RANS Simulations for Sensitivity Analysis of Compressor Transition Duct

Ivana Milanovic1

University of Hartford, West Hartford, CT, 06070

and

John Whiton2, Razvan Florea3, Stephen Lynch4, Gregory Tillman5, Lary W Hardin6, Mark Stucky7

United Technologies Research Center, East Hartford, CT, 06108

This paper presents a numerical investigation of a strutted annular transition duct used to connect the low pressure and high pressure compressors of aircraft gas turbine engine. Data was generated for one- and six-strut configurations with the United Technologies (UTC) flow solver. For reference, some cases were also run with the ANSYS FLUENT software package. The validity of the computed velocity and pressure loading distributions was established with the existing experimental data available in the open literature. Mean streamwise velocity profiles are in good agreement with data along the entire length of the duct with some discrepancies in the boundary layers adjacent to each casing. Axial velocity distributions at all locations show a good match in the width of the wake. However, the wake strength is not always well matched. Axial velocity contours illustrate a well-defined wake at the strut trailing edge with no significant regions of flow separations, similar to the experimental data set. Obtained axial static pressure loading distributions along the midline of the duct, about the strut and duct-strut combination indicate good match with measurements. The static pressure loading distributions along the strut at different heights are presented and compare very well with experiment. Additionally, at 10 percent strut height the superposition of strut and duct static pressure distributions shows good agreement between simulation and theoretical data up to 70 percent length. The loss distributions across the duct at four axial locations and integrated losses indicate increasing development of loss near the endwalls through the duct. Finally, sensitivity studies examining the influence of duct length and Mach number on pressure loading were also performed for the case of the one-strut duct.

Nomenclature

\[ A = \text{area} \]
\[ C_p = \text{pressure coefficient} \]
\[ L = \text{transition duct axial length} \]
\[ M = \text{Mach number} \]
\[ P = \text{stagnation pressure} \]
\[ R = \text{radius of flow curvature} \]
\[ U = \text{mean streamwise velocity component} \]
\[ U_{\text{mean}} = \text{area-weighted spatial mean value} \]

1 Professor of Mechanical Engineering, Department of Mechanical Engineering, 200 Bloomfiled Ave., Senior Member.
2 Principal Eng/Sci, Aerodynamics, 411 Silver Lane, MS 129-89.
3 Principal Eng/Sci, Aerodynamics, 411 Silver Lane, MS 129-89.
4 Eng/Sci, Aerodynamics, 411 Silver Lane, MS 129-89, Member.
5 Sr Eng/Sci, Program Management, 411 Silver Lane, MS 129-89, Associate Fellow.
6 Staff Eng/Sci, Aerodynamics, 411 Silver Lane, MS 129-89, Senior Member.
7 Lead Tech, Thermal Management, 411 Silver Lane, MS 129-89.

American Institute of Aeronautics and Astronautics
I. Introduction

Transition (intermediate, S-shaped) ducts connect the low-pressure and high-pressure compressors of aircraft turbine engine. Well designed ducts minimize flow separation losses and have uniform flow at the exit. Current efficiency and noise requirements for the turbo-fan engine result in high by-pass ratios and considerable differences in radial dimensions between high- and low-pressure components. Consequently, the design of more aggressive compressor transition duct needs to address potential separation, structural, and service issues. While shorter duct length reduces the overall weight of the engine, it increases separation, i.e. duct 'loading.' Additionally, the duct is usually intersected with struts both for structural and service purposes. Their thickness-to-chord ratio, \( t/c \), likely determines whether the flow along the strut-hub corner separates.\(^{1}\)

A number of investigations performed with the Loughborough University of Technology (LUT) low-speed one-spool compressor rig have examined unstrutted and strutted annular transition ducts with area ratio \( (AR=A_{ex}/A_{in}) \) of one and low and moderate loading. The early work by Britchford et al.\(^{2,4}\) provided experimental data on the mean flow field and turbulence within the duct using five-hole probes and Laser Doppler anemometry (LDA). Two inlet cases were considered, 'clean' inlet, in which boundary layers were developed along an upstream entry length, and a single stage axial compressor upstream of the duct. Numerical simulations of the 'clean' inlet case with two turbulence models, standard \( k-\varepsilon \) and Reynolds stress transport (RST), indicated that the mean velocity profile and shear stress are predicted better by latter. The most pronounced difference was the RST prediction of the effect of curvature in suppressing shear stress on the inner wall, not revealed by the \( k-\varepsilon \) model.

The detailed experimental investigation of the strutted LUT transition duct had been performed by Bailey et al.\(^{5,6}\) A single strut was characterized with NACA 65 profile and 0.12 thickness-to-chord ratio on the mean surface of revolution, while hub and tip were defined with NACA 63 \( (t/c=0.104) \) and NACA 66 \( (t/c=0.138) \) profiles, respectively. Again, both aforementioned inlet cases were examined. Duct and strut combined had a low loading. While the localized strut effect was relatively large, its influence on overall performance was not significant.

The LUT duct geometry and associated measurements have also been utilized in a number of computational investigations validating numerical results and optimizing transition duct. One of the first studies on the reduction of the transition duct length and radius was done by Kim et al.\(^{7}\) Three dimensional Reynolds Averaged Navier-Stokes (3D RANS) simulations used Yang-Shih \( k-\omega \) turbulence closure model and experimental data obtained with the rotor and outlet guide vane (OGV) as the inlet flow condition. The axial length and the radius of the transition duct were investigated separately by length reduction until the occurrence of flow separation. The axial and radial limits of the LUT geometry were found to be 76\% and 86\% of the original length and radius, respectively.

Studies performed at Whittle Laboratory, University of Cambridge, utilized a low-speed two-spool compressor rig for a duct with axial flow. The duct radius change, the inlet height, and the area ratio \( (AR = 1) \) were fixed and could not be changed. Ortiz Duenas et al.\(^{8}\) examined the effect of the length on the performance of the unstrutted ‘linear’ (two-dimensional) duct. Computational investigation with FLUENT had produced duct designs subsequently tested to verify the results of simulations. The authors used both one equation Spalart-Allmaras (SA) and the two-equation \( k-\omega \) SST turbulence models and noted slightly better results for the former. Experimental investigation with duct lengths of 100\%, 74\% and 64\% revealed an increase in loss as the duct becomes shorter and
the flow approaches separation. The size of the separation was found to be dependent on Reynolds number. The difference in loss generation caused by length reduction raised questions on operational design space.

Engine representative inlet condition with an upstream compressor was studied by Karakasis et al. on two transitional ducts with eight struts. The first duct was axisymmetric, and the second was a non-axisymmetric and 20% shorter. With fixed radius change, inlet height, and area ratio, the loading between the ducts was varied by reducing the length of the second duct and increasing the strut thickness-to-chord ratio. The three dimensional hybrid mesh flow solver HYDRA used the Spalart-Allmaras turbulence model and experimental data as the inlet flow condition. Comparison of ducts indicated identical performance and, consequently, a possibility of more extreme designs with no increase in loss.

The survey of published work has revealed relatively small number of studies on compressor transitional duct that may reflect the reluctance on the part of gas turbine industry to publish design and test data rather than a lack of interest in the topic. Investigations on the effect of duct length and radius change, Reynolds number, inlet swirl, design integration and design space limits have been conducted using both experimental and numerical approaches. RANS simulations have exhibited varied prediction accuracy depending on the turbulence model.

This paper presents a numerical investigation of the strutted annular transition duct. Data was generated for one- and six-strut configurations with the UTC flow solver. For reference, some cases were also run with the ANSYS FLUENT software package. Three dimensional RANS simulations were adopted as an investigation tool with \( k-\omega \) and SA turbulence closure models. After the main flow features had been calculated and the computed velocity and pressure loading distributions were validated, sensitivity studies examining the influence of duct length and Mach number on pressure loading were performed for the case of one-strut duct.

Computational predictions indicate very good agreement with experimental data. The current study suggests that modern computational fluid dynamics (CFD) can be used in the duct design as the alternative to time and resource consuming measurements to rapidly evaluate broad regions of design space, provide better spatial resolution, and determine the interplay between flow physics and geometry of the duct.

II. CFD Prediction Tool

A. Flow Solver

The UTC flow solver used in the current simulations is a structured, multi-block code based on a numerical method for compressible flow equations first presented in Ni. The RANS equations for an ideal gas are solved numerically using a procedure based on the Lax-Wendroff multiple grid scheme. The algorithm employs centered second-order spatial differencing and second- and fourth order smoothing. The smoothing is applied along the flow direction and set to a minimal level in order to limit numerical dissipation. Turbulence closure is achieved using the \( k-\omega \) turbulence model and quasi-two-dimensional non-reflecting conditions are applied at the inflow and outflow boundaries. Domain decomposition and parallel capability permit large problems to be handled. The core of the current multi-block structured solver has been extensively validated for turbomachinery flows over the last 30 years. For reference, some six strut cases were also run with ANSYS FLUENT 14.0 software package. The second order accurate segregated implicit solver was used. Both one equation Spalart-Allmaras (SA) and the two-equation \( k-\omega \) turbulence models were considered.

B. Computational Domain

Three-dimensional view of one-strut configuration is illustrated in Fig.1. The computational domain for the present study included definition of the duct geometry (details of which are provided in the section on Experimental Data Set), and the boundary conditions defining mass flow rate.

An in-house mesh generator produced a hybrid O-H meshes. An orthogonal body-fitted O mesh captured gradients close to the strut, while an H mesh was used in the passage. In the current work, an O-H grid topology was constructed using 177 × 17 × 81 nodes in the O-block, and 193 × 65 × 81 nodes in the H-block.

Radial profiles of velocity and turbulence quantities were used as inlet boundary condition and matched the measured profiles at the duct inlet plane. Radial equilibrium outlet boundary conditions were used to match the mass flow.

III. Results and Discussion

A. Experimental Data Set
As indicated previously, the survey of published work has revealed a relatively small number of studies on compressor transitional ducts. Moreover, only a few of them were comprehensively documented. The LUT cases feature detailed descriptions of the facility, duct geometry, test conditions and experimental measurements in a number of papers and two Ph.D. theses. Without an axial compressor stage upstream of transition duct, air was drawn from the atmosphere into a plenum by compressors located within the exhaust. The flow adjacent to the inner and outer casings was artificially tripped before entering the inlet section, which was seven hydraulic diameters long. ‘Clean’ inlet allowed the casing boundary layers to develop along the entry length. Downstream of the duct a parallel settling length of 3.1 hydraulic diameters was provided for air prior to the exhaust plenum.

The duct was of constant flow area with an inlet passage height \( h_{in} \) of 2.8 in (71.1 mm) and an axial length \( L \) of 9.52 in (241.8 mm). The radius of curvature along the mean line at inlet and outlet was 2.55 \( h_{in} \) and 3.23 \( h_{in} \), respectively. Figure 2 illustrates the geometry of the transitional duct and locations of some of the measurement planes with respect to mid-passage height. The duct inlet and exit boundary conditions are at \( x/L = -0.4 \) and 1.4, respectively.

Figure 1. One-strut configuration. Figure 2. Locations of the measurement planes in the duct.

A single strut was characterized with NACA 65 profile and 0.12 thickness-to-chord ratio on the mean surface of revolution, while hub and tip were defined with NACA 63 \((t/c = 0.104)\) and NACA 66 \((t/c = 0.138)\) profiles, respectively. Consequently, the annular strutted duct was defined with the five non-dimensional parameters: the area ratio \( A_{ex}/A_{in} = 1 \), length to inlet height ratio \( L/h_{in} = 3.4 \), the inlet hub to tip radius ratio \( r_{in,h}/r_{in,t} \) = 0.8, exit to inlet mean radius ratio \( r_{ex,m}/r_{in,m} \) = 0.8, and thickness-to-chord ratio, \( t/c \) = 0.12.

B. Validation of Predicted Results

Validation of the predicted results was done with the LUT data set for the case of ‘clean’ inlet and one strut. Current CFD investigation adopted LUT definitions of pressures and pressure coefficients. The spatially averaged values of stagnation and static pressure were derived by mass weighting the appropriate individual values:

\[
\bar{p} = \frac{1}{m} \int p dm, \quad \bar{p}_a = \frac{1}{m} \int p dm
\]

The static pressure recovery coefficient and the stagnation pressure loss coefficient were defined by:

\[
C_p = \frac{\bar{p} - \bar{p}_a}{\bar{p}_a}, \quad \lambda_{a-b} = \frac{\bar{p}_a - \bar{p}_b}{\bar{p}_a - \bar{p}_a}
\]

where ‘a’ and ‘b’ are upstream and downstream planes, respectively, and \( p \) is the static pressure at the given location.

Table 1 illustrates computational test matrix. Validation data was generated for one- and six-strut configurations, cases A and F, respectively, with the UTC flow solver. Three dimensional RANS simulations were adopted as an investigation tool with \( k-\omega \) turbulence closure model. For reference, case F was also run with the ANSYS FLUENT software package. Both one equation Spalart-Allmaras (SA) and the two-equation \( k-\omega \) turbulence...
models were considered. Sensitivity studies examining the influence of Mach number (cases B and C) and duct length (cases D and E) on the pressure loading were performed for the one-strut duct with the UTC flow solver only.

Table 1 Computational test matrix

<table>
<thead>
<tr>
<th>PARAMETERS</th>
<th>CASE 1 STRUT A</th>
<th>CASE 1 STRUT B</th>
<th>CASE 1 STRUT C</th>
<th>CASE 6 STRUTS D</th>
<th>CASE 6 STRUTS E</th>
<th>CASE 6 STRUTS F</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach number (M)</td>
<td>0.08</td>
<td>0.25</td>
<td>0.59</td>
<td>0.08</td>
<td>0.08</td>
<td>0.08</td>
</tr>
<tr>
<td>Mass flow rate ratio</td>
<td>1</td>
<td>3</td>
<td>6</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Duct length (L)</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>0.9</td>
<td>0.8</td>
<td>1</td>
</tr>
</tbody>
</table>

Figure 3 compares predicted mean streamwise velocity profiles of the six-strut configuration with the experimental data at four axial locations (x/L = 0, 0.125, 0.5, 1.0). The spatial average velocity (U_{mean}) for the annular passage of 92.85 ft/s (28.3 m/s) was used as a reference value for nondimensionalizing the mean velocity at a given plane.

Figure 3 compares predicted mean streamwise velocity profiles of the six-strut configuration with the experimental data at four axial locations (x/L = 0, 0.125, 0.5, 1.0). The spatial average velocity (U_{mean}) for the annular passage of 92.85 ft/s (28.3 m/s) was used as a reference value for nondimensionalizing the mean velocity at a given plane.

Velocity profiles are in good agreement with data along the entire length of the duct with some minor discrepancies in the boundary layers adjacent to each casing. The inner wall region is captured better by the UTC k-ω and FLUENT SA models at the first three locations, while the FLUENT k-ω accurately outlines inner velocity profile at the last station. Predictions for the outer wall boundary layer feature larger discrepancies increasing with the axial distance. Over-prediction and local maxima in the profiles that were otherwise not registered by the experiment are seen in all three computational models. This is consistent with the results of Britchford et al.² and attributed to the inaccuracy of the turbulence models. Notwithstanding the small differences seen between the measurements and simulations in Fig. 3, the overall predicted profile shapes and velocity levels are in very good agreement with the measurements.

Comparison of normalized mean streamwise velocity profiles in the wake of the six-strut configuration with measurements at plane x/L = 1 and two span locations (50% and 10%) are presented in Fig. 4. Velocity distributions at all locations for RANS calculations indicate a good match in the width of the wake; however, as expected, the strength is less well matched.

Contour plots of normalized streamwise velocity at the duct exit (plane x/L = 1) are shown in Fig. 5 for cases A and F. Axial velocity contours illustrate a well-defined wake with no significant regions of flow separations, similar
to the experimental data set. The computations also predict thicker boundary layer at the inner wall, which is comparable to the measurements.

Figure 4. Streamwise velocity profiles ($x/L = 1$).

A comparison of the downstream wake evolutions obtained with (1) FLUENT SA and $k-\omega$ models, (2) FLUENT and UTC $k-\omega$ models, as well as (3) UTC flow solver predictions for cases F and A, although qualitatively similar, reveal two differences. A low speed inner wall region appears to persist longer and has larger spread for the former models. Similarly, a region with velocity overshoot is more pronounced for the former models.

Figure 5. Velocity contours ($x/L = 1$).

The UTC flow solver model predictions of the axial static pressure distributions along the midline of the duct for cases A and F are compared with the experimental data of Bailey et al.\textsuperscript{5} in Fig. 6. Both simulations underpredict the maximum static pressure recovery coefficient. However, case A appears to follow more closely the measurements and provides a better estimate of the minimum static pressure coefficient.

Figure 7 illustrates the static pressure distributions along the strut at 10, 50, and 90 percent strut heights with experimental and computational data for the case A. Also shown in Fig. 7 is the theoretical curve, i.e. the distribution obtained from measurements of the midspan NACA 65 profile under free-stream conditions\textsuperscript{11} for comparison purposes. The pressure distribution is a composite of the pressure profile of the airfoil section in the
absence of the duct (i.e. the ‘theoretical’ curve) and that of the duct with no struts. At midspan, the impact of the duct pressure gradient is a minimum. Near the hub and tip, the differences between the in-situ pressure distribution and the airfoil pressure profile in a free stream will be greater. While only minor differences are observed between the two data sets at any given height, it is noted that simulations somewhat overpredict static pressure coefficients. At midheight the pressure gradients are favorable up to about 60 percent chord location, a value slightly higher than those obtained by measurements and from the theoretical data. As indicated by previous studies, distribution differences at various strut heights are primarily due to the streamwise pressure gradient in the duct. For example, abrupt change in wall curvature creating pressure gradient is clearly seen in distributions at 10 and 90 percent strut height with unfavorable pressure gradients above 15 and 65 percent chord, respectively. The pressure gradient trends in the experimental data are in general quite well predicted by the CFD.

Finally, the overall stagnation pressure loss within the duct was also obtained. The loss coefficient ($\lambda$), based on the difference in mass weighted stagnation pressure between duct inlet ($x/L = 0.0$) and exit ($x/L = 1.0$), relative to the total mass weighted dynamic head at duct inlet, is about 1.95 percent. This value is in close agreement with the experiment (2 percent). Although only the exit plane losses were compared, the results suggest high quality predictions.

C. Sensitivity Analysis
Sensitivity studies examining the influence of Mach number (cases B and C) and duct length (cases D and E) on pressure loading were performed for the one-strut duct with the UTC flow solver.

Figures 8 and 9 illustrate the static pressure distributions along the strut at 10, 50, and 90 percent strut heights for three Mach numbers and duct lengths, respectively. All figures feature baseline configuration A for comparison purposes. Pressure distributions for various Mach numbers, shown in Fig. 8, exhibit the same trend for all strut heights, with case C having the lowest pressure coefficient values between approximately 10 and 80 percent chord. At midheight, the pressure gradients are favorable up to about 60 percent chord location for all cases. Similarly to the baseline case, distributions at 10 and 90 percent strut height, have unfavorable pressure gradients above 15 and 65 percent chord, respectively. Increase in Mach number results in lower pressure coefficients; however, in the given range, it does not seem to influence the start of unfavorable pressure gradients.

Figure 8. Static pressure distribution about strut (cases A, B and C).
Pressure distributions for different duct lengths have the same trends for all three strut heights, with case E featuring the lowest values throughout, as seen in Fig. 9. At midheight, the pressure gradients for cases D and E are favorable up to about 60 and 45 percent chord location, respectively. Pressure distributions for the aforementioned configurations at 10 and 90 percent strut height reveal unfavorable pressure gradients above 10 and 65 percent chord, respectively. Shortening of the duct length results in lower pressure coefficients; however, in the given range, it does not seem to influence the start of unfavorable pressure gradients.

Figure 9. Static pressure distribution about strut (cases A, D and E).

Axial static pressure distributions along midline of the duct are shown in Figs. 10 and 11 for three Mach numbers and duct lengths, respectively. Pressure distributions for various Mach numbers feature predominantly positive pressure gradient as illustrated in Fig. 10. Cases B and C have similar trends with the rise in static pressure coefficient from a local minimum of about -0.35 to local maximum of +0.25 along approximately 65 percent of the duct length. Pressure distributions for different duct lengths are shown in Fig. 11. As the duct length is changed from the baseline length to 90 and 80 percent of baseline, the static pressure coefficient rises from a local minimum (-0.3, -0.4 and -0.5, respectively) to +0.2 along approximately 70 percent of the duct length.

Figure 10. Static pressure distribution (cases A, B and C).

Contour plots of streamwise velocity at the duct exit (plane x/L = 1) for cases D and E are shown in Fig. 12. Similar to the baseline case, both contour plots feature a well-defined wake with no significant regions of flow separation and thicker boundary layer at the inner wall. The decrease in the duct length results in tighter and higher low speed inner region, and larger velocity overshoot zone.

Figure 11. Static pressure distribution (cases A, D and E).

IV. Conclusion

A numerical investigation of strutted annular transition duct was undertaken to assess the UTC flow solver as an investigation tool, and examine the influence of duct length and Mach number on pressure loading. Comparison of computational results with the existing experimental data showed good agreement for mean streamwise velocity profiles, axial velocity distributions, static pressure recovery and overall stagnation pressure loss coefficients. Next, sensitivity studies examining the influence of duct length and Mach number on pressure loading were performed. Both increase in Mach number and shortening of the duct length result in lower pressure coefficients; however, in the given range, these changes do not influence the start of unfavorable pressure gradients.

American Institute of Aeronautics and Astronautics
Current study suggests that modern CFD can be used in the duct design as the alternative to time and resource consuming measurements to rapidly evaluate broad regions of design space, provide better spatial resolution, and determine the interplay between flow physics and geometry of the duct.

Figure 12. Streamwise velocity contours ($x/L = 1$).

Acknowledgments

This work was carried out while the first author was on a sabbatical leave from the University of Hartford. The authors wish to acknowledge Dr. Yuan Dong, Ms. Lisa Brilliant, and Dr. Anil Prasad of Pratt & Whitney for their technical assistance and oversight in conducting the subject research.

References