Numerical and experimental investigation of a serpentine inlet duct

Aaron M. Kirk, Joaquin I. Gargoloff, Othon K. Rediniotis and Paul G.A. Cizmas*

Department of Aerospace Engineering, Texas A&M University, College Station, Texas 77843-3141

(Received 10 February 2009; final version received 17 February 2009)

This article presents a numerical and experimental investigation of the flow inside an ultra-compact, serpentine inlet duct. The numerical analysis used two flow solvers: FLUENT®, a commercial code, and UNS3D, an in-house code. The flow was modelled using the Reynolds-averaged Navier-Stokes equations. The turbulence effects were modelled by using the shear-stress transport $k$–$\omega$ model. The numerical investigation was compared against experimental data obtained in an open-circuit, low-speed wind tunnel in the Fluid Dynamics Laboratory at Texas A&M University. The numerical simulations and experimental testing were performed to reveal the separation points and the strong secondary flow phenomena within the inlet. UNS3D overpredicted the location of the first separation point by 9 mm and the location of the second separation point by 1 mm, while the area-averaged pressure loss coefficient was 5% higher than in the experiment. The numerical results of UNS3D agreed better with the experiment than those of FLUENT.

Keywords: inlet flow; secondary flows; flow separation; Reynolds-averaged Navier-Stokes equations; verification and validation

1. Introduction

Low-observability has become a major design factor in the current development of military aircraft. With regards to propulsion, this trend has led to engines buried within the fuselage fed by S-shaped inlets that do not provide a direct line of sight to the engine blades. The improvements in stealth capabilities provided by such inlet ducts come at the expense of reduced engine performance. Examples of aircraft with S-shaped inlet include not only military aircraft, such as General Dynamics F-16 and McDonnell-Douglas F-18, but also commercial aircraft such as Boeing 727 and Lockheed L1011 Tristar. Serpentine inlets are also used on turbofan engines with offset gearboxes (McDill and Tolle 1983).

Serpentine inlets are often susceptible to the emergence of strong secondary flows that decrease efficiency, hinder compressor stability and reduce stall and surge margins (Brear et al. 2004). Therefore, it is necessary to employ analytical and computational techniques to predict such flow behaviour and improve aerodynamic performance when designing these inlet ducts.

The development and evolution of the secondary flow structures in S-shaped ducts have been well documented in the literature. Prandtl formally separated steady secondary flows into two categories (Prandtl 1952, p. 145–149): secondary flows of the first kind, which are due to the mean flow skewing, and secondary flows of the second kind, which are due to the non-uniformities in wall turbulence. The secondary flows of the second kind occur principally in non-circular ducts or in boundary regions formed at the streamwise corner or edge of a surface (Perkins 1970). Since the inlet investigated herein has an elliptical/circular cross-section, the discussion of the secondary flows will be limited to first kind only.

The initial development of the secondary flow can be explained by the growth of the streamwise component of vorticity. The mechanism of vortex formation in a bend is properly captured by the Squire-Winter-Hawthorne inviscid secondary flow formula, which states that a stream line deflection $\alpha$ in a plane containing the stream line and the vorticity vector $\Omega$ results in a rotation of the vorticity vector towards the flow direction (Squire and Winter 1951, Hawthorne 1951). This rotation takes place in an opposite sense to the stream line deflection $\alpha$ in a plane containing the stream line and the vorticity vector $\Omega$. Because of the secondary flow vorticity, the boundary layer fluid migrates towards the bottom side of the pipe in the first bend of the S-duct.

*Corresponding author. Email: cizmas@tamu.edu

International Journal of Computational Fluid Dynamics
Vol. 23, No. 3, March 2009, 245–258
ISSN 1061-8562 print/ISSN 1029-0257 online
© 2009 Taylor & Francis
DOI: 10.1080/10618560902835558
http://www.informaworld.com
As the flow negotiates the bends of the inlet, a centrifugal force is generated on the core flow, causing it to accelerate. This action, in conjunction with flow separation towards the inboard region of the bends, produces a cross-stream pressure gradient. With lower static pressure near the centre of the duct, the low energy boundary layer flow migrates inward, where the merging flow is pushed away from the wall and back towards the outside of the bend. From this motion, the lift-off of two counter-rotating vortices is produced at each bend.

1.1. Previous work

Fox and Kline (1962) experimentally investigated the flow in curved subsonic diffusers with turning angles up to 90°. Each diffuser had a circular arc centre line and a linear area distribution normal to the centre line. Secondary flow in the form of a roller of fixed height was observed at turning angles equal to or larger than 40°. Rowe (1970) did measurements and computation of the flow in bend pipes. Both the experimental and numerical results showed that secondary flows cause a complete interchange of the slow moving wall fluid and the faster central core. The flow was numerically simulated by solving the Poisson equation for the secondary flow stream function. The measured flow pattern was reasonably well predicted up to a bend angle of approximately 75°.

The experiments of Bansod and Bradshaw (1972) on S-shaped ducts showed that the large departures from axisymmetry in the exit plane are due to the expulsion of boundary layer fluid by strong longitudinal vortices embedded in the boundary layer near the outside of the second bend. These vortices are generated by a mechanism that involves a combination of cross-flow, convergence and acceleration but is basically inviscid, given an initial boundary layer.

Vakili et al. (1985) used a parabolised Navier-Stokes solver to predict the compressible flow in an S-duct with a Mach number of 0.6 and a Reynolds number based on the duct parameter of 1,760,000. The turbulence effects were modelled using an unspecified algebraic mixing length turbulence model. The numerical results underpredicted the effects of flow distortion.

The flow in the S-duct investigated by Vakili et al. (1985) was later simulated by Harloff et al. (1993) and Tsai et al. (1995). Both numerical simulations used a Navier-Stokes solver. Harloff et al. (1993) used the Baldwin-Lomax algebraic turbulence model while Tsai et al. (1995) used the Johnson-King algebraic turbulence model. The computed total pressures of Harloff et al. (1993) were in good agreement with the experimental data, while the static pressure did not compare as well with the experiments. The numerical results reported by Tsai et al. (1995) showed that the Johnson-King turbulence model produced better results than the Baldwin-Lomax turbulence model.

Nichols (1991) used a Navier-Stokes solver to simulate the flow in the RAE M2129 circular S-duct. The solver used either the $k-\varepsilon$ or the Baldwin-Lomax turbulence models. The agreement between the numerical simulation and the experimental data were excellent for the case of low mass flow rate, in which the flow did not separate. For the case of high mass flow rate, in which there was massive flow separation, the numerical simulation correctly predicted the trends, although the absolute levels were not reproduced. In all cases, the $k-\varepsilon$ model performed better than the algebraic model.

The RAE M2129 circular S-duct was subsequently numerically investigated by several researchers, including Anderson and Gibb (1993), and Anderson et al. (1994), Menzies et al. (2002) and Mohler (2004). Anderson et al. (1994) used two flow solvers: a full Navier-Stokes solver with the Baldwin-Lomax turbulence model and a reduced Navier-Stokes solver with the McDonald-Camarata turbulence model. The two solvers generated similar results that agreed reasonably well with the experimental performance data. Both solvers, however, consistently predicted separation further downstream than what was found in the experiment. Menzies et al. (2002) used an Euler solver and a Navier-Stokes solver with three turbulence models: Spalart-Allmaras, $k-\omega$, and Menter’s shear stress transport (SST). The results obtained using the SST turbulence model were the closest to the experimental
data, although the predicted pressure levels were approximately 6% higher.

In another computational fluid dynamics (CFD) analysis of the M2129 S-duct, Mohler (2004) used a Navier-Stokes solver with a Spalart-Allmaras turbulence model to predict the flow with and without vane effectors. The computations predicted vortices that were larger than in the experiment and had a greater amount of pressure loss. These inaccuracies were attributed to the Spalart-Allmaras turbulence model because it had been shown to overpredict the size of separation bubbles.

Brear et al. (2004) used a Navier-Stokes solver with a $k-o$ turbulence model to predict the flow in a serpentine inlet of an uninhabited combat air vehicle at cruise conditions. The numerical predictions were performed for the full aircraft whereas experiments used a bellmouth contraction to simulate cruise conditions on a stationary rig. The numerical prediction was performed at a higher Reynolds number and lower Mach number than the experimental flow visualisation. The numerical prediction, however, captured the main flow features in the serpentine inlet. In a study that employed a code with a similar turbulence model to the one used herein, CFD properly captured the effect of Mach number on duct performance, but tended to overpredict pressure loss (Berrier and Allan 2004).

On the basis of the results of the previous work on simulating the flow in S-ducts, one can conclude that computational methods capture the salient features of the flow but fail to accurately portray the physical phenomena when applied to a flow pattern as complex as that present within a compact, serpentine duct. Limitations in turbulence modelling and errors associated with meshing and solution schemes diminish the accuracy of CFD predictions.

The purpose of this study is to compare the experimental results obtained in-house on a serpentine inlet with an aggressive centreline curvature and area diffusion against two sets of numerical results: one generated using a commercial code and the other one generated using an in-house flow solver. Both flow solvers used the same turbulence model. The following sections of this article present the experimental setup, the numerical simulation and the comparison of the numerical results and experimental data.

2. Experimental setup

The flow path of interest in this study is an ultra-compact, serpentine inlet designed by Lockheed Martin Aeronautics Company. The inlet is identical to the test article of the research conducted by Hamstra et al. (2002) and Kirk (2006). The duct features a length-to-diameter ratio of 2.5 and consists of a 4:1 aspect ratio, bi-convex entrance section followed by a dual-turn offset section and a diffusing, elliptical-to-circular exit section. From a typical UAV application, the flow path model is scaled approximately 40–50%, yielding a length of 63.5 cm and an exit diameter of 25.4 cm. The duct geometry is shown in Figures 2 and 3.

The duct was mounted to an open-circuit, low-speed wind tunnel in the Fluid Dynamics Laboratory at Texas A&M University. To prevent local separation and subsequent vortex shedding from the flow negotiating the lip of the entrance section, a bellmouth contraction with an area ratio of 7.4 was added. A flow conditioner was constructed and placed upstream of the bellmouth to provide low-turbulence ingestion of

Figure 2. Assembled inlet duct.
the ambient air. Several experiments were performed to reveal the separation points and the formation of strong secondary flows within the inlet. Using a miniature, seven-hole probe, the engine face plane was surveyed to obtain static pressure, stagnation pressure, and the magnitude and direction of velocity. For this test, the data acquisition grid consisted of 18 radial points, spaced 6.35 mm apart, and 36 circumferential rays at 10° increments. Because of issues involving probe access and wall effects, no measurements were taken within 19.05 mm of the wall. To locate the regions of flow separation and map the static pressure distribution along the length of the inlet, pressure taps incorporated in the duct upper and lower walls along the symmetry plane were utilised. In addition, surface flow visualisation tests were conducted to qualitatively present the secondary flow development. These experiments, which utilised a titanium dioxide, kerosene, oleic acid and mineral oil mixture painted onto the duct surface, revealed the locations of the separation and vortex lift-off points. The uncertainty in the experimental measurements was reported in Kirk (2006). The flow conditions at the entrance to the inlet duct for testing are shown in Table 1.

3. Numerical simulation

This section presents the details of the computational methods employed in this study. The serpentine duct geometry proved to be a particularly challenging CFD problem because of its highly curvilinear nature leading to large gradients in the flow at multiple locations. The problem required careful investigation of the flow physics to develop suitable meshes capable of resolving all important characteristics at various length scales.

3.1. UNS3D flow solver

3.1.1. Grid generation

The computational domain was discretised using a hybrid grid, which consisted of hexahedral and three-sided (triangular) prismatic elements (Kim and Cizmas 2002). Using a computer-aided design model provided by Lockheed Martin, discrete points defining the surface of the duct were extracted and imported into the grid generation code. The computational domain was then divided into layers that were topologically identical, and that spanned from the entrance to the exit of the duct. The number of layers was chosen as a result of a grid refinement process. The topologically identical nodes of adjacent layers were interconnected to generate the volume elements that were either triangular prisms or hexahedra. Each layer had a structured O-grid around the duct walls, and an unstructured grid at the interior of the duct. The O-grid allowed good control over the mesh size near the walls and permitted clustering cells in the direction normal to the wall to properly capture boundary layer effects. Parameters governing the grid generation included the overall O-grid height, the number of nodes in the O-grid (both radially and circumferentially), and the height of the first and last cells of the O-grid. The unstructured grid was flexible in filling the rest of the domain. The grid generator allowed the control of the maximum area of the faces of the triangular prisms of the unstructured grid and the degree of mesh smoothing.

The UNS3D flow solver does not yet have a symmetry boundary condition, and for this reason the computational domain of the duct was not halved at the symmetry plane as was done for the FLUENT® flow solver. After a grid refinement study was performed, the selected mesh had 134 layers and 7840 nodes per layer, that is, a total of 1,050,560 nodes. The structured O-grid contained 263 circumferential nodes and 21 nodes in the direction normal to the wall. The cell height had a growth rate of 1.3 in the direction normal to the wall. The $y^+$ number was less than 2.0.

The wall mesh for the entire duct is shown in Figure 4a. It is evident from the figure that the layers were not evenly spaced throughout the length of the duct. Tighter clustering was utilised in regions of larger

<table>
<thead>
<tr>
<th>Table 1. Flow conditions at the entrance of the inlet duct.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Velocity</td>
</tr>
<tr>
<td>Mach number</td>
</tr>
<tr>
<td>Mass flow</td>
</tr>
<tr>
<td>Duct inlet area</td>
</tr>
<tr>
<td>Reynolds number based on exit diameter</td>
</tr>
</tbody>
</table>

Figure 3. Dimensions of inlet duct in metres.
flow gradients near the bends. Figure 4b and 4c show the meshes at the inlet and exit of the duct, respectively.

3.1.2. Numerical method

The flow was modelled using the Reynolds-averaged Navier-Stokes equations. Turbulence effects were modelled by using the SST $k$–$\omega$ turbulence model of Menter (Menter 1994). This model was implemented in the UNS3D code that was originally developed for turbomachinery flows (Han and Cizmas 2003) and then used for aeroelastic applications (Gargoloff et al. 2006). The governing equations were solved using a finite volume method. Cell-averaged variables were stored at the nodes of the grid, that is, the vertices of the cells. The governing equations were discretised using mesh duals as control volumes. The median dual-mesh was adopted herein because of its flexibility to handle unstructured mixed meshes (Barth 1991).

An edge-based data structure was utilised for the discretisation of the governing equations because of its flexibility and efficiency in handling different element types. The numerical scheme, however, was node-based and consequently the solution was obtained at the nodes of the mesh. The surface integral of the inviscid and viscous fluxes were approximated as a sum over the faces of each control volume. The source terms were calculated using the control volume-averaged solution and the derivatives of flow variables at the cell centroid.

The numerical algorithm was second-order accurate in time and space. Upwinding was used for computing the convective fluxes. Roe’s approximate Riemann solver (Roe 1981) was used with Harten’s entropy fix (Harten 1983). The gradients of the viscous fluxes were computed using the least-squares method. Piecewise linear reconstruction with a limiter function (Venkatakrishnan 1995) was used to achieve second-order spatial accuracy. Implicit residual smoothing...
was used to add an implicit flavour to the four-stage Runge-Kutta explicit time integration.

The UNS3D flow solver was run for 20,000 iterations with a Courant-Friedrichs-Lewy (CFL) number of 2.0. The first 4000 iterations were performed using a first-order solver and the remaining 16,000 with a second-order solver. The simulation took approximately 120 h on a single processor of a Mac G5 with 2.5 GHz dual-core processors and 4 GB of RAM. Figure 5 shows that all the maximum residuals are smaller than $10^{-6}$, and consequently it is expected that the solution is sufficiently converged.

### 3.2. FLUENT® flow solver

#### 3.2.1. Grid generation

The computational domain was discretised using a hybrid grid created in Gambit®. The original geometry was split into fifteen parts using the three-dimensional modelling software, SolidWorks®. This allowed for varying mesh size in different zones of the geometry as required by the flow characteristics of the zone. The geometry was later exported to Gambit as an IGES file. The computational domain was discretised using a hybrid grid that consisted of structured hexahedra around the duct walls and unstructured triangular prisms elsewhere.

A grid convergence test was not done for the FLUENT solver. Instead, the grid size was determined based on the conclusions drawn from the grid size variation done for the UNS3D solver. The grid had 520,000 nodes with 168 grid points along the circumference and 165 layers in the axial direction. The mesh size along the walls was specified such that the $y^+$ number was less than 3.0. The growth rate of the cell height in the direction normal to the wall was 1.3. The computational domain contained only half of the S-duct, as the duct was symmetric about the mid-plane. The grid for the FLUENT simulation is shown in Figure 6.

#### 3.2.2. Numerical method

The CFD simulation was run using FLUENT 6.2. An implicit, cell based scheme was employed to run the simulations. The SST $k$–$\omega$ model of Menter (1994) was used herein. This turbulence model smoothly transitions from the $k$–$\omega$ model in the inner regions of the boundary layer to the high Reynolds number version of the $k$–$\epsilon$ model in the outer part of the boundary layer. In the SST $k$–$\omega$ model, the turbulent viscosity is modified to account for the transport of principal turbulent shear stress. Consequently, the SST $k$–$\omega$ model is superior in performance to both the standard $k$–$\omega$ and the standard $k$–$\epsilon$ model. In addition, the presence of a cross-diffusion term in the transport equation for $\omega$ and a blending function ensure that the model equations behave appropriately in both the near-wall and far-field regions. The algorithm employed to solve the Reynolds-averaged Navier-Stokes equations used a Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) pressure-velocity coupling with a second-order upwind spatial discretisation for momentum, pressure, kinetic energy of turbulent fluctuations, $k$, and specific dissipation rate, $\omega$.

The boundary types were defined as pressure inlet and pressure outlet for the inlet and outlet plane respectively. The mid-plane was defined as a symmetry plane. The remaining boundaries used wall-type boundary conditions.

The solution was run for 1700 iterations, which took approximately 12 CPU hours on an SGI Altix 3700 supercomputer with 1.3 GHz Itanium-2 processors and 256 GB of RAM. Residuals of continuity, $x$-velocity, $y$-velocity, $z$-velocity, $k$ and $\omega$ equations were monitored. As shown in Figure 7, the residuals of continuity equation decreased to approximately $10^{-6}$ whereas the velocities in all directions and $\omega$ dropped below $10^{-5}$. The residuals of turbulent kinetic energy, $k$, were the slowest to decrease as compared to the other variables.

#### 3.3. Boundary conditions

The geometry and flow were symmetrical with respect to the vertical plane along the inlet centreline. Symmetry boundary conditions were applied in FLUENT to reduce the computational effort. These types of boundary conditions are not yet implemented in the UNS3D code. For this reason, the UNS3D code simulated the flow in the entire inlet, while FLUENT simulated only half of the inlet, as mentioned in the previous sections. Except for the symmetry boundary conditions, the boundary conditions applied to
FLUENT and UNS3D solvers were identical. The inlet and exit boundary conditions were taken from the experimental testing parameters and are summarised in Table 2. Adiabatic, no-slip boundary conditions were imposed at walls. In both cases, it was assumed that the inlet turbulence intensity was 1% and the

Figure 6. Gambit/FLUENT grid: (a) wall, (b) inlet and (c) exit.

Figure 7. Residuals of the FLUENT flow solver.
turbulence length scale was 1 mm. The wall roughness height was assumed to be 5. The turbulence kinetic energy was set to zero at walls and the specific dissipation rate was calculated using the rough-surface wall approximation (Wilcox 2000).

4. Results

This section presents the results of the experimental and computational investigation of the serpentine duct flow. The flow phenomena inside the inlet duct were explored and the accuracy of the two CFD methods was assessed by comparing the velocity vectors and the pressure loss coefficient at the engine face, and the surface static pressure on the upper and lower walls along the symmetry plane against experimental results.

4.1. Engine face velocity vectors and pressure loss coefficient

The secondary flow structures that develop within the inlet duct convect downstream and interact at the engine face with the compressor blades to severely reduce engine performance. The shape, size and degree of loss associated with these structures were quantified by examining the velocity vectors and pressure contours at the engine face plane. The pressure loss coefficient, $C_{\text{ploss}} = (P_{\text{tot,}\infty} - P_{\text{tot,ef}})/q_{\infty}$, was used to evaluate losses in the inlet duct. Herein, $P_{\text{tot,}\infty}$ was the total pressure at the entrance to the inlet duct, $P_{\text{tot,ef}}$ was the total pressure at the engine face (the fan face, also called the aerodynamic interface plane), and $q_{\infty}$ was the dynamic pressure at the entrance to the duct.

Figure 8 shows the contours of the pressure loss coefficient $C_{\text{ploss}}$ for the FLUENT simulation, UNS3D simulation, and experimental testing. The colour legend to the right of the experimental plot applies to all plots. Recall that, due to probe access and wall effect considerations, the diameter of the experimental domain is 15% smaller than the computational domain.

Figure 8 shows that there is good qualitative agreement between both CFD solvers and experiment. The CFD simulations predict the emergence of two large, counter-rotating vortices at the top of the engine face plane that shed from the upper surface of the duct just after the second bend. There is some discrepancy in the evolution of the first bend vortices, however. Analysing the FLUENT contour plot, the vortices appear to flatten near the wall, dissipate, and migrate outwards. In opposite fashion, the first bend vortices remain intact and do not migrate in the UNS3D simulation. Neither of these CFD predictions for the first bend vortices are accurate. Instead, the experimental contour plot of $C_{\text{ploss}}$ indicates that the first bend vortices disperse and diffuse toward the second bend vortices.

Quantitatively, the size, shape, and strength of the vortices are not consistent between the sets of results. The FLUENT flow solver predicts second bend vortices that are approximately the correct size, but are more oblong than that of the experimental results. In addition, the pressure loss within the vortices is underpredicted. The area-averaged value of $C_{\text{ploss}}$ for the FLUENT simulation is 20.85% compared with 44.0% for the experiment. The opposite is true for the UNS3D results. The vortices are the correct shape, but are slightly oversized. In the UNS3D simulation, the predicted pressure loss was 48.94%, that is, nearly 5% higher than in the experiment.

Figure 9 shows the effect of grid size and aspect ratio in a crosswise section ($z$-plane) on the pressure loss at engine face. The sensitivity of the results to grid size is remarkable for the flow in this S-duct.

Table 2. Boundary conditions used in the CFD analyses.

<table>
<thead>
<tr>
<th>Boundary Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Mach Number</td>
<td>0.1738</td>
</tr>
<tr>
<td>Inlet Total Pressure</td>
<td>99991.78 Pa</td>
</tr>
<tr>
<td>Inlet Static Pressure</td>
<td>97313.02 Pa</td>
</tr>
<tr>
<td>Inlet Total Temperature</td>
<td>298.15 K</td>
</tr>
<tr>
<td>Exit Static Pressure</td>
<td>97889.49 Pa</td>
</tr>
<tr>
<td>Exit Static Temperature</td>
<td>296.37 K</td>
</tr>
</tbody>
</table>

Figure 8. Pressure loss coefficient at the engine face: (a) FLUENT (b) UNS3D and (c) experimental.
Table 3 shows the number of grid points for the four meshes, the pressure loss coefficient at engine face, and the locations of the separation points. The number of layers that spanned from the entrance to the exit of the duct was varied between 64 and 214. The best compromise between accuracy and computational cost was obtained for 134 layers, as shown in the following paragraph. All grids discussed in Table 3 used 134 layers. These grids had approximately the same number of points per layer, except for grid C1 that had fewer grid points. Grids C1 and C2 had the smallest number of grid points along the circumference. These two grids did not predict flow separation at first bend. When more points were added in the circumferential direction and the number of points in the radial direction was reduced such that the number of points in the O-grid be approximately the same, the results improved significantly (compare grids L2 and

![Table 3](image_url)

**Table 3. Variation of UNS3D results with mesh size in crosswise layers (x plane).**

<table>
<thead>
<tr>
<th>Mesh</th>
<th>C1</th>
<th>C2</th>
<th>L2</th>
<th>F3</th>
<th>Experiment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nodes</td>
<td>5080</td>
<td>7430</td>
<td>7687</td>
<td>7840</td>
<td></td>
</tr>
<tr>
<td>Duct walls (O-grid)</td>
<td>3290</td>
<td>5640</td>
<td>5778</td>
<td>5502</td>
<td></td>
</tr>
<tr>
<td>Total</td>
<td>94</td>
<td>94</td>
<td>214</td>
<td>262</td>
<td></td>
</tr>
<tr>
<td>Circumferential</td>
<td>35</td>
<td>60</td>
<td>27</td>
<td>21</td>
<td></td>
</tr>
<tr>
<td>Radial</td>
<td>1790</td>
<td>1790</td>
<td>1909</td>
<td>2338</td>
<td></td>
</tr>
<tr>
<td>Duct centre (unstructured grid)</td>
<td>1790</td>
<td>1790</td>
<td>1909</td>
<td>2338</td>
<td></td>
</tr>
<tr>
<td>Pressure loss, $C_{ploss}$ [%]</td>
<td>25.99</td>
<td>18.83</td>
<td>49.92</td>
<td>48.94</td>
<td>44.01</td>
</tr>
<tr>
<td>First bend separation [m]</td>
<td>–</td>
<td>–</td>
<td>0.317</td>
<td>0.339</td>
<td>0.330</td>
</tr>
<tr>
<td>Second bend separation [m]</td>
<td>0.604</td>
<td>0.639</td>
<td>0.586</td>
<td>0.601</td>
<td>0.602</td>
</tr>
</tbody>
</table>

Figure 9. Pressure loss coefficient at engine face predicted by UNS3D using grid: (a) C1, (b) C2, (c) L2 and (d) F3.
A further improvement was obtained by adding more points in the radial direction (compare grids F3 and C2).

Figure 10 shows three grids that were used to assess the effect of the grid size in the \( z \)-direction on the solution. The discretisation of the crosswise layers (\( z \)-planes) was done using the F3 mesh because this mesh generated the closest numerical results to the experimental data, as shown in Table 3. The meshes shown in Figure 10 had 87, 134 and 169 layers in the \( z \)-direction. The locations of the two separation points and the pressure loss at the engine face are shown in Table 4. The contour plots of the pressure loss coefficient at the engine face are shown in Figure 11. Both Table 4 and Figure 11 show that the numerical results are converging toward the experimental data as the number of layers in the \( z \)-direction is increased. The grid with 134 layers was considered the best compromise between accuracy and computational cost and therefore was used in the rest of the paper.

The representation of the low pressure regions at the engine face plane as counter-rotating vortices is shown in Figure 12a for the FLUENT computations, Figure 12b for the UNS3D computations, and Figure 12c for the experimental results. These vector plots echo the above discussion regarding the size and shape of the second bend vortices. Additionally, analysing the lower portion of the plots reveals the state of the first bend vortices at the engine face plane. The vector plot created by the FLUENT flow solver shows no rotation over the lower region of the plane. However, the vectors from the experimental data portray some vorticity remaining from the first bend, but not nearly with the strength or structure that UNS3D predicts.

Figure 13 shows the streamlines generated by equidistant seeds placed at the inlet of the duct. The streamlines that start close to the corners of the duct inlet face, where the vorticity is stronger, are pushed under the other streamlines by the secondary flow vorticities \( \Omega_{RS} \) and \( \Omega_{LS} \) shown in Figure 1. Because of the secondary flow vorticity the two corner streamlines bend more than the rest of streamlines and migrate towards the symmetry plane (\( y-z \)) of the duct. These two streamlines then enter the separation bubble downstream of the first bend, turn first toward the symmetry plane (\( y-z \)) of the duct and then turn toward inlet face. As the two streamlines leave the separation bubble they turn sharply toward the exit and then they barely turn as they pass through the second bend. The two streamlines exit the duct close to the central lower part of the engine face.

### 4.2. Static pressure on upper and lower walls along symmetry plane

The surface static pressure is an indicator of the regions where flow accelerates and decelerates near the wall. Surface static pressure also provides information on flow separation. The variation of the static pressure distribution along the centreline of the inlet duct is shown in Figure 14 for the CFD results and experimental data.

Figure 14 shows that the flow separation locations predicted by both CFD solvers are comparable to the experiment. However, it can be inferred that the numerically predicted separations are not as large as that experienced during experimental testing, especially at the first bend. This discrepancy is likely a result of the turbulence modelling. Both CFD solvers appear to correctly capture the trends in the

<table>
<thead>
<tr>
<th>Number of layers in ( z ) direction</th>
<th>87</th>
<th>134</th>
<th>169</th>
<th>Experiment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure loss, ( C_{ploss} ) [%]</td>
<td>40.61</td>
<td>48.94</td>
<td>47.87</td>
<td>44.01</td>
</tr>
<tr>
<td>First bend separation [m]</td>
<td>0.325</td>
<td>0.339</td>
<td>0.332</td>
<td>0.330</td>
</tr>
<tr>
<td>Second bend separation [m]</td>
<td>0.609</td>
<td>0.601</td>
<td>0.603</td>
<td>0.602</td>
</tr>
</tbody>
</table>

Figure 10. UNS3D grids with (a) 87, (b) 134 and (c) 169 \( z \)-layers.
near-wall flow velocity. The only portion of the duct inaccurately modelled is near the first bend, where UNS3D, and to a greater extent FLUENT, predict a too large local velocity.

4.3. Locations of flow separation
For further analysis, the exact locations of the separation points for each bend are included in

Figure 11. Pressure loss coefficient at the engine face: (a) 87, (b) 134 and (c) 169 z-layers.

Figure 12. Velocity vectors at the engine face: (a) FLUENT, (b) UNS3D and (c) experimental.
Table 5. The values in this table are measured from the entrance of the inlet duct. UNS3D predicts the separation points very accurately, being off by 2.67% at the first bend and 0.17% at the second bend. FLUENT gives poor calculations of the separation locations. At the first bend, the difference is 22.05%, whereas the error at the second bend is moderately better at 1.83%.
4.4. Symmetry plane axial velocity

Figure 15 shows contour plots of the axial velocity at the symmetry plane for the FLUENT and UNS3D simulations. Both plots reveal acceleration of the flow as it passes each bend of the duct, followed by areas of rapid deceleration and flow separation. It should be noted that FLUENT predicts a smaller region of separation at the first bend and larger region at the second bend than UNS3D does. This difference accounts for the discrepancies at the engine face plane discussed earlier, especially for the lower portion of the duct. Unfortunately, there are currently no experimental data to validate either solution.

5. Conclusions

This article presented a computational and experimental study of the development of secondary flow phenomena within a compact, serpentine inlet duct. Results of laboratory testing, CFD analysis by a commercial code (FLUENT), and CFD analysis by an in-house code (UNS3D) were compared.

Qualitatively, there was good agreement between the CFD results of both codes and the experimental data. All three data sets revealed the presence of two counter-rotating vortices that covered much of the upper portion of the engine face plane. These structures were shed from a location just downstream of the second bend separation. There was disagreement between the three methods regarding the vortices shed from the first bend. The experimental results suggested that the first bend vortices diffuse and migrate towards the second bend vortices, whereas the FLUENT solver showed them flattened near the lower wall and the UNS3D solver kept the vortices intact with little diffusion or migration.

Overall, the UNS3D results were closer to the experimental data than FLUENT results. Note, however, that although both solvers used the same governing equations and turbulence model, the numerical algorithms were different. Furthermore, the grids used for the two flow solvers were different, although the values of maximum $y^+$ numbers were close. In addition, the effect of grid size on the flow solution was investigated only for the UNS3D solver. The number of grid points of the mesh that offered the best compromise between accuracy and computational time for the UNS3D solver was then increased by approximately 20% and used for FLUENT. It is possible, however, that this grid is not the most appropriate for FLUENT. Furthermore, the quality of the FLUENT grid deteriorated at the engine face near the intersection of the symmetry plane with the duct walls. These two aspects could explain why FLUENT results did not match experimental results as well as UNS3D.

When considering the area-averaged pressure loss coefficient, the UNS3D solution differed from experiment by 11.23% compared with 52.61% for FLUENT. The locations of flow separation were also more accurately simulated by UNS3D. At the first bend, UNS3D was off the correct position by less than 0.009 m downstream, whereas FLUENT erred by 0.073 m downstream. For the second bend location, both flow solvers predicted flow separation slightly too far upstream. Here, UNS3D was inaccurate by only 0.001 m and FLUENT was off by 0.011 m.

Much of the differences between the experimental and numerical results can be attributed to the turbulence model used. For a flow field as complex as the one within the compact S-duct, shortcomings in

<table>
<thead>
<tr>
<th>Table 5. Separation locations for each bend, measured from the duct entrance.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td>-------------------------------------------------</td>
</tr>
<tr>
<td>First bend</td>
</tr>
<tr>
<td>Second bend</td>
</tr>
</tbody>
</table>

Figure 15. Axial velocity at symmetry plane: (a) FLUENT and (b) UNS3D.
the modelling become apparent. A current effort is underway at Texas A&M University and Lockheed Martin to use Particle Image Velocimetry data to mend current turbulence models and possibly develop new ones for this type of wall-bounded flow. Once this task is complete, another study similar to that presented in this article should be initiated to validate the model.

Acknowledgements
This work was sponsored by AFOSR under Grants No. FA9550-04-1-0166 and FA9550-07-1-0187. The authors are thankful to the Texas A&M University Supercomputing Facility for making the computing resources available. The authors would like to thank Daniel Miller, Neal Domel and Philip Truax of Lockheed Martin Aeronautics Company for providing the serpentine inlet duct used in this study. The authors would like thank Mr. Abhinav Kumar, a former graduate student in the Department of Aerospace Engineering at Texas A&M, for generating the FLUENT® results.

References