JOURNAL OF MECHANICAL ENGINEERING AND SCIENCES (JMES)
ISSN: 2289-4659 e-ISSN: 2231-8380
VOL. 14, ISSUE 4, 7446 – 7468
DOI: https://doi.org/10.15282/jmes.14.4.2020.13.0587

ORIGINAL ARTICLE

Review on the aerodynamics of intermediate compressor duct

M. Sharma, B.D. Baloni

Mechanical Engineering Department, Sardar Vallabhbhai National Institute of Technology, Surat, India
Phone: +918890162865; Fax: +0261-2228394

ABSTRACT – In a turbofan engine, the air is brought from the low to the high-pressure compressor through an intermediate compressor duct. Weight and design space limitations impel to its design as an S-shaped. Despite it, the intermediate duct has to guide the flow carefully to the high-pressure compressor without disturbances and flow separations hence, flow analysis within the duct has been attractive to the researchers ever since its inception. Consequently, a number of researchers and experimentalists from the aerospace industry could not keep themselves away from this research. Further demand for increasing by-pass ratio will change the shape and weight of the duct that uplift encourages them to continue research in this field. Innumerable studies related to S-shaped duct have proven that its performance depends on many factors like curvature, upstream compressor's vortices, swirl, insertion of struts, geometrical aspects, Mach number and many more. The application of flow control devices, wall shape optimization techniques, and integrated concepts lead a better system performance and shorten the duct length. This review paper is an endeavor to encapsulate all the above aspects and finally, it can be concluded that the intermediate duct is a key component to keep the overall weight and specific fuel consumption low. The shape and curvature of the duct significantly affect the pressure distortion. The wall static pressure distribution along the inner wall significantly higher than that of the outer wall. Duct pressure loss enhances with the aggressive design of duct, incursion of struts, thick inlet boundary layer and higher swirl at the inlet. Thus, one should focus on research areas for better aerodynamic effects of the above parameters which give duct design with optimum pressure loss and non-uniformity within the duct.

ARTICLE HISTORY
Received: 20th Aug 2019
Revised: 06th Feb 2020
Accepted: 22nd Mar 2020

KEYWORDS
S-shaped intermediate duct;
CFD;
Curvature;
Pressure loss;
Flow uniformity

INTRODUCTION

Commercial aircraft engines demand lower noise and less specific fuel consumption (SFC). Turbofan engine meets these requirements by by-passing some portion of the total airflow from the compressor, combustion chamber, turbine, and finally through the nozzle. Long-range Civilian engines like GE90, GP7000 and Rolls-Royce Trent 1000 are operated at a high bypass ratio of 8-11 to lower the specific fuel consumption. This SFC also helps to improve engine fuel efficiency [1]. However, military aircraft engines like Boeing KC-135 and Lockheed C-5 Galaxy were designed for a lower bypass ratio of 0.5-1 in order to keep the lesser frontal area and quicker throttle response. The further increment into the bypass ratio of aero engines is limited by the lower tip speed [2]. However, for adequate bypass ratio and overall pressure ratios a twin-spool arrangement is most preferable. The difference of the Meridional flow path of the SFC optimized twin-spool turbofan engine with a bypass ratio of 6 and 10 can be estimated from Figure 1. Flow has to be directed from low-pressure system to high-pressure system and vice versa through annular ducts with a particular radial offset ($\Delta R/L$), hence higher the bypass ratio, larger the radial offset and disk bore diameter.

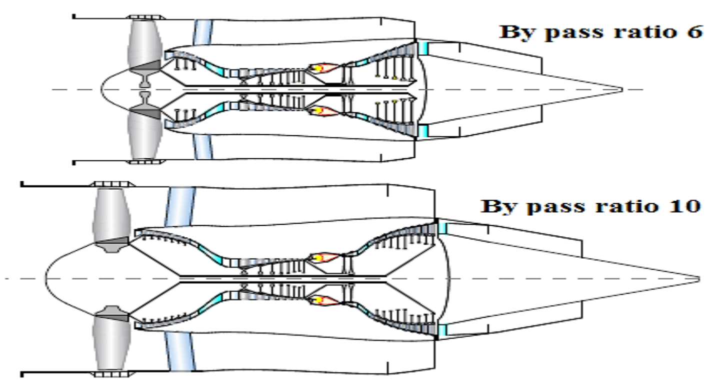


Figure 1. Turbofan engines with a bypass ratio of 6 and 10 [2]

© The Authors 2020. Published by Penerbit UMP. This is an open access article under the CC BY license.

In a twin-spool arrangement, interconnecting ducts are used to make flow continuity between the turbo-machinery passages. One reason behind to design the twin-spool engine is to obviate the blades to be stalled due to incorrect axial velocity under the off-design conditions. Although blowing-off the air at few rear stages might be an option, it is not a better solution. Most of the compressed air will be wasted; hence a multi-spool arrangement gives a more satisfactory solution [1]. Despite it, one more problem arises with the multistage compressor whenever it operates under off-design conditions. If the speed of the compressor's first few stages is reduced from its design value, the incidence angle will be increased and at the rear stages, it is decreased. This incidence angle can be maintained at its design value by increasing the speed of the rear stages as well as reduction at the upstream stages hence, multi-staging of the compressor might be the better choice. Besides the compressors, the application of the interconnecting ducts can be found in-between various components of the gas turbine which is shown in Figure 2.

As air intakes, it acts as a subsonic diffusing duct to supply the ambient air from the wing and fuselage to the compressor inlet for thrust generation. These ducts mainly are designed to efficiently decelerate the incoming air-flow to the desired value in order to achieve maximum total pressure recovery and ensure flow uniformity to the compressor inlet face. As an inter-stage turbine duct, it continues to flow from the high-pressure turbine to low- pressure turbine. Duct connecting low to the high-pressure compressor is designed with virtually negligible diffusion rate, hence it ceases to constant area ratio (AR) whereas inter-stage turbine duct shows area variation throughout the passages hence, a significant amount of diffusion is taken place [3]. Moreover, optimization and weight penalties dictate that the duct must be as short as possible. Hence, it is designed as an S-shaped and this curved S-shape passage gives rise to the secondary as well as three-dimensional effects to the flow.

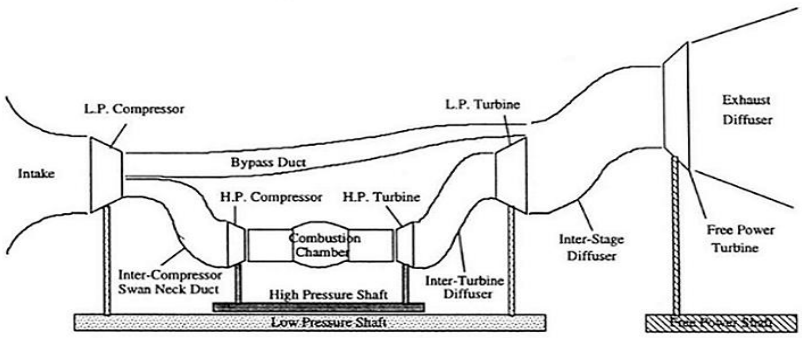


Figure 2. Gas turbine components [4]

Based on the complexity of the aero-engine design, the intermediate compressor duct might either be a clean annular channel or be designed with load-carrying struts for structural support as well as oil and cooling service. These non-lifting struts are designed like airfoils shape with negligible occurrence to flow disturbance. Apart from it, lifting struts are also used to make a more integrated design which further helps to reduce the overall flow path length. This integrated design adds more complexity to the duct so it becomes necessary to optimize the shape of the intermediate compressor duct with wall contouring. Moreover, in order to completely avoid or delay the flow separation within the S-shaped duct active and passive flow control techniques are used. More detail about these measures will be discussed in subsequent sections.

This paper reviews the progress in the field of S-shaped intermediate compressor duct by emphasizing the past and recently published research work related to this topic. Flow within the annular S-shaped duct is influenced by turbulence, Reynolds number, inlet flow conditions and finally Mach number. The summary of available research on the basis of the above aspects has been presented here. Apart from it, future research work and challenges have also been outlined on the basis of this review.

MECHANISMS INSIDE THE CURVED DUCT

Because of weight and space penalties, the annular intermediate duct is deliberately designed as an S-shaped form. The inner wall of the compressor intermediate S-shaped duct is made of convex followed by concave whereas the outer wall consists of concave followed by convex curvature. These curvatures provoke secondary flow within the annular duct so flow becomes complex. In fact, fluid under the curvatures is subjected by the combined effect of the radial pressure gradient and centrifugal force, hence detail flow understanding within the S-shaped duct has always been challenging for the researchers.

As soon as flow approaches to the outer wall of the first bend, its pressure will get higher than the inner wall hence, across the first bend, a pressure gradient is always observed. However, this occurrence is altered within the second bend. It is found that curvature is responsible for a pressure gradient along the stream-wise as well as radial direction and therefore boundary layer development along the curvature of the duct will significantly get modified. In addition to it, curvature induces an imbalance between the centrifugal acceleration of the fluid and its surrounding pressure field hence, turbulence mixing is get enhanced over the concave curvature whereas it gets suppressed over the convex surface.

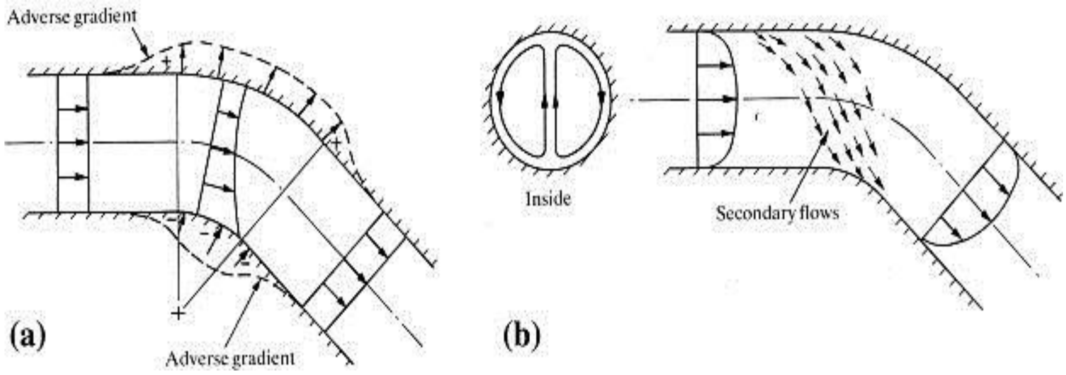
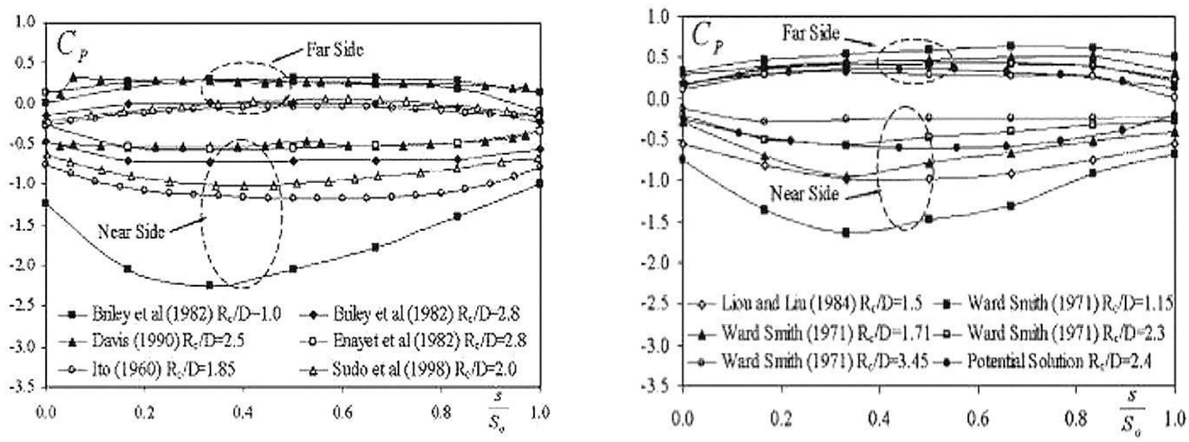


Figure 3. Radial pressure gradient and secondary flow evolution [5]

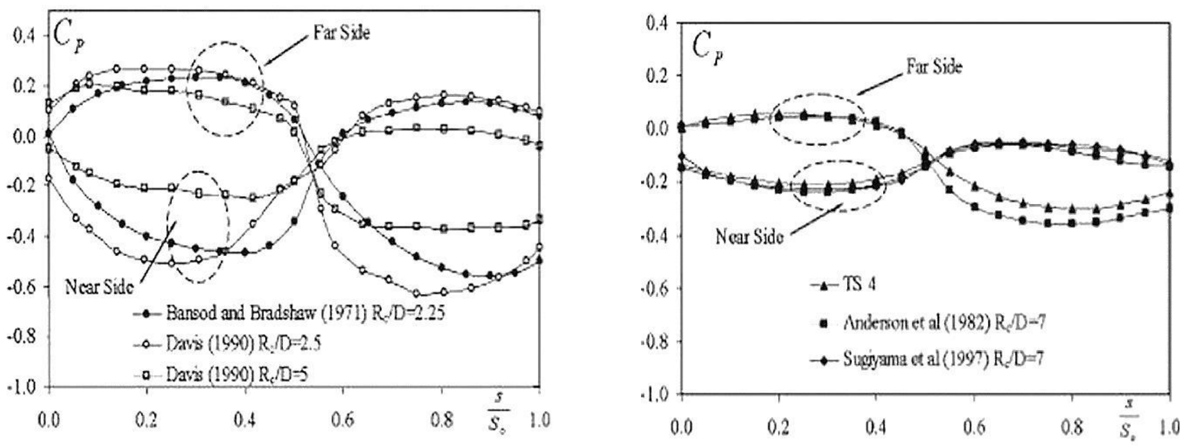
Fluid along the axis of the duct is subjected to a larger centrifugal force due to having high velocity than the slower moving fluid adjacent to the duct walls. Consequently, it starts to move away from the center axis towards the outer wall of the duct. As soon as, this core fluid approaches the outer wall, it has to deal with an adverse pressure gradient because of concave curvature of the wall and eventually it becomes slow down. This energy paucity fluid cannot vanquish to the adverse pressure gradient, therefore, fluid adjacent to the outer wall does not get moved longer along the duct wall and eventually, it returns back to the center axis as shown in Figure 3. This flow is known as secondary flow and an aggressive S-shaped duct (duct having a low radius of curvature) always contains the secondary flow.

Variation of C_p along the Walls for Different Curved Geometries

It is evident that curvature plays a vital role in S-shaped duct flow and significantly affects the pressure distribution within the duct. Flow under the curvature is subjected by the combined effect of the pressure gradient and centrifugal force, hence, duct having a small radius of curvature (sharp turn) exhibits a higher pressure difference along the sidewalls as well as between the walls than the duct having a high radius of curvature (moderate turn). Hence, the duct curvature can be classified as mild curvature ($\delta/R \approx 0.01$) and strong curvature ($\delta/R \approx 0.1$). Bansod and Bradshaw [6] investigated the effect of curvature for an S-shaped circular duct having a different radius of curvature (R) and found that duct having a small radius of curvature exhibits a strong adverse pressure gradient along the near side (inner wall). Like the S-shaped duct, 90° circular bend also exhibits pressure gradient along the walls, however, the variation would not be like sinusoidal form [7–9] . Subsequently, apart from the circular duct, flow through the square duct had also been investigated and it was understood that as increasing the R/D ratio, the duct becomes less curved consequently, radial pressure gradient along with the walls decreases [10, 11]. In order to differentiate the pressure gradient variation for different curvature ratios of the square and circular duct, Ng et al. [5] made an effort to present the pressure gradient variation for the different duct into a single frame. Figure 4 depicts the C_p variation along the walls of the square and circular duct for 90°curved and S-shaped geometries having different curvature ratios. Authors noted that within the first bend, the outer-wall of both 90°curved and S-shaped duct exhibited higher C_p distribution compared to the inner-wall. However, C_p distribution of the square and circular S-shaped duct followed a sinusoidal-like variation along the center-line because of the continuous alteration of the curvature. They concluded that duct having a high curvature ratio (R_c/D) became less curve consequently, it showed a slight reduction in the difference of the radial pressure between the walls.



(a) Circular cross-sectional 90° curved duct (b) Square cross-sectional 90°curved duct



(c) Circular cross-sectional S-shaped duct (d) Square cross-sectional S-shaped duct

Figure 4. Variation of wall C_p along the duct length [5] (Reproduced from [5], with the permission of AIP Publishing)

Thus, available literature reveals that the R/D ratio plays a significant role in pressure distribution. If it is small then as per aircraft requirement length of duct is small but it increases strong adverse pressure gradient along the walls and eventually non-uniformity of flow increases. Whereas, one can decrease the non-uniformity by increasing the R/D ratio with compromise on optimum length of the duct.

PERFORMANCE PARAMETERS OF INTERMEDIATE COMPRESSOR DUCT

Coefficient of Static Pressure (C_{ps}):

Performance of the axisymmetric and non-axisymmetric duct can be evaluated by C_{ps} as shown in Eq. (1). Basically, it is used to know the energy transformation within the passage [3]. In this, P_{sx} and P_{si} refer to static pressure at any location and inlet, respectively.

$$C_{ps} = \frac{2(P_{sx} - P_{si})}{\rho U_{avg}^2} \quad (1)$$

Ideal Coefficient of Static Pressure (C_{psi}):

It indicates the outlet pressure recovery of an ideal diffuser or duct where the flow is frictionless and uniform throughout the duct [3]. In this case, it is represented in terms of area ratio (AR) in the absence of stagnation pressure loss as presented by Eq. (2).

$$C_{psi} = 1 - \frac{1}{AR^2} \quad (2)$$

Effectiveness of Duct (ε):

If a duct does not have a constant area than its pressure recovery rate in comparison to ideal duct/diffuser is represented by ε [3] as shown in Eq. (3)

$$\varepsilon = \frac{C_{ps}}{C_{psi}} \quad (3)$$

Coefficient of Total Pressure Loss (C_{pt}):

The total pressure loss coefficient for a duct or diffuser can be calculated from Eq. (4). It depicts how much of the total pressure has been lost in order to overcome the viscous effect as well as the turbulence of the fluid [3, 12]. P_{ti} and P_{to} stand for the total pressure at inlet and outer, respectively.

$$C_{pt} = \frac{2(P_{ti} - P_{to})}{\rho U_{avg}^2} \quad (4)$$

Non-Uniformity Index (S_{io}):

It shows the percentage amount of the secondary flow at a particular location [12] and represented by Eq. (5).

$$S_{io} = \frac{\left(\sum \sqrt{V_y^2 + V_z^2}\right)}{nU_{avg}} \tag{5}$$

Distortion Coefficient (DC):

It shows that how much flow has been distorted at a particular location due to existing secondary flow and vortices [13] and given by Eq. (6). $\bar{P}_0(\emptyset)$ and $\bar{q}$ denotes the mean total pressure of $\emptyset$ degree sector and dynamics pressure at outlet, respectively.

$$\frac{\overline{P_0} - \overline{P_0}\,(\emptyset)}{\bar{q}} \tag{6}$$

FUNDAMENTAL AERODYNAMIC OF INTERMEDIATE COMPRESSOR DUCT

Flow within the annular S-shaped intermediate duct is dominated by the combined effect of the curvature and pressure gradient. Boundary layer development inside the duct is also influenced by the stream-wise and radial pressure gradient. Since the intermediate duct is placed between the compressors, it becomes more interesting to investigate the flow behavior under the wakes induced by the upstream compressor. In order to provide structural support and to facilitate the cooling, sometimes struts are provided within the duct however, these cause extra flow blockage in the duct. Furthermore, flow within the S-shaped duct is affected by the inlet conditions like swirl, Mach number and many more. Therefore, the aerodynamics study of the intermediate duct gives under these environments gives a significant insight. In the beginning, most of the researches were dedicated to understanding the basic flow physics of the S-shaped duct under the influence of the curvature and pressure gradient. But lately, the studies based on optimization techniques to overcome the space and weight penalties have also emerged. The behavior of flow under different environments is discussed in subsequent sections.

Influence of Wall Curvature

Wall curvature is one of the most important factors of a duct as it is not only responsible for radial and stream-wise pressure gradient but also affects the flow field and as a consequence of the above, wall boundary layers and velocity profiles are significantly altered. This is the reason why a huge majority of researchers have given paramount importance to the influence of wall curvature on the performance of the S-shaped duct in their studies and proved that it causes secondary flow and counter-rotating vortices into the duct. Literature reveals that a larger secondary flow formed in the laminar case because of thicker boundary layer development. However, in the turbulent flow case, a thinner boundary layer was developed compared to laminar one. The shape of the duct does not affect the thickness and magnitude of secondary flow in the case of the laminar boundary layer whereas the effects are there for the turbulent boundary layer.

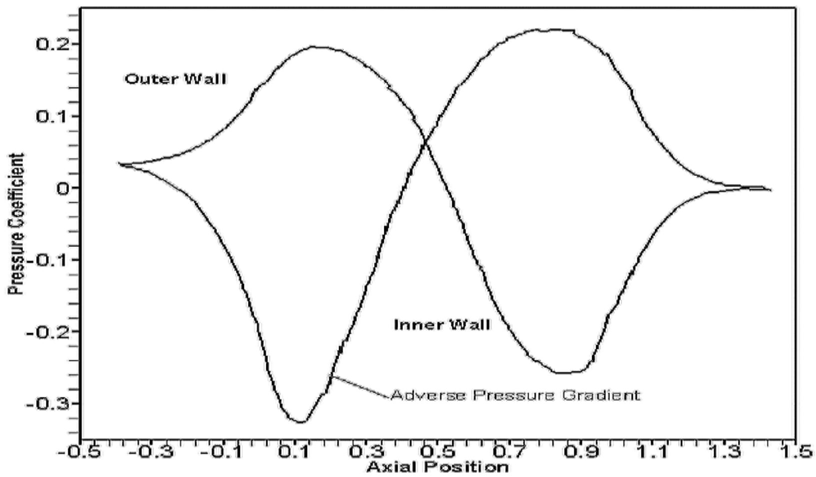


Figure 5. Axial variation of static pressure coefficient

The secondary flows indeed engendered a high level of total pressure loss and in fact, the magnitude and distribution of the secondary flow also depend on the duct shape as well as curvature. Apart from this, flow uniformity is one of the main objectives and many researchers have also focused their research on how much flow is being distorted under secondary flow and vortices. Exit flow is usually considered as uniform only if the magnitude of the secondary flow is less than 10% of the average velocity. Lots of work has been done on the prediction of the uniformity with offsetting by keeping the inlet and outlet in two different horizontal planes. Offsetting basically changes the difference between the mean radius of inlet and outlet of S-duct and it can be concluded that an increase in the offsetting also enhances the exit losses in order to accommodate the increase in non-uniformity at the outlet [12, 14]. Also, secondary flow and counter-rotating vortices were induced due to strong curvature. Boundary layer separation is likely to be occurring along the inner wall of the first bend due to a large adverse gradient as shown in Figure 5. A summary of the influence of wall curvature on the flow development of the S-shaped duct is shown in Table 1.

Table 1. Summary of influence wall curvature on duct performance

Authors	Type of Duct	Significant Findings
Vakili et al. [15, 16], Wellborn et al. [17]	Circular S-duct R/D_h =5	- The adverse pressure gradient thickens the boundary layer. - Counter-rotating secondary flow found throughout the duct passage, however, it was strong in the second bend. - A large extent of the low-pressure zone was identified near the outer wall of the second bend. - These vortices deteriorated the flow uniformity and caused additional pressure losses.
Taylor et al. [18]	Square S-duct R/D_h=7	- The laminar boundary layer occupies 25% of passage height whereas the turbulent layer covers 15% of passage height. - Due to the thicker boundary layer, a larger secondary flow formed in the laminar case
Taylor et al. [19]	Circular S-duct R/D_h=7	- The laminar boundary layer occupies 25% of passage height whereas the turbulent layer covers 10-20% of passage height. - The larger and stronger secondary flow was observed in the laminar flow compared to turbulent flow.
Abdellatif [20]	Rectangular S-duct	- Duct shape and curvature significantly affected the pressure distributions. - The reversal of the duct curvature also altered the pressure variations along the walls. - The maximum core flow shifted from the inner wall to the outer wall as the flow passes from inlet to outlet. - The maximum turbulent kinetic energy was found near the concave curvature in both bends.
Vaccaro et al. [21]	Rectangular S-duct L/D_h =1.5	- Counter-rotating vortices enhanced total pressure loss at the aerodynamic interface plane. - Lower surface was more prone to separation due to strong secondary flow.
Gopaliya et al. [12, 14]	Rectangular to semicircular S-duct	- The coefficient of pressure recovery (C_{ps}), effectiveness and non-uniformity decreased with the increase of offsetting .
Anand et al. [22]		- Even for the uniform flow applied at the inlet, the secondary flow was presented throughout the duct passage due to strong curvature.
Harloff et al. [23–25]	Diffusing and non-diffusing S-duct	- O-grid predicted a better velocity field and smooth turbulent viscosity variations than the H-grid. - The separation bubble was greater for H-grid therefore; a lower static pressure field was observed for H-grid than O-grid. - Due to inaccurate turbulence modeling or boundary layer resolution (y^+), the numerical results did not predict a strong secondary flow as was obtained in the experiment.
Britchford et al. [26] and mullick et al. [27]	S-shaped annular duct	- The curvature effect on the development of flow and pressure gradient was captured better by the Reynolds stress model than the k-ε turbulence model.

Authors	Type of Duct	Significant Findings
Whitelaw and Yu [28]	Diffusing S-shaped circular duct	- The separation was found in both cases of the thinner and thicker boundary layer, however, it was larger for the thinner case. - In the thinner case, contra-rotating vortices occupied almost half of the cross-section at the exit.
Majumdar et al. [29]	S-shaped diffusing duct	- The flow reversal was seen at the inflection plane. - Pressure recovery was observed lower than the straight duct due to flow separation. - The wall static pressure distribution along the convex-concave curvature was significantly higher than that of the concave-convex wall.

Influence of Wakes Coming Out from the Upstream Compressor stage

In a modern integrated turbofan engine, where even marginal performance enhancement due to improved compressor blade design has become difficult to achieve, potential gains achieved by an improved duct design offer much larger potential returns. During the earlier phase of aero-engine design, the designer did not have proper knowledge of the blade designing as well as its effects on duct flow hence, the designer was compelled to select a relatively conservative duct design. Here conservative design means that the duct is designed without the occurrence of flow separation and the design space limitation does not matter. Due to the growing popularity of integrated design in the past few decades, it becomes necessary to understand that how the compressor representative inlet flow conditions affect the limits of the duct design space and how the integrated design raises the confidence to select a more aggressive design. A few researchers devoted their work in this area and found that with the upstream compressor stage, the tendency of flow to be separated along the inner wall had significantly reduced. The wakes induced by the compressor blades reenergize the inner wall boundary layer and eventually reduced the adverse pressure gradient.

To compare total pressure losses caused by the wakes induced from the upstream compressor, axisymmetric and non-axisymmetric S-shaped duct were designed. Literature suggests that in spite of a reduction in pressure gradient along the inner wall, the continuous generation of counter-rotating vortices enhanced duct losses. Although non-axisymmetric duct profiling suppressed these wakes up to a certain limit, it could not be succeeded to remove the loss completely. The non-axisymmetric approach also motivated to design an aggressive duct with higher $\Delta R/L$ than the conventional design without increasing the loss penalty. Table 2 shows a summary of the existing literature related to the effect of the wakes emanated by the upstream compressor on the flow development in the S-shaped duct.

Table 2. Summary of influence of wakes coming out from upstream compressor stage

Authors	Type of Investigations	Significance Findings
Britchford et al. [30]	With complete compressor stage	- The tendency of flow being separated along the inner wall was reduced. - Pressure gradient along the inner wall for without compressor stage was found from -0.35 to +0.25 whereas, with the compressor stage, it was from -0.3 to 0.20. - Wakes reenergized the boundary layer along the inner wall. - The shape parameter was significantly reduced along the inner wall from 1.65 for without compressor to 1.47 with the compressor stage.
Bailey [4]	With complete compressor stage	- Due to the rapid mixing of slower and faster fluid, the flow was less prone to separation. - Rapid mixing increased overall total pressure loss.
Karakasis et al. [31]	With complete compressor stage	- In the case of the axisymmetric duct, the presence of the compressor stage increased the duct pressure loss by 54%. - The maximum losses occurred along the inner wall. - Non-axisymmetric duct wall profiling suppressed the strut-hub corner separation and increased duct losses only by 28%. - Non-axisymmetric wall profiling allowed designing the duct with higher $\Delta R/L$ without increasing the losses.

Authors	Type of Investigations	Significance Findings
Britchford et al. [32]	Integrated outlet guide vanes	- Most of the pressure loss and static pressure rise happened between Outlet guide vane (OGV) exit and duct inlet.
Walker et al. [33]	With complete compressor stage or integrated outlet guide vanes in the duct	- In both cases, flow development was almost identical and the performance of the downstream component was unaffected. - Both designs offered wakes and these wakes found throughout the duct passage. - Due to the shorter length, the modified pressure field was slightly different in the second case.

The summary of the review suggests that the compressor stage effect on inlet flow conditions affects the limits of the duct design space. Therefore, one should go with the integrated design of the duct with a compressor stage to select a more aggressive duct. However, related to the naturally developed inlet conditions the flow was less likely to separate due to the rapid mixing of the slower and fast-moving fluid. This rapid mixing increases the overall pressure loss.

Influence of Swirl

In modern integrated engines, especially in military engines where the engine is installed in the aircraft fuselage, the design of each and every component significantly affects the overall performance of the engine. In such engines, components are designed in a robust and complex manner. For example, the interconnecting annular duct is designed as an S-shaped form which consists of a series of curvatures, which further adds the complication in pressure recovery and flow uniformity. Generally, the more complicated the S-shaped duct is, the more significant is its contribution to total pressure losses and to the quality of flow, hence, it is always appreciated to study all characteristic aspects of flow into the subsonic duct of modern integrated engines. In most cases, generally, the flow enters to S-duct maybe with swirl either because of insufficient straightening or for deliberately matching to the requirement of the downstream component. It is evident that flow separation is attributed to the strong curvature of the S-shaped duct, however, it depends on many factors, not on a particular one. Most researchers have proven that the flow separation augmented bulk vortices which ultimately results in the rotational flow (swirl) at the downstream compressor and eventually increased pressure losses.

Sometimes, the presence of a swirling flow also altered the turbulence structure of flow into the duct and promoted the premature flow separation as well. It is well known that turbulence mixing over the convex curvature is suppressed whereas enhanced over the concave curvature, hence the inner wall which is made of convex followed by concave curvature is vulnerable to be separated due to the higher positive pressure gradient. Flow separation can be avoided or delayed by keeping the conservation of tangential momentum along a continuously varying radius of the duct. Therefore, as the duct radius decreases, swirl velocity at that point will have to be increased. In this manner, changes in swirl velocity along the inner wall will be highest and eventually lower the positive pressure gradient which results in delay flow separation. Authors summarised that after allowing the swirl, pressure differences across the duct were get modified and an interesting fact was that most significant modifications happened along the inner wall where flow separation was more pronounced. In contrast, due to a high level of turbulence, higher total pressure loss was attributed compare to without swirl and upstream compressor stage cases. Table 3 depicts the previous work regarding the influence of the swirl on the duct performance.

The incorporation of an optimum swirl enhances the velocity flow throughout the duct passage. This tends the flow towards the turbulent flow region which reduces the boundary layer thickness and secondary flow effects with the enhancement of the pressure loss.

Table 3. Summary of influence of swirl on duct performance

Authors	Swirl Providing Method	Remark
Anand et al. [34]	Through swirler with $\pm 20°$ (clock and anti-clockwise)	- Despite the increase in the loss coefficient, swirl increased static pressure recovery by 40%. - A pair of unequal intensity was found throughout the passage. - At the outlet, the more uniform flow was observed for the clockwise case in comparison to the anti-clockwise case.
Weng and Guo [35]		- The excessive swirl at the duct exit was controlled using an automatically adjustable blade (AAB) method. - The AAB method offers an effective swirl control approach at the expense of an acceptable total pressure loss.

Authors	Swirl Providing Method	Remark
Mohan et al. [36]	Through tangential velocity component in numerical simulation of annular wall diffuser	- Increasing the inlet swirl at a particular level improved the pressure recovery but thereafter it decreased as swirl increased.
Lohmann et al.[37]	Through swirler with 0-48°	- Increased swirl angle and cant angle promoted high distortion in the velocity profile at the diffuser exit. - The presence of the swirling flow changed the turbulence structure of the flow and promoted premature separation.
Bailey and Carrotte [38]	Using the compressor stage and by removing stator blades from the stage	- Nature of the streamwise velocity profiles was similar, however, swirl modified the streamwise pressure gradient. - Without swirl, the pressure gradient along the inner wall rose from -0.35 to +0.25 whereas, with swirl, it was from -0.30 to -0.12. - In the swirl flow case, the pressure gradient was significantly reduced along the inner wall. - In the swirl case, the higher velocity was observed along the outer wall due to tip leakage of the rotor. - Without the swirl, pressure loss was obtained 0.019 whereas, it was 0.039 and 0.043 for the compressor stage and after removing the stator blades respectively.

Influence of Struts

In a multi-spool turbofan engine, the diameter of each successive spool is reduced to increase the air density through the compressor system. Consequently, the intermediate annular S-shaped duct is used to connect low and high-pressure compressors. The curvature of this duct persuades the pressure gradient in the radial and stream-wise direction of the flow. It is well known that along the inner wall, static pressure first falls and then rises. However, along with the outer wall, its opposite happened. Figure 6 shows the diffusion phenomena of the flow along the duct wall. It can be seen that due to curvature, adjacent to the inner wall, the flow is subjected first to a small acceleration thereafter it has to encounter a large deceleration and finally under a small acceleration, it continues. On the contrary, close to the outer wall, flow follows opposite trends. In modern engines, it might be necessary to facilitate the intermediate duct with the provision of the load-carrying struts to provide extra support to the bearings against the forces coming from the upstream and downstream components. Struts can also be used to provide a path for essential engine services such as cooling air and lubricating oil. Despite a lot of advantages, struts create the blockages to the flow within the S-shaped duct which ultimately causes additional pressure gradient to the duct wall and modify the flow field of the duct as well.

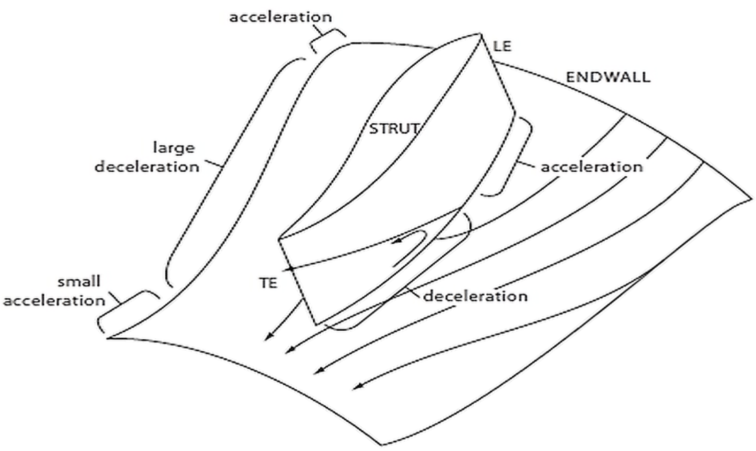


Figure 6. Schematic of duct pressure fields [39]

From Figure 6, it can be understood that as the flow approaches to the strut, it becomes stagnant and after then it starts to move with a higher velocity and finally near the trailing edge of the strut, it is being decelerated. Consequently, deceleration caused by the trailing part of the strut usually takes place in that region of the outer wall where curvature causes large acceleration. Therefore, the boundary layer at the outer wall is unlikely to be separated. However, in the case of the inner wall, its opposite happens. Here, deceleration caused by the strut takes place in that region where large deceleration of the flow has already been caused by the curvature.

In addition to the wall curvature, struts also add complexity to the flow due to physical blockages and eventually create a convergent-divergent passage. This passage further generates a strong pressure gradient throughout the duct passage. Struts often are mounted as a cascade within the duct which creates an aerodynamical interaction among them. In the cascade, struts are kept in such a way that the suction side of one strut falls adjacent to the pressure side of another strut. In addition to the duct curvature, this arrangement of the struts creates another pressure field. Also, struts may induce vortices between the pressure fields created by the struts and the combination of such pressure field and vortices leads to premature flow separation. This premature flow separation causes extra losses as well as asymmetric distortions which can be spread out to the downstream components. Eventually, the system has to face severe losses in terms of efficiency as well as flow uniformity. Therefore, since the beginning, researchers have always been concentrating on this area of research. In the series of investigations, the secondary flow characteristics in the annular duct with strut were experimentally and numerically reported by many researchers and represented in Table 4.

Table 4. Summary of influence of strut on duct performance.

Authors	Number of struts	Remark
Naylor et al. [39]		- The presence of the struts caused extra pressure gradient on the duct wall. - Along the inner wall, the combined deceleration caused by the strut and the curvature promoted the flow separation. - The non-axisymmetric profile minimized the pressure gradient imposed by the strut. - Such a profile allowed the strutted duct with 34% higher ΔR / L with only an 11% increase in pressure loss. - Pressure loss was highly sensitive to the inner wall rather than the outer wall.
Bu et al. [40]	Eight struts (Four thick and four thin)	- The strong secondary flow was observed when fluid flows under the curvature, cross-sectional area variation, and the presence of struts. - Vortex pairs are formed in the vicinity of the hub and shroud due to interference of the struts. - The hub-side and shroud-side vortices exhibit different characteristics due to the influence of different streamwise pressure gradients.
Norris et al. [41]	Twenty sex struts	- The separation bubbles were found in both cases (with struts and without struts), however, strutted duct had a large size of bubbles. - Due to the larger size of bubbles, pressure loss was high in the strutted duct. - The static pressure rise coefficient decreased by 28.5% in the strutted duct.
Bailey et al. [42]	One strut	- Due to blockage, velocity profiles got modified. At the upstream of the strut, the boundary layer became thicker with a small core region. - The wakes were found throughout the duct passage. - The influence of a single strut was very small on the duct performance. The loss coefficient increased from 0.040 to 0.042 for without compressor case whereas it increased from 0.035 to 0.038 with compressor case.
Milanovic et al. [43] Baloni et al. [44]	One, six and eight strut	- Computational fluid dynamics showed the capability to capture the flow physics of the duct accurately under the strutted environment. - Similar to the experimental study, numerical simulation also captured wakes near to the trailing edge of the strut. - Increasing the Mach number and shortening duct length result in lower pressure coefficients. - The magnitude of pressure loss was depended on the number of struts.

Authors	Swirl Providing Method	Remark
Ross et al. [45]	Two different struts (one with t/c=13%, another with t/c=17%)	- Loss in the production-like duct is 30-40% higher than in the baseline duct. - The duct with high t/c depicted higher pressure loss for all cases. - Higher surface roughness enhanced the pressure deficits in the secondary flow and wakes regions.

Influence of Inlet Conditions and Geometrical Parameters

An intermediate compressor duct installed in a modern integrated turbofan engine has to deal with different flow conditions like take-off, cruise, and approach. Hence, the intermediate duct has to fulfill the flow demands of the downstream compressor for better performance according to the conditions. Since last successive decades, modern engines have been designing comparatively with a lesser number of stages and higher pressure ratio per stage so they have to tackle highly complicated maneuvers and at the same time, need to be very sensitive according to flow conditions. To clinch this dependency, a designer has to devote sustainable efforts to ensure duct-compressor compatibility along with the optimum integration of the engine. Hence, from the perspective of the S-shaped duct, the field of research fraternity is evenly divided into two groups. The first group of researchers believes in the direct study of the influence of inlet flow conditions on the performance of duct which is mainly caused by the different ambient conditions whereas other groups of researchers have studied the performance of S-duct according to parametric changes while keeping same ambient conditions. Both categories are discussed as follows:

Influence of Inlet Conditions

Fluid passing through the compressor intermediate duct is slightly decelerated and at the same time, it is subjected by the series of the curved bends. The main challenge with the design of the curved bend is to avoid flow separation. Hence, a good correlation must exist between flow regimes and optimum performance. The stability margin of the downstream compressor is the main important factor which must be taken into account. The angle of attack, skewed velocity profiles and angle of turn strongly promote uneven pressure distribution which ultimately causes secondary flow structures. Furthermore, the strength of the secondary flow structures strongly depends on the discussed inlet conditions. Due to secondary flow structures, outlet velocity and pressure profiles completely get modified and can lead to stall the downstream compressor. Table 5 shows a summary of the existing literature which described the effect of the inlet conditions on duct performance.

Table 5. Summary of influence of inlet conditions on duct performance

Authors	Inlet condition type	Remark
Waitman et al. [46]	Thin boundary layer to fully developed flow	- Provision of ample systematic and detailed data that were based on the effect of the inlet conditions of the curved duct to allow a more thorough study of the underlying problems of stall and energy redistributions in adverse pressure gradient flows and to develop improved correlations. - The static pressure recovery of the curved duct is a strong function of the inlet boundary layer (IBL) conditions and turbulence intensity. - The static pressure recovery reduced as the boundary layer had thickened.
Whitelaw et al. [47]	Three asymmetric inlet conditions at a fixed Reynolds number. (1) Thick bounder layer (2) Zero angle of attack (3) Low angle of attack	- The first and third cases contained large contra-rotating vortices whereas these were absent in the second case. - These vortices spoiled the duct outlet uniformity. - Turbulence quantities and pressure recovery for all three cases were higher than the corresponding cases of symmetric conditions.

Authors	Inlet condition type	Remark
Sonoda et al. [48]	Thin and thick inlet boundary layer (IBL)	- These two IBL conditions showed large dissimilarities in the flow patterns at the duct outlet. - The aerodynamic sensitivity of the S-shaped duct to the IBL thickness is very high, hence, total pressure losses significantly increased as IBL thickness was increased. - The strength of the secondary flow structures was strongly dependent on the imposed IBL conditions.
Patel et al. [49]	Skewed and uniform velocity profile at the inlet	- Static pressure recovery reduced with increases in skewness, and strong secondary flows were observed throughout the length of the duct with a highly complex flow pattern. - The performance of the duct deteriorates with the increase in the skewness of the inlet velocity profile.
Saha et al. [50]	Angle of attack (0-30°)	- The total pressure loss coefficient and distortion coefficient show a reasonably high value of 23.94% and 0.7835 respectively at a 30° angle of attack. - The static pressure recovery coefficient at the exit plane of the duct reduces (for 0.801 to 0.508) with the increase of angle of attack.
Singh et al. [51]	Reynolds number	- Reynolds number had no significant effect on the value of CL for Reynolds numbers higher than 10^5. - Pressure recovery decreased as the Reynold number increased. - The distortion coefficient was found lower at a low Reynold number.
Gartner et al. [52]	Varying honeycomb height and Inlet Mach number (0.2,0.44 and 0.58)	- The incoming boundary layer was manipulated for modifying the flow and pressure fields without penalty of pressure loss. - Manipulation of the velocity deficit using a 2D honeycomb modified the secondary flow structures and corrected the flow asymmetry at the AIP.
Brehm et al. [53]	Mach number and Reynolds numbers	- At high Mach and Reynolds number, radial, and circumferential distortion intensity at the outlet shown a gradual increment.
Gao et al. [54]	Pre swirl and Mach number	- The pre-swirl condition strongly influenced the flow near the strut. - As the swirl angle increased, the backflow region enlarged as well as vortices were pushed inward. - As the Mach number increased, pressure recovery and exit flow uniformity deteriorated.

Influence of Geometrical Parameters

Performance of S-shaped ducts strongly depends on the inflow conditions and geometrical parameters such as AR, radial offset (angle of the turn), aspect ratio (length to diameter ratio) and length of the duct [55–57]. Hence, a deep study of these parameters has much importance as much as others. In the continuous series of research, Duenas et al. [58] presented a work in which the effect of length on duct performance had been investigated. Initially, a duct with a specified length was designed to understand the flow physics using CFD. Afterward, two different ducts with 74% and 64% length respectively were designed and a comparison of the performance characteristics for different Reynolds numbers was established. The authors concluded that during the fully attached flow, the shape of the exit loss profile for both walls did not depend on duct length. Furthermore, net loss on the wall was a feeble function of length and Reynolds number; however, the inner wall encountered higher net loss than the outer wall. During the separated flow, for a particular duct as Reynolds number increased, the magnitude of C_p into the region of peak suction for the inner wall did not change whereas it showed systematic changes for the outer-wall.

Sonoda et al. [59] investigated experimentally the effect of a downstream passage kept at the exit of an S-shaped duct on the flow and also validated the loyalty of in-house developed 3-D Navier –stokes code. For this work, one straight and another curved annular passage was used. They found that during the curved passage, total pressure loss adjacent to the inner wall increased compared to the straight passage and the main reason for this loss was the presence of the horseshoe vortex near to the inner wall. Moreover, numerical prediction also confirmed the formation of this horseshoe vortex which showed that experimental and numerical results were in good agreement. Pressure recovery of a duct and magnitude of cross-flow velocity also depends on aspect ratio and angle of turn. The magnitude of the cross-flow velocity greatly increased with an increase in the angle of turn. On the contrary, as the aspect ratio increased, flow uniformity and pressure recovery increased up to a certain angle of turn due to the mitigation of the corner effect [60], [61].

Lee et al. [62] conducted experiments on the conical duct that connects the exit of the compressor to the inlet of the combustor. This duct was smoothly transitioned into an S-shaped duct while maintaining the constant cross-sectional area. After a flow investigation using the SST turbulence model, the authors performed the sensitivity analysis of turbulence intensity and Reynolds number. It was observed that the stream-wise velocity profile was flattened in the divergent section afterward it had been pushed towards the outer wall. However, for the parent duct, friction coefficients were got lowest, static pressure recovery was highest and total pressure loss was lowest among its all variants. One more fact was observed that with increased in Reynolds number or decreased in turbulence intensity, uniformity in stream-wise velocity diminished. Some other researchers [63]–[68] had also shown their interest to investigate the influence of the change in inlet shape such as elliptic, semi-circular, oval, rectangular and square on the performance of the S-duct.

ADVANCE TECHNIQUES FOR PERFORMANCE ENHANCEMENT OF DUCT

Shape Optimization and End Wall Contouring

The prime objective of an S-shaped intermediate duct is to supply high quality (maximum uniformity of flow) compressed air to the downstream component with minimum pressure loss. However, an aggressive S-shaped duct causes a higher pressure loss due to its curvature and finally distorts air quality. The performance of the downstream component mainly depends on the quality of supplied air and if the duct fails to supply the high-quality air, the performance of the downstream component is deteriorated. In addition to the curvature, it has been evident that the duct performance depends on many factors like the presence of struts (causes corner separation between the strut and the inner wall of the duct), inlet conditions (Mach number or Reynolds number), boundary layer thickness (thin or thick), geometrical shape and many more. Mitigation of these effects forces to design an optimum duct which ensures maximum air quality with minimum pressure loss. Hence, the optimization of the S-shaped duct cannot be sidelined.

Even though a number of researches have shown their interest to increase the performance of the duct by using various optimization techniques, this area of research, particularly for compressor S-duct is still needed to be more explored. Due to the lack of practical design rules and performance correlations for an annular S-shaped duct, CFD can be used as a design approach for performance evaluation with a combination of various optimization techniques to obtain better design. Literature reveals that the RSM together with the design of experiments (DOE) can be a very efficient and robust method for CFD-based optimization techniques. Also, to optimize the un-strutted annular S-duct mainly four non-dimensional parameters ΔR/L, h_{in}/L A_{out}/A_{in} and r_{in}/h_{in} are needed to be varied. The presence of strut adds one more parameter which is the thickness to chord ratio (t/c). It is concluded that a high loading strutted duct is subjected to a large strut-hub corner separation and non- axisymmetric end wall optimization has successfully defended the duct by controlling the secondary flow evolution. Table 6 provides concise information about the existing literature related to the shape optimization and end-wall contouring of the S-shaped duct.

Table 6. Summary of shape optimization and end-wall contouring

Authors	Technique	Remark
Ghisu et al. [69]	A modified version of the steepest descent algorithm	- The objective was to minimize pressure loss. - A significant reduction in total pressure losses of 12.5% was achieved through end-wall shape optimization.
Wallin et al. [70]	RSM with sequential quadratic programming (SQP) method	- A total pressure loss reduction of about 24% was achieved by optimization.
Donghai et al. [71]	DOE, artificial neural network, and genetic algorithm	- Reduction of total pressure loss within the S-shaped duct by using optimized axisymmetric and non-axisymmetric end wall profiling. - The optimal axisymmetric duct showed a decrease of 27.7% in the pressure loss than the datum duct. - The optimized non-axisymmetric end-wall contouring duct showed a reduction of the total pressure loss by 32.7%. - The possibility to suppress the separation and to reduce the loss in the fairly aggressive duct was high in the non-axisymmetric end-wall contouring.
Lu et al. [72]	DOE, RSM and genetic algorithm	- The optimum duct showed a reduction of 12% in ΔR/L and a 37% decrease in total pressure loss.

Authors	Technique	Remark
Gan et al. [73]	Modified k-ω SST turbulence model and a multi-objective genetic algorithm	- Distortion coefficient and pressure recovery were considered as an objective function. - The flow distortion coefficient was reduced by 16.3% and total pressure recovery was increased by 1.1% at the design condition.
Shan et al. [74]	Semi-inverse method	- To control wall pressure gradient distribution and the wall velocity distribution, a semi-inverse design method for the transition duct was proposed. - This method suppressed the maximum gradient as much as possible and avoid the separation. - This method found the applications in the field of isoenergetic inviscid incompressible flow, isoenergetic isentropic compressible flow, as well as for viscous compressible flow.
Sturzebecher et al. [75]	AutoOpti algorithm and End wall contouring	- The duct was shortened up by 42% length while keeping the loss as much as low. - Implementing non-axisymmetric end wall contouring on hub and shroud wall, a 19% length reduction of the baseline duct could be further possible along with a 2% reduction in loss.
Sharma and Baloni [76]	Teaching learning based optimization algorithm	- The proposed design showed 28.80%, 36.67% reduction in pressure loss and non-uniformity respectively despite 14.74% length reduction.

Flow Controlling Devices

In multi-spool turbo-fan engines, the speed of the low-pressure compressor is limited by the tip speed of the fan. For achieving higher work per stage, maintaining reasonable efficiency and keeping tip speed higher, low and high-pressure compressors need to design with higher radial offset. Flow from the low-pressure compressor must be turned to the radially inward without occurrences of flow separation through the intermediate duct. Hence, the length of the intermediate duct is restricted by the flow separation within the duct. It is evident that flow separation might be presented within the duct due to secondary flow and can encompass throughout the duct with the evolution of the vortices eventually it becomes the cause of inferior flow quality. Hence, it is essential to control the flow separation as flow control permits the designer to design a more aggressive intermediate duct with a larger radial offset [77]. Flow separation devices; depending on the type of flow control method, can be devised as active or passive. For example, submerged vortex generators (VGs) (wheeler doublet and wish-bone types), span-wise cylinders and large eddy breakup are passive devices, whereas vortex generators jet (VGJs) and either injection of the higher energy fluid or removal of lower energy fluid to energize the boundary layer from the inner wall are categorized into active methods[78–81].

Some notable advantages are found in active methods over passive methods. First, the actuation of the active methods is generally effortless and shows a very quick response in conditions where rapid deployment is necessary to control the separation. Second, the drag penalty of active devices is almost negligible when they are not being used [82]. During most of the cases of multistage turbofan engines, air flows at a significantly higher temperature into intermediate duct and extraction of this air from the flow is always a challenging task. Regardless of that, Walker et al. [80] reported a numerical investigation in which a novel bleed system was practised to re-energize the low momentum boundary layer which is usually found on the inner wall of the S-shaped intermediate duct. However, it was a tedious task to find the exact location of the bleed system at which fundamental mechanisms of the proposed bleed system do not get altered. Hence, CFD played again his vital role to find the best location for the bleed. This type of special arrangement combined two mechanisms: The first one was that during acceleration through the bleed system (duct), fluid got stream-wise momentum. This momentum was transported across to the diffusing mainstream flow where it re-energized to the mainstream flow and got it enabled to remain attached to the inner wall. The second was that the bleed system created a radial pressure gradient which motivated transport of higher momentum fluid into the low momentum fluid. The result was obtained for a 5% bleed flow rate of the entering flow and similar velocity profiles at rotor exit and inlet guide vanes compared to the non-bleed case were observed, however, after the outlet guide vanes (OGVs) some deviations in profiles were observed due to bleed system slot. Along with fully attached flow, this arrangement had given a total reduction of 30-40% in length compared to conventional design.

Later on, Siggeirsson et al. [83] compared the flow field at different operating conditions that were controlled by extracting a fraction of mass flow rate through a bleed pipe by putting just upstream of the intermediate S-duct. The extraction of the mass flow rate could avail the best performance of the downstream high-pressure compressor. Moreover, the amount of extraction depends on the performance requirement of the high-pressure compressor. On the other hand, extracting mass flow rate might result in a higher risk of flow separation and eventually, intermediate duct could show its ever worst performance. The authors concluded that the predicted pressure coefficient for the lower bleed case was almost equal to the experimental case. Unlike the lower bleed extraction, CFD computed outer wall behavior quite well, but some discrepancies had been shown over the inner wall. Similarly, during lower bleed fraction, boundary conditions and geometry depicted more stability than a higher bleed fraction. Siggeirsson et al. [84] performed measurements on the intermediate S-shaped duct (carrying bleed pipe) through experiment and numerical simulation. They also presented a comparison between different turbulence techniques (RANS, unsteady-RANS (URANS) and Delayed-Detached Eddy-Simulation (DDES)). The authors found large differences between the experiment and simulation results of the bleed pipe because of slight dissimilarity between actual and modeled bleed geometry. The RANS and URANS calculations were simple, less expensive and taken less computational time. The authors reported that URANS results showed more stability than DDES. The DDES results showed large instabilities and were capable in producing similar results as the RANS and URANS simulations,

To control the evolution of the secondary flows and flow separation by incorporating the passive means, a number of researchers had devoted their research. Most of the researchers worked on submerged VGs which indicate that the height of the vortex generator is shorter than the maximum height of the generated boundary layer within the S-duct. VGs normally placed in arrays which either may be counter-rotating or co-rotating. Here, Counter-rotating means that VGs should be set in pairs at the incidence of equal but opposite in sign whereas co-rotating means that VGs should be set at the same nominal incidence and equally spaced. Moreover, performance enhancement of an S-shaped due to the incorporation of VGs is calculated by the static pressure recovery coefficient, the total pressure loss coefficient, and the distortion coefficient. Table 7 sum up the research carried out on various flow controlling devices and this table is categorized on the basis of the types of flow control devices. The summary of available research on various aspects such as the arrangement of VGs devices, how many VGs are being used, the effect of the combination of active and passive VGs, VGs parametric studies, and VGs locations have been presented here.

Integrated Concept

In the conventional design method, the S-shaped duct was subjected to a large flow separation because a large radius change was obtained in a shorter axial length in order to meet design space limitation. It has been discussed that the optimization demands a reduction in space and weight without significant change in duct's aerodynamic performances. The previous discussion revealed that these objectives had been achieved by using some popular optimization techniques. Also, it had been found that the presence of OGVs at the upstream of the S-shaped duct extended the duct loading which ultimately evokes flow separation along the inner wall. To overcome this problem, researchers introduced an approach in which OGVs of the existing upstream compressor had been placed inside the first bend of the S-shaped duct. Basically, the main purpose behind this approach was to impose a force in the radial direction of the flow such that the OGVs were leaned tangentially to the flow. This radial force helped to turn the flow and reduced the static pressure rise along the inner wall significantly. Additionally, this approach also helped to reduce the overall axial length of the system.

Literature reveals that in such design, the rotor outlet pressure field must be maintained in such a way that rotor performance is unaffected. Moreover, the design of the S-shaped compressor duct is a tedious task because a slight disturbance to the flow will directly affect the performance of the downstream high-pressure compressor. The merits and demerits of the strut within the S-shaped duct have already been discussed and it is preferred to use symmetrical airfoils as struts to minimize the interference. However, flow is still disturbed and finally, the performance of the duct is deteriorated.

Recently, to meet the design space limitations, researchers have proposed a "turning or lifting strut" concept. The fundamental idea behind this concept is to serve as a combined purpose of the struts as well as OGVs of the low-pressure compressor. In such a concept, OGVs are omitted from the low-pressure compressor which results in a potential reduction of length and weight of the turbofan engine. The main function of turning strut is to redirect and straighten the flow to the desired swirl angle. Based on the particular application, either it can be as moderate or fully turning strut. It was seen that at the aerodynamic design point, fully turning strut had shown lower pressure loss compared to moderate turning strut [85]. However, for the practical execution of such a concept, it has to satisfy some basic requirements. First, the pressure loss of the proposed design concept must be equal or less than the conventional design. Second, swirl angle, circumferential and radial distortion within the duct must be of the acceptable range. Third, the design should not cause extra flow separation.

In a multistage compressor system, efficiency can be improved by allowing the swirl into the rear stages. However, this swirl has to completely be removed within the intermediate duct before entered to the downstream compressor. Otherwise, it will significantly spoil the performance of the compressor. A uniform flow condition must be supplied for its better performance. A summary of the previous work carried out related to the integrated design of the S-shaped duct is presented in Table 8.

Table 7. Summary of the flow control devices in the S-shaped duct

	Vortex generators Types	Arrangement	Locations	S-duct	Conclusion
Reichert et al.[86]	Tapered-fin	Total eight arrangement: one bare, five sets as narrow spaced (tapered fins start from a set of two and the last was set of ten with increasing in even order) and two wide-spaced (one with 4 and another with 6 tapered fins) combinations	Along the inner wall immediately upstream to the flow separation	Circular	Best total pressure recovery obtained by two narrow spaced VGs combination whereas; distortion coefficient was improved with wide-spaced combinations
Reichert et al.[87]	Low profile Wishbone	Height (h/D_1), Location (s/D_1) and spacing (l/λ) were parameters. One parameter out of three was varied while the others were constant. Total eight arrangements found.	Along the inner wall immediately upstream to the flow separation	Circular	Case-1: Maximum pressure recovery occurred for VGs which had a height equal to boundary layer thickness. The distortion coefficient improved with increasing VGs height. Case II- Effectiveness of VGs reduced when the axial location of VGs is closed to the separation point. Case III- Circumferential extent of separation region was reduced as VGs spacing reduced.
Paul et al.[88]	Submerged Trapezoidal	Two VGs with different geometrical configurations and placed co and counter-rotating manners.	Either at both side of walls or at the top and bottom wall on the inflection plane	Y-shaped rectangular	- VGs at sidewalls reduced the flow separation. - VGs at top and bottom wall improved flow uniformity. - VGs with larger height, co-rotating sequence and at both side walls gave the best performance.
Paul et al.,[89]	Submerged Trapezoidal	With three, four and five pair of combinations	At the top and bottom surfaces and placed either before or after and at the inflection plane	Rectangular	- Three pairs of combinations at both sidewalls placed after the I nflection plane gave the best performance in terms of pressure recovery, pressure loss and flow-uniformity.
Anabtawi et al.[90]	Flat plate	At throat's flat and curved surface with different configurations	Both sidewalls	Circular	- Reduced distortion up to 11% in some cases not in all.
Sullerey et al.[91]	-fences -Wishbone and tapered fin VGs	With different configurations	At the top and bottom wall	Two Rectangular ducts with different curvature ratios (4 and 6)	- Fences of a higher radius ratio showed the best performance in the duct. - Tapered fin VGs of lower radius ratio showed the best performance.

	Vortex generators Types	Arrangement	Locations	S-duct	Conclusion
Sullerey and Pradeep [92]	-Tapered fin VGs -Vortex generator jets	Three sets of jets under uniform and distorted inlet conditions	Two locations. (1) At the maximum secondary flow strength region (2) Just after the inlet.	Rectangular	- Performance improvement by using VGJs was higher than the tapered fin VGs. - 25% reduction in pressure loss by the Combination of the VGJ and tapered VGs. - VGJ alone reduced by 30%.
Pradeep and Sullerey [93]	Vortex generator jets	Three sets of jets with -Only VGJs -Both steady VGJs -VGJs with feedback control	Two locations. (1) At the maximum secondary flow strength region (2) Just after the inlet.	-Rectangular -Rectangular to circular	- Total pressure loss reduced by 14% in the transition duct (Rectangular to circular). - Distortion coefficient reduced by 20% in the circular duct. - Feedback arrangement further improved performance.
Paul et al.[94]	Fishtail submerged	-With sharp 90° and 45° chamfered edges -3*3 VGs -2*2 VGs	-Top and bottom walls of inflection plane -Top and bottom walls of the inlet plane	Rectangular	- Locations have more influence rather than the number of VGs. - 45° chamfered with 3*3 at both walls of the inflection plane gave better performance.
Ahmad et al.[95]	Trapezoidal submerged	Variations obtained -The angle of incidence (14°&18°) -VGs spacing -Counter and co-rotating arrangement -Stream-wise position	Different locations at upstream of the inflection point.	Rectangular	- Counter-rotating was most effective. - The flow was sensitive to the spacing (narrow case best). - Incidence angle has little effect (18° best). - The best position just upstream to separation.

Table 8. Summary of integrated concept in S-shaped duct

Authors	Integrated concept	Remark
Britchford et al. [32]	OGV located in the first bend of duct	- Upstream OGV row had been leaned tangentially in such a way that it helps to turn the flow within the first bend of the S-shaped duct. - The aerodynamic loading on the critical inner wall boundary layer was reduced.
Walker et al. [33]	OGV located in the first bend of duct	- The flow development was almost identical to conventional design besides some minor deviation. - The integrated design reduced the system length by 21%. - The total pressure loss coefficient over the OGVs is reduced by 20% using the integrated concept. However, the overall loss increased slightly by 3%.
Walker et al. [85]	Incorporated the turning strut	- Turning strut redirected and straightened the flow to the desired swirl angle. - Reduced the length of the compressor system by 25% of the duct length through the elimination of the last stator row was achieved. - Due to the turning strut, a strong vortices structure at the outlet was found.
Bergstedt et al. [96]	Incorporated the turning strut	- The compressor module was shortened by incorporating turning struts. - The length of the system was reduced by 20% by removing the compressor stator blade row.
Walker et al. [97]	Incorporated the turning strut	- Removed tangential momentum or 12.5° swirl from the flow within an S-shaped compressor transition duct. - Due to turning strut, the total pressure loss coefficient from the rotor exit to duct exit had increased only by 14% and any other unexpected losses were not found. - Turning strut could not remove swirl completely, hence, some sort of secondary flows were found at duct outlet.
Walker et al. [98]	Incorporated the engine suction splitter (EES)	- ESS showed a very small effect on flow development and due to wakes produced by ESS, flow did not move towards the separation. - The rapid mixing of wakes enhanced the duct loss by 12%.

CONCLUSIONS AND FUTURE PERSPECTIVES

This article reviews the aerodynamics of an S-shaped intermediate compressor duct for multistage high by-pass turbofan engines and their significance for the overall performance of engines as well. The inception of this article begins with the basic introduction of the intermediate compressor duct where the need for the duct and its applications in the various engine have been discussed. After this, mechanisms taken place within the S-shaped duct are discussed in detail. A special focus has been given to figure out the forces such as centrifugal forces and radial and stream-wise pressure gradient imposed by the duct curvature on the fluid and reaction taken by the fluid as well. When fluid flows through the S-shaped duct then it encounters by the centrifugal force as well as radial and stream-wise pressure gradients. Under these forces, fluid behavior gets modified and reacts differently which can be clearly understood by the C_p Variations along the walls. However, this variation is again influenced by the curvature ratio. The performance of the duct can be evaluated by using some specific parameters; these have been summarized over here. After this, the discussion steps forward with the emphasis of the fundamental aerodynamic of the intermediate compressor duct. Especially stress has been given onto the wall curvature influences, swirl wakes emanated by the upstream compressor stages, rotating vortices, the formation of secondary flow led by adverse pressure gradients. Even though wakes emanated by the upstream compressor suppress the flow separation, total pressure loss significantly increased. Moreover, flow quality at the duct exit is also degraded by the rotating vortices.

The aerodynamic behavior of the duct with the insertion of struts is also explained very clearly and difficulties created for the designer are highlighted. Strut causes blockages that promote additional diffusion on the inner wall. This diffusion is attributed to the strong flow separation on the inner wall and deteriorates the duct performance as well. After this, the influence of inlet flow conditions caused by different flight modes along with geometrical parametric studies has been focused. Due to inlet flow conditions, Mach number, Reynolds number, and boundary layer development within the S-duct get affected which eventually degrade its performance. Higher Mach number helps to get increased in size of counter-rotating vortices. However, the Reynolds number has not so much influence. Using CFD, parametric studies had been performed which are also discussed here. Continuous variations in the AR, curvature ratio, length of the S-shaped duct and angle of turn show that the overall performance of the duct and downstream compressor as well and the flow quality at duct outlet is the strong function of these parameters. The next topic is dedicated to the discussion of some advanced techniques to control the flow separation, shorten the design space and overall performance improvement of the engines.

To shorten the axial length of the intermediate duct, a large radial offset between the low and high pressure has to maintain. Unlike the straight annular duct, the fluid physics of an S-shaped duct mainly depends on the flow area distribution and shape of the end walls formed by the curvature. Therefore, optimization of the duct shape through the computational approach and parametric variations without increasing significant pressure loss is discussed over here and supremacy of the axisymmetric and non-axisymmetric end wall profiling approach to shorten axial duct length even into the strutted duct has also been illuminated. Subsequently, applications of the active and passive flow control methods in order to make possible to withstand the boundary layers against the strong curvature and large adverse pressure gradients are also discussed. It is also suggested that even though the practical application of these methods can be beneficiated in order to delay the flow separation, they might increase to pressure losses. Hence, care must be taken while using them. CFD plays a vital role to detect the optimum locations of these devices such as VGs height, blowing rate, and locations, etc. This paper ends up with a pioneering idea of the integrated concept that can be worthwhile to shorten the axial length of the aero-engine. Furthermore, the idea of turning strut can be an interesting approach for high by-pass engines.

All discussed keynote points talk about to get the better overall performance of an aero engine. Increasing demands of the high by-pass engine and to decrease the weight of the engine which basically helps to save fuel, an intermediate compressor duct became a key component for an aero-engine designer. However, a slight increase in the performance of an intermediate duct is not as simple as it is considered. Therefore for a designer, it is important to point out the physical relevance of the duct concerning the high performance of turbo-fan engines and they must also be considered as the main component.

REFERENCES

[1] H. I. H. Saravanamuttoo, G. F. C. Rogers, and H. Cohen, *Gas turbine theory (5th Ed.)*. Pearson Education, Ltd., 2001.

[2] J. Kurzke, "Fundamental differences between conventional and geared turbofans," in *Proceedings of ASME Turbo Expo 2009: Power for Land, Sea and Air*, Jan. 2009, pp. 145–153, doi: 10.1115/GT2009-59745.

[3] R. G. Norris, Glyn Dominy, "Flow through S-shaped annular inter-turbine diffusers," University of Durham, 1998.

[4] D. W. Bailey, "The aerodynamic performance of an annular S-shaped duct," Loughborough University, 1997.

[5] Y. T. Ng, S. C. Luo, T. T. Lim, and Q. W. Ho, "On the relation between centrifugal force and radial pressure gradient in flow inside curved and S-shaped ducts," *Physics of Fluids*, vol. 20, no. 5, pp. 055109–1--055109–15, May 2008, doi: 10.1063/1.2926759.

[6] P. Bansod and P. Bradshaw, "The flow in S-shaped ducts," *Aeronautical Quarterly*, vol. 23, no. 2, pp. 131–140, 1972, doi: 10.1017/s0001925900006004.

[7] K. Sudo, M. Sumida, and H. Hibara, "Experimental investigation on turbulent flow in a square-sectioned 90-degree bend," *Experiments in Fluids*, vol. 30, no. 3, pp. 246–252, 2001, doi: 10.1007/s003480000157.

[8] M. M. Enayet, M. M. Gibson, A. M. K. P. Taylor, and M. Yianneskis, "Laser-Doppler measurements of laminar and turbulent flow in a pipe bend," *International Journal of Heat and Fluid Flow*, 1982, doi: 10.1016/0142-727X(82)90024-8.

[9] H. Ito, "Pressure losses in smooth pipb Bends," *Journal of Basic Engineering*, vol. 82, no. 1, pp. 131–140, Mar. 1960, doi: 10.1115/1.3662501.

[10] W. R. Briley, R. C. Buggeln, and H. McDonald, "Computation of laminar and turbulent Flow in 90-degree square-duct and pipe bends using the Navier-Stokes equations," 1982.

[11] A. J Ward-Smith, *Pressure losses in ducted flows*. Butterworths, 1971.

[12] M. K. Gopaliya, M. Kumar, S. Kumar, and S. M. Gopaliya, "Analysis of performance characteristics of S-shaped diffuser with offset," *Aerospace Science and Technology*, vol. 11, no. 2, pp. 130–135, 2007.

[13] P. Aref, M. Ghoreyshi, A. Jirasek, and M. Satchell, "CFD validation and flow control of RAE-M2129 S-duct using cREATE-AV kestrel simulation tools," *Aerospace*, vol. 05, no. 31, pp. 1–23, Jan. 2018, doi: 10.2514/6.2018-1512.

[14] M. K. Gopaliya and K. K. Chaudhary, "CFD analysis of performance characteristics of Y-shaped diffuser with combined horizontal and vertical offsets," *Aerospace Science and Technology*, vol. 14, no. 5, pp. 338–347, Jul. 2010, doi: 10.1016/j.ast.2010.02.008.

[15] A. Vakili, J. Wu, P. Liver, and M. Bhat, "Measurements of compressible secondary flow in a circular S-duct," Jul. 1983, doi: 10.2514/6.1983-1739.

[16] A. Vakili, J. Wu, W. Hingst, and C. Towne, "Comparison of experimental and computational compressible flow in a S-duct," Jan. 1984, doi: 10.2514/6.1984-33.

[17] S. Wellborn, B. Reichert, and T. Okiishi, "An experimental investigation of the flow in a diffusing S-duct," in *Proceedings of 28th Joint Propulsion Conference and Exhibit*, Jul. 1992, pp. 1–12, doi: 10.2514/6.1992-3622.

[18] A. M. K. P. Taylor, J. H. Whitelaw, and M. Yianneskis, "Developing flow in S-shaped ducts I-square cross section duct," 1982.

[19] P. Taylor and J. H. Whitelaw, "Developing flow in S-shaped ducts II-circular cross-section duct," 1984.

[20] O. Abdellatif, "Experimental study of turbulent flow characteristics inside a rectangular S-shaped diffusing duct," in *Proceedings of 44th AIAA Aerospace Sciences Meeting and Exhibit*, Jan. 2006, pp. 1–10, doi: 10.2514/6.2006-1501.

[21] J. C. Vaccaro, Y. Elimelech, Y. Chen, O. Sahni, K. E. Jansen, and M. Amitay, "Experimental and numerical investigation on the flow field within a compact inlet duct," *International Journal of Heat and Fluid Flow*, vol. 44, no. 01, pp. 478–488, Dec. 2013, doi: 10.1016/j.ijheatfluidflow.2013.08.004.

[22] R. B. Anand, A. Chandraprabhu, X. J. A. Richards, and N. Hareshram, "Flow and performance characteristics of a Y-shaped diffusing duct using CFD," *International Journal of Aerodynamics*, vol. 1, no. 2, pp. 115–129, 2010, doi: 10.1504/IJAD.2010.037923.

[23] G. J. Harloff, C. . Smith, and B. J.E, "Three-dimensional compressible turbulent computations for a non-diffusing S-duct," 1992.

[24] C. F. Smith, J. E. Bruns, G. J. Harloff, and J. R. DeBonis, "Three-dimensional compressible turbulent computations for a diffusing S-duct," 1992.

[25] G. Harloff, B. Reichert, and S. Wellborn, "Navier-Stokes analysis and experimental data comparison of compressible flow in a diffusing S-duct," in *Proceedings of 10th Applied Aerodynamics Conference*, Jun. 1992, pp. 719–727, doi: 10.2514/6.1992-2699.

[26] K. M. Britchford, J. F. Carrotte, S. J. Stevens, and J. J. McGuirk, "The development of the mean flow and turbulence structure in an annular S-shaped duct," in *Proceedings of Turbo Expo: Power for Land, Sea, and Air*, Jun. 1994, p. V001T01A144, doi: 10.1115/94-GT-457.

[27] R. A. K. Mullick A N, "A numerical simulation of turbulent flow through a curved duct," *Mapana Journal of Sciences*, vol. 3, no. July, pp. 169–178, 2012.

[28] J. H. Whitelaw and S. C. M. Yu, "Turbulent flow characteristics in an S-shaped diffusing duct," *Flow Measurement and Instrumentation*, vol. 4, no. 3, pp. 171–179, 1993.

[29] B. Majumdar, S. N. Singh, and D. P. Agrawal, "Flow characteristics in S-shaped diffusing duct," *International Journal of Turbo and Jet Engines*, vol. 14, pp. 45–57, 1997.

[30] K. M. Britchford, A. P. Manners, J. J. McGuirk, and S. J. Stevens, "Measurement and prediction of low in annular S-shaped ducts," *Experimental Thermal and Fluid Science*, vol. 9, no. 2, pp. 197–205, Aug. 1994, doi: 10.1016/0894-1777(94)90112-0.

[31] M. K. Karakasis, E. M. J. Naylor, R. J. Miller, and H. P. Hodson, "The effect of an upstream compressor on a non-axisymmetric S-duct," in *Proceedings of the ASME Turbo Expo 2010: Power for Land, Sea, and Air.*, Oct. 2010, pp. 477–486, doi: 10.1115/GT2010-23404.

[32] K. M. Britchford, J. F. Carrotte, J. H. Kim, and P. M. Hield, "The effect of operating conditions on the aerodynamic performance of an integrated OGV and S-shaped duct," in *Proceedings of ASME Turbo Expo 2001: Power for Land, Sea, and Air*, Jun. 2001, pp. 1–12, doi: 10.1115/2001-GT-0347.

[33] A. D. Walker, A. G. Barker, J. F. Carrotte, J. J. Bolger, and M. J. Green, "Integrated Outlet Guide Vane Design for an Aggressive S-Shaped Compressor Transition Duct," *Journal of Turbomachinery.*, vol. 135, no. 1, p. 11035, 2013.

[34] R. B. Anand, S. N. Singh, and L. Rai, "Effect of swirl on the flow characteristics of S-shaped diffusing duct," *Indian Journal of Engineering and Materials Sciences*, vol. 15, no. 1, pp. 317–325, 2008, [Online]. Available: http://hdl.handle.net/123456789/1988.

[35] P. F. Weng and R. W. Guo, "New method of swirl control in a diffusing S-duct," *AIAA J.*, vol. 30, no. 7, pp. 1918–1919, Jul. 1992, doi: 10.2514/3.11156.

[36] R. Mohan, S. N. Singh, and D. P. Agrawal, "Optimum inlet swirl for annular diffuser performance using CFD," *Indian Journal of Engineering and Materials Sciences*, vol. 5, no. 1, pp. 15–21, 1998.

[37] R. P. Lohmann, S. J. Markowski, and E. T. Brookman, "Swirling flow through annular diffusers with conical walls," *Journal of Fluids Engineering*, vol. 101, no. 2, pp. 224–229, Jun. 1979, doi: 10.1115/1.3448939.

[38] D. W. Bailey and J. F. Carrotte, "The influence of inlet swirl on the slow within an annular S-shaped duct," in *Proceedings of ASME 1996 International Gas Turbine and Aeroengine Congress and Exhibition*, Jun. 1996, pp. 1–11, doi: 10.1115/96-GT-060.

[39] E. M. J. Naylor, C. O. Dueñas, R. J. Miller, and H. P. Hodson, "Optimization of non-axisymmetric endwalls in compressor S-shaped ducts," *Journal of Turbomachinery*, vol. 132, no. 1, p. 011011, 2010, doi: 10.1115/1.3103927.

[40] H. X. Bu, H. J. Tan, H. Chen, and X. M. He, "Investigation on secondary flow characteristics in a curved annular duct with struts," *Flow, Turbulence and Combustion*, vol. 97, no. 1, pp. 27–44, 2016.

[41] G. Norris, R. G. Dominy, and A. D. Smith, "Strut influences within a diffusing annular S-shaped duct," in *Proceedings of ASME 1998 International Gas Turbine and Aeroengine Congress and Exhibition*, 1998, p. V001T01A100-1--V001T01A100-9.

[42] D. W. Bailey, K. M. Britchford, J. F. Carrote, and S. J. Stevens, "Performance assessment of an annular S-shaped duct," *Journal of Turbomachinery*, vol. 119, no. 1, pp. 149–156, 1997, doi: 10.1115/1.2841003.

[43] I. M. Milanovic *et al.*, "RANS simulations for sensitivity snalysis of compressor transition duct," in *Proceeding of 50th AIAA/ASME/SAE/ASEE Joint Propulsion Conference*, Jul. 2014, pp. 1–9, doi: 10.2514/6.2014-3631.

[44] B. D. Baloni, K. V. Kumar, and S. A. Channiwala, "Study and numerical analysis of compressor transition duct," in *Proceedings of the 4th International Conference of Fluid Flow, Heat and Mass Transfer (FFHMT'17)*, Aug. 2017, no. 108, pp. 1–10, doi: 10.11159/ffhmt17.108.

[45] F. Wallin, M. H. Ross, M. Rusche, S. Morris, and S. Ray, "Investigation of loss impact from production-like features in a compressor duct under engine realistic conditions," in *Proceedings of the ASME Turbo Expo 2017: Turbomachinery Technical Conference and Exposition*, Jun. 2017, pp. 1–9, doi: 10.1115/GT2017-64274.

[46] B. A. Waitman, L. R. Reneau, and S. J. Kline, "Effects of inlet conditions on performance of two-dimensional subsonic diffusers," *Journal of Basic Engineering*, vol. 83, no. 3, p. 349, 1961, doi: 10.1115/1.3658962.

[47] J. H. Whitelaw and S. C. M. Yu, "Flow characteristics in an S-shaped diffusing duct with asymmetric inlet conditions," in *Proceedings of 11 th Australasian Fluid Mechanics Conference*, 1992, pp. 139–142.

[48] T. Sonoda, T. Arima, and M. Oana, "The effect of inlet boundary layer thickness on the flow within an annular s-shaped duct," *Journal of Turbomachinery*, vol. 121, no. 3, pp. 626–634, 1999, doi: 10.1115/1.2841361.

[49] T. Patel, S. N. Singh, and V. Seshadri, "Characteristics of Y-shaped rectangular diffusing duct at different inflow conditions," *Journal of Aircraft*, vol. 42, no. 1, pp. 113–120, Jan. 2005, doi: 10.2514/1.4690.

[50] K. Saha, S. N. Singh, and V. Seshadri, "Effect of yaw angle on the performance characteristics of twin intake duct," *Journal of Aircraft*, vol. 54, no. 3, pp. 1211–1216, 2017, doi: 10.2514/1.C033947.

[51] R. K. Singh, S. N. Singh, and V. Seshadri, "Performance and Flow Characteristics of Double-Offset Y-Shaped Aircraft Intake Ducts," *Journal of Aircraft*, vol. 45, no. 4, pp. 1230–1243, Jul. 2008, doi: 10.2514/1.34137.

[52] J. Gartner and M. Amitay, "Effect of boundary layer thickness on secondary structures in a short inlet curved duct," *International Journal of Heat and Fluid Flow*, vol. 50, pp. 467–478, 2014.

[53] S. Brehm, T. Kächele, and R. Niehuis, "CFD investigations on the influence of varying inflow conditions on the aerodynamics," in *Proceedings of 50th AIAA/ASME/SAE/ASEE Joint Propulsion Conference*, Jul. 2014, pp. 1–14, doi: 10.2514/6.2014-3595.

[54] L. Gao, X. Deng, X. Feng, and Z. Yang, "Effect of inlet conditions on compressor intermediate duct," *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, vol. 229, no. 6, pp. 1154–1168, 2015, doi: 10.1177/0954410014542624.

[55] B. Majumdar, S. N. Singh, and D. P. Agrawal, "Flow characteristics in a large area ratio curved diffuser," *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, vol. 210, no. 1, pp. 65–75, 1996.

[56] B. Majumdar, R. Mohan, S. N. Singh, and D. P. Agrawal, "Experimental Study of Flow in a High Aspect Ratio 90 Deg Curved Diffuser," *Journal of Fluids Engineering*, vol. 120, no. 1, p. 83, 1998, doi: 10.1115/1.2819668.

[57] J. Wendtt, "Flow through a rectangular-to-semiannular diffusing transition duct," *Journal of Propulsion and Power*, vol. 13, no. 2, pp. 312–317, 2018.

[58] C. Ortiz Dueñas, R. J. Miller, H. P. Hodson, and J. P. Longley, "Effect of length on compressor Inter-stage duct performance," in *Proceedings of GT2007 ASME Turbo Expo 2007: Power for Land, Sea and Air*, 2007, pp. 319–329, doi: 10.1115/GT2007-27752.

[59] T. Sonoda, T. Arima, and M. Oana, "The influence of downstream passage on the flow within an annular S-shaped duct," *Journal of Turbomachinery*, vol. 120, no. 4, pp. 714–722, Oct. 1998, doi: 10.1115/1.2841782.

[60] V. Gupta, R. Devpura, S. N. Singh, and V. Seshadri, "Effect of aspect ratio and curvature on characteristics of S-shaped diffusers," *Indian Journal of Engineering and Materials Sciences*, vol. 08, pp. 141–148, 2001.

[61] R. B. Anand, L. Rai, and S. N. Singh, "Effect of the turning angle on the flow and performance characteristics of long S-shaped circular diffusers," *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, vol. 217, no. 1, pp. 29–41, Jan. 2003, doi: 10.1243/095441003763031815.

[62] G. G. Lee, W. D. E. Allan, and K. G. Boulama, "Flow and performance characteristics of an Allison 250 gas turbine S-shaped diffuser: Effects of geometry variations," *International Journal of Heat and Fluid Flow*, vol. 42, pp. 151–163, 2013.

[63] R. K. Singh, Si. Singh, and V. Seshadri, "Influence of the inlet shape on the performance of double pffset transition S-duct with diffusin," *International Journal of Computational Methods*, vol. 05, no. 01, pp. 1–19, Mar. 2008, doi: 10.1142/S0219876208001364.

[64] R. K. Singh, S. N. Singh, and V. Seshadri, "Effect of corner rounding on the performance of the double offset Y-shaped aircraft intake duct," *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, vol. 222, no. 7, pp. 969–983, 2008, doi: 10.1243/09544100JAERO368.

[65] S. Bharani, S. N. Singh, V. Seshadri, and R. K. Dey, "Performance Characteristics of S-Shaped Diffusers with Elliptical Cross-Section," *International Journal of Turbo & Jet-Engines*, vol. 20, no. 2, pp. 141–154, Jan. 2003, doi: 10.1515/TJJ.2003.20.2.141.

[66] K. Saha, S. N. Singh, V. Seshadri, and S. Mukhopadhyay, "Computational analysis on flow through transition s-diffusers: effect of inlet shape," *Journal of Aircraft*, vol. 44, no. 1, pp. 187–193, 2007.

[67] P. K. Sinha *et al.*, "Flow investigation through annular curved diffusing duct," in *AIP Conference Proceedings*, 2010, vol. 1298, pp. 80–90, doi: 10.1063/1.3516429.

[68] P. K. Sinha and B. Majumdar, "Numerical investigation of flow through annular curved diffusing duct," *International Journal of Engineering & Technology IJET-IJENS*, vol. 3, no. 03, pp. 190–196, 2011.

[69] T. Ghisu, M. Molinari, G. Parks, W. Dawes, J. Jarrett, and P. Clarkson, "Axial compressor intermediate duct design and optimisation," in *Proceedings of 48th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conference*, Apr. 2007, pp. 1–14, doi: 10.2514/6.2007-1868.

[70] F. Wallin and L. Eriksson, "Response surface-based transition duct shape optimization," in *Proceedings of ASME Turbo Expo 2006: Power for Land, Sea and Air*, 2006, vol. 2006, pp. 1465–1474, doi: 10.1115/GT2006-90978.

[71] D. Jin, X. Liu, W. Zhao, and X. Gui, "Optimization of endwall contouring in axial compressor S-shaped ducts," *Chinese Journal of Aeronautics*, vol. 28, no. 4, pp. 1076–1086, Aug. 2015, doi: 10.1016/j.cja.2015.06.011.

[72] H. Lu, X. Zheng, and Q. Li, "A combinatorial optimization design method applied to S-shaped compressor transition duct design," *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, vol. 228, no. 10, pp. 1749–1758, Aug. 2014, doi: 10.1177/0954410014531922.

[73] W. Gan and X. Zhang, "Design optimization of a three-dimensional diffusing S-duct using a modified SST turbulent model," *Aerospace Science and Technology*, vol. 63, pp. 63–72, Apr. 2017, doi: 10.1016/j.ast.2016.12.016.

[74] P. Shan, J. Wang, and Z. Lv, "Study of semi-inverse problem for the design of annular S-shaped compressor transition duct," in *Proceedings of ASME Turbo Expo 2015: Turbine Technical Conference and Exposition*, 2015, pp. 1–13, doi: 10.1115/GT2015-43969.

[75] T. Stürzebecher, G. Goinis, C. Voss, H. Sahota, P. Groth, and S. Hammer, "Automated aerodynamic optimization of an aggressive S-shaped intermediate compressor duct," in *Proceedings of the ASME Turbo Expo 2018: Turbomachinery Technical Conference and Exposition*, Jun. 2018, p. V02DT46A002, doi: 10.1115/GT2018-75184.

[76] M. Sharma and B. D. Baloni, "Design optimization of a compressor transition S-shaped duct using a teaching–learning-based optimization algorithm," *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, vol. 41, no. 12, pp. 1–18, Dec. 2019, doi: 10.1007/s40430-019-2072-5.

[77] T. . Lord, W.K., MacMartin, D.G., Tillman, "Flow control opportunities in gas turbine engines.pdf," Denver, 2000.

[78] J. Lin, F. Howard, and G. Selby, "Exploratory study of vortex-generating devices for turbulent flow separation control," in *Proceedings of 29th Aerospace Sciences Meeting*, Jan. 1991, pp. 1–12, doi: 10.2514/6.1991-42.

[79] M. K. Gopaliya, A. Singla, and A. Saini, "Performance improvement of S-shaped diffuser using suction and blowing methods," *IOSR Journal of Mechanical and Civil Engineering Ver. I*, vol. 12, no. 3, pp. 2278–1684, 2015, doi: 10.9790/1684-12312328.

[80] A. D. Walker, A. G. Barker, and J. F. Carrotte, "Numerical design and experimental evaluation of an aggressive s-shaped compressor transition duct with bleed," in *Proceedings of ASME 2011 Turbo Expo: Turbine Technical Conference and Exposition*, 2011, pp. 151–161, doi: 10.1115/GT2011-45628.

[81] S. N. Singh, V. Seshadri, S. Chandel, and M. Gaikwad, "Analysis of the improvement in performance characteristics of S-shaped rectangular diffuser by momentum injection using computational fluid dynamics," *Engineering Applications of Computational Fluid Mechanics*, vol. 3, no. 1, pp. 109–122, 2009, doi: 10.1080/19942060.2009.11015258.

[82] E. Göttlich, "Research on the aerodynamics of intermediate turbine diffusers," *Progress in Aerospace Sciences*, vol. 47, no. 4, pp. 249–279, 2011, doi: 10.1016/j.paerosci.2011.01.002.

[83] E. M. V Siggeirsson, N. Andersson, and F. Wallin, "Numerical and experimental study on bleed impact on intermediate compressor duct performance," in *Proceedings of ASME Turbo Expo 2018 Turbomachinery Technical Conference and Exposition*, Jun. 2018, p. V02BT43A008, doi: 10.1115/gt2018-76649.

[84] E. M. Siggeirsson, N. Andersson, and M. Olander Burak, "Off design simulations of an S-shaped intermediate compressor duct: Experimental validation of DDES and RANS using G3D::Flow," 2020, no. January, doi: 10.2514/6.2020-1761.

[85] A. D. Walker, F. Wallin, R. Bergstedt, and G. Peacock, "Aerodesign and validation of turning struts for an intermediate compressor duct," in *Proceedings of the 22nd International Symposium on Air Breathing Engines.*, 2015, pp. 1–8.

[86] B. A. and B. J. W. Reichert, "Improving diffusing S-duct pefformance by secondary flow control," in *Proceedings of 32nd Aerospace Sciences Meeting and Exhibit*, 1994, pp. 1–14.

[87] B. Reichert and B. Wendt, "An experimental investigation of S-duct flow control using arrays of low-profile vortex generators," in *Proceedings of 31 st Aerospace Sciences Meeting and Exhibit*, 1993, pp. 1–14.

[88] A. R. Paul, P. Ranjan, R. R. Upadhyay, and A. Jain, "Flow control in twin air-intakes using vortex generators," in *Proceedings of the 2011 international conference on Applied & computational mathematics*, 2011, pp. 107–116.

[89] A. R. Paul, P. Ranjan, V. K. Patel, and A. Jain, "Comparative studies on flow control in rectangular S-duct diffuser using submerged-vortex generators," *Aerospace Science and Technology*, vol. 28, no. 1, pp. 332–343, 2013.

[90] A. Anabtawi, R. Blackwelder, P. Lissaman, and R. Liebeck, "An experimental study of vortex generators in boundary layer ingesting diffusers with a centerline offset," in *Proceedings of 35th Joint Propulsion Conference and Exhibit*, Jun. 1999, pp. 1–10, doi: 10.2514/6.1999-2110.

[91] R. K. Sullerey, S. Mishra, and A. M. Pradeep, "Application of boundary layer fences and vortex generators in Improving Performance of S-duct diffusers," *Journal of Fluids Engineering*, vol. 124, no. 1, pp. 136–142, 2002, doi: 10.1115/1.1436096.

[92] R. K. Sullerey and A. M. Pradeep, "Secondary flow control using vortex generator jets," *Journal of Fluids Engineering*, vol. 126, no. 4, p. 650, 2004, doi: 10.1115/1.1779661.

[93] A. M. Pradeep and R. K. Sullerey, "Active flow control in circular and rransitioning S-duct diffusers," *Journal of Fluids Engineering*, vol. 128, no. 6, pp. 1192–1203, 2006, doi: 10.1115/1.2353263.

[94] A. R. Paul, K. Kuppa, M. S. Yadav, and U. Dutta, "Flow improvement in rectangular air intake by submerged vortex generators," *Journal of Applied Fluid Mechanics*, vol. 4, no. 2, pp. 77–86, 2011.

[95] K. Ahmad, J. Watterson, J. Cole, and I. Briggs, "Sub-boundary layer vortex generator control of a separated diffuser flow," in *Proceedings of 35th AIAA Fluid Dynamics Conference and Exhibit*, Jun. 2005, pp. 1–16, doi: 10.2514/6.2005-4650.

[96] R. Bergstedt, "Aero-Design of Aerodynamically Lifting Struts for Intermediate Compressor Ducts." pp. 1–11, 2014.

[97] A. D. Walker, A. G. Barker, I. Mariah, G. L. Peacock, J. F. Carrotte, and R. M. Northall, "An aggressive S-shaped compressor transition duct with swirling flow and aerodynamic lifting struts," in *Proceedings of ASME Turbo Expo 2014: Turbine Technical Conference and Exposition*, Jun. 2014, pp. 1–12, doi: 10.1115/GT2014-25844.

[98] A. D. Walker, I. Mariah, D. Tsakmakidou, H. Vadhvana, and C. Hall, "The influence of fan root flow on the aerodynamic of a low-pressure compressor transition duct," *Journal of Turbomachinery*, vol. 142, no. 1, pp. 1–11, Jan. 2020, doi: 10.1115/1.4045272.