-duct with/without guide vanes. Figure 17 shows the comparison of dimensionless vortex strength ($\Gamma^*D_in$) for S-duct with/without guide vanes. The secondary vortex strength $\Gamma$ is obtained by the following formula,

$$\Gamma = \iint |\omega|^*dA'$$
In which, $A_0$ is the area (Figure 16) enclosed by the curve surrounding the secondary vortex. With the increase of $A_0$, $\frac{\int \omega \cdot dA'}{C_{13} v_{jj}^*} dA_0$ increases first and then remains unchanged. When $\int \omega \cdot dA'$ remains unchanged, the value is taken for $G$.

It can be seen from Figure 17 that the strength of secondary vortices gradually increases in the streamwise direction, and the strength of secondary vortices is relatively weakened in the S-duct with guide vanes.

The distribution of turbulent kinetic energy on different cross sections is shown in Figure 18, from which it can be seen that the turbulent kinetic energy increases gradually in the streamwise direction. Figure 19 shows the change curve of average turbulent kinetic energy for different cross sections versus $s/s_m$, combined with Figure 18(a) and (b), it can be seen that the average turbulent kinetic energy on the cross sections is relatively smaller in the S-duct with guide vanes, which is mainly because the guide vanes weaken the secondary vortex strength and flow separation.

Figure 20 shows the contour map of total pressure distribution on different cross sections for S-duct with/without guide vanes. The three-dimensional streamline is shown in Figure 21. Due to the present of secondary vortices and flow separation, there will be a low pressure area in the upper part of cross section, and the range of low pressure area will gradually increase in streamwise direction. In the S-duct with guide vanes, the range of low pressure area will be smaller.

The variation of total pressure recovery coefficient and distortion coefficient with $s/s_m$ is shown in Figure 22. Total pressure recovery coefficient $\sigma$ and distortion coefficient $DC(60)$ are given by

$$\sigma = \frac{P_{0,\text{avg}}}{P_{0,\text{ref}}}$$

$$DC(60) = \frac{P_{0,\text{avg}} - P_{0,60\deg}}{q_{\text{avg}}}$$

In which, $P_{0,\text{avg}}$ is average total pressure on a cross section, $P_{0,\text{ref}}$ is reference average total pressure at inlet of S-duct, $P_{0,60\deg}$ is average total pressure in the $60^\circ$ sector that results in the lowest value, and $q_{\text{avg}}$ is average dynamic pressure on a cross section. For S-duct with guide vanes, the total pressure recovery coefficient on the downstream section of guide vanes is relatively higher, and the distortion coefficient is relatively lower. It can also be seen from Figure 21 that guide vanes can adjust the direction of the main flow in transition zone, and inhibit the secondary vortex and streamwise flow.
The uniformity of flow field is better for S-duct with guide vanes. The comparison of total pressure recovery coefficient and distortion coefficient on AIP section is given in Table 2, it can be seen that the total pressure recovery coefficient increases and the distortion coefficient decreases for the S-duct with guide vanes, and the quality of flow field is improved. The total pressure recovery coefficient of AIP section is increased by 0.34%, and the distortion coefficient is decreased by 10.5%.

Conclusions

Based on the detailed analysis of the flow characteristics in S-duct, in order to better improve the flow field quality, a new method adjusting the direction of the main flow by using guide vanes in transition zone is proposed. In order to determine the effect of guide vanes on the flow field, RANS equations are employed to simulate the flow field for a 30°/30° S-duct with/without guide vanes. By comparing the flow field in S-duct with/without guide vanes, the following conclusions can be obtained:

1. The method adjusting the direction of the main flow by using guide vanes in transition zone is beneficial to improve the quality of flow field. In transition zone of S-duct, guide vanes can not only adjust the direction of the main flow, but also restrain the flow separation, reduce the total pressure loss, increase the diffusivity of the secondary vortex, and decrease the strength of the secondary vortex. For S-duct with guide...
vanes, due to present of the guide vanes, the total pressure recovery coefficient on the AIP section is increased by 0.34%, and the distortion coefficient is decreased by 10.5%.

(2) In the entrance zone, the main flow in the core zone of S-duct with/without guide vanes are in good agreement with theoretical mode and present potential flow behavior. In the transition zone, the main flow of S-duct without guide vanes deviates from the theoretical model, and the main flow of S-duct with guide vanes is in good agreement with theoretical mode. In the exit zone, both of them deviate from the theoretical model.
Table 2. Comparison of performance parameters (AIP) for S-duct with/without guide vanes.

|                  | Without guide vanes | With guide vanes | Relative increasement |
|------------------|---------------------|-----------------|-----------------------|
| \( \sigma \)     | 0.9732              | 0.9765          | 0.34%                 |
| \( DC(60) \)     | 0.4362              | 0.3902          | -10.5%                |

Declaration of conflicting interests

The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

Funding

The author(s) received no financial support for the research, authorship, and/or publication of this article.

ORCID iD

Shuili Ren https://orcid.org/0000-0003-2463-7834

References

1. Tormalm M. Design and analysis of compact UAV ducts. In: 24th AIAA applied aerodynamics conference, San Francisco, CA, 2006.
2. Kamali S, Reza Ahrabi B, Webster RS, et al. Numerical simulation of compressible flow in a diffusing S-duct with and without vortex generators. In: 33rd AIAA applied aerodynamics conference, Dallas, TX, 2015.
3. Gerolymos GA, Joly S, Mallet M, et al. Reynolds-Stress model flow prediction in aircraft-engine intake double-s-shaped duct. J Aircr 2010; 47: 1368–1381.
4. Johansson M. Propulsion integration in an UAV. In: 24th AIAA applied aerodynamics conference, San Francisco, CA, 2006.
5. Kirk A, Kumar A, Gargoloff J, et al. Numerical and experimental investigation of a serpentine inlet duct. In: 45th AIAA aerospace sciences meeting and exhibit, Reno, NV, 2007.
6. Anderson BH, Reddy DR and Kapoor K. Study on computing separating flows within a diffusion inlet S-duct. J Propulsion Power 1994; 10: 661–667.
7. Wellborn SR, Reichert BA and Okiishi TH. Study of the compressible flow in a diffusing S-duct. J Propulsion Power 1994; 10: 668–675.
8. Wendt BJ and Reichert BA. Vortex ingestion in a diffusing S-duct inlet. J Aircr 1996; 33: 149–154.
9. Brear MJ, Warfield Z, Mangus JF, et al. Flow separation within the engine inlet of an uninhabited combat air vehicle (UCAV). J Fluid Eng 2004; 126: 266–272.
10. Harloff GJ, Smith CF, Bruns JE, et al. Navier-Stokes analysis of three-dimensional S-ducts. J Aircr 1993; 30: 526–533.
11. Berrier B and Allan B. Experimental and computational evaluation of flush-mounted, s-duct inlets. In: 42nd AIAA aerospace sciences meeting and exhibit, Reno, NV, 2004.
12. Foster J, Wendt BJ, Reichert BA, et al. Flow through a rectangular-to-semiannular diffusing transition duct. J Propulsion Power 1997; 13: 312–317.
13. Fiola C and Agarwal RK. Simulation of secondary and separated flow in diffusing S ducts. J Propulsion Power 2015; 31: 180–191.
14. Gil-Prieto D, MacManus DG, Zachos PK, et al. Delayed detached-eddy simulation and particle image velocimetry investigation of S-duct flow distortion. AIAA J 2017; 55: 1893–1908.
15. Lin JC. Review of research on low-profile vortex generators to control boundary-layer separation. Prog Aerosp Sci 2002; 38: 389–420.
16. Vakili A, Wu J, Liver P, et al. Flow control in a diffusing S-duct. In: Shear flow control conference, Boulder, Colorado, 1985.
17. Sullerey RK, Mishra S and Pradeep AM. Application of boundary layer fences and vortex generators in improving performance of S-duct diffusers. J Fluid Eng 2002; 124: 136–142.
18. Vakili A, Wu J, Liver P, et al. Passive flow control study in an S-duct using stereo particle image velocimetry. AIAA J 2017; 55: 1862–1877.
19. Liu M, Tan L and Cao S. Theoretical model of energy performance prediction and BEP determination for centripetal pump as turbine. Energy 2019; 172: 712–732.
20. Liu Y and Tan L. Theoretical prediction model of Tip leakage vortex in a mixed flow Pump with Tip clearance. J Fluid Eng 2020; 142: 021203-1–021203-11.
21. Liu M, Tan L and Cao S. Influence of geometry of inlet guide vanes on pressure fluctuations of a centrifugal Pump. J Fluid Eng 2018; 140: 091204.
22. Delot A-L, Garnier E and Pagan D. Flow control in a high-offset subsonic air intake. In: 47th AIAA/ASME/SAE/ASEE joint propulsion conference and exhibit, San Diego, CA, USA, 2011.
23. Reza Ahrabi B, Sreenivas K and Webster RS. Computational investigation of compressible flow in a diffusing S-duct. In: 49th AIAA/ASME/SAE/ASEE joint propulsion conference, San Jose, CA, 2013.

Appendix

Notations

- \( D \) = cross section diameter of S-duct
- \( D_{in} \) = inlet diameter of S-duct
- \( D_{out} \) = outlet diameter of S-duct
- \( R \) = cross section radius of S-duct
- \( R_{in} \) = inlet radius of S-duct
- \( A \) = cross section area
- \( r \) = radial coordinate
- \( s \) = streamwise coordinate
- \( s_m \) = the length of center line for S-duct
- \( \theta \) = circumferential coordinate of cross section
- \( \eta \) = dimensionless coordinate
- \( Q \) = volume flow of S-duct
- \( L \) = horizontal length of S-duct
- \( H \) = offset distance of S-duct
\[ r = \text{radius of center line for S-duct} \]
\[ \sigma = \text{total pressure recovery coefficient} \]
\[ \text{DC}(60) = \text{distortion coefficient} \]
\[ C_p = \text{pressure coefficient} \]
\[ \omega = \text{vorticity} \]
\[ \Gamma' = \text{secondary vortex intensity} \]
\[ A' = \text{the area enclosed by the curve surrounding the secondary vortex} \]
\[ T_0 = \text{total temperature} \]
\[ P_0 = \text{total pressure} \]
\[ P_{0\text{-inlet}} = \text{total pressure for the inlet of computational domain} \]
\[ P_{0\text{-ref}} = \text{reference average total pressure at inlet of S-duct} \]
\[ P = \text{static pressure} \]
\[ P_{\text{ref}} = \text{reference average static pressure on the cross section} \ (s/s_m = 0) \]
\[ P_{\text{outlet}} = \text{static pressure for the outlet of computational domain} \]
\[ P_{0\text{-avg}} = \text{average total pressure on cross section} \]
\[ P_{0\text{-60deg}} = \text{average total pressure in the 60° sector that results in the lowest value} \]
\[ q_{\text{avg}} = \text{average dynamic pressure on cross section} \]
\[ U = \text{streamwise velocity} \]
\[ U_{\text{ref}} = \text{reference average velocity on the cross section} \ (s/s_m = 0) \]
\[ d = \text{thickness of guide vane} \]
\[ L_a = \text{width of guide vane} \]
\[ L_b = \text{length of guide vane} \]
\[ L_c = \text{length of connecting rod of guide vane} \]
\[ l = \text{spacing between guide vanes} \]
\[ t = \text{turbulent kinetic energy} \]
\[ \text{AIP} = \text{aerodynamic interface plane} \ (0.5D_m \text{ downstream of the outlet for S-duct}) \]