1 Introduction

Computational Fluid Dynamics (CFD) has gained great importance as an aero/hydrodynamic analysis and design tool in recent years. CFD simulations with different levels of fidelity, ranging from linear potential flow solvers to full Navier-Stokes codes, are widely used in the multidisciplinary design and optimization (MDO) of advanced aerospace and ocean vehicles. Although low-fidelity CFD tools have low computational cost and easy implementation, the full viscous equations are needed for the simulation of complex turbulent separated flows, which occur in several practical cases such as high-angle-of attack aircraft, high-lift devices, maneuvering submarines and missiles.\(^1\) Even for cases when there is no flow separation, the use of high-fidelity CFD simulations is desirable for obtaining higher accuracy. Due to modeling, discretization and computation errors, the results obtained from CFD simulations have a certain level of uncertainty. It is important to understand the sources of CFD simulation errors and their magnitudes in order to be able to assess the magnitude of the uncertainty in the results.

The objective of the proposed paper is to illustrate different sources of uncertainty in CFD simulations, by careful study of several examples. We will try to compare the magnitude and importance of each source of uncertainty.

\(^\text{1}\)Graduate student, Department of Aerospace and Ocean Engineering, Student Member AIAA
\(^\text{1}\)Professor and Department Head, Department of Aerospace and Ocean Engineering, Fellow AIAA
\(^\text{4}\)Professor, Department of Aerospace Engineering, Mechanics and Engineering Science, University of Florida, Gainsville, FL, Fellow AIAA
\(^\text{5}\)Professor, Department of Aerospace and Ocean Engineering, Associate Fellow AIAA
\(^\text{b}\)Professor, Departments of Computer Science and Mathematics
To have a better understanding of the accuracy in CFD simulations, the main sources of errors and uncertainties should be identified. Oberkampf and Blottner\textsuperscript{2} classified CFD error sources. In their classification, the error sources are grouped under four main categories: (1) physical modeling errors, (2) discretization and solution errors, (3) programming errors, and (4) computer round-off errors.

Physical modeling errors originate from the inaccuracies in the mathematical models of the physics. The errors in the partial differential equations (PDEs) describing the flow, the auxiliary (closure) physical models and the boundary conditions for all the PDEs are included in this category. Turbulence models used in viscous calculations are considered as one of the auxiliary physical models, usually the most important one. They are used for modeling the additional terms originated as the result of Reynolds averaging, which in itself is a physical model.

Oberkampf and Blottner\textsuperscript{2} define discretization errors as the errors caused by the numerical replacement of PDEs, the auxiliary physical models and continuum boundary conditions by algebraic equations. Consistency and the stability of the discretized PDEs, spatial (grid) and temporal resolution, errors originating from the discretization of the continuum boundary conditions are listed under this category. The difference between the exact solution to the discrete equations and the approximate (or computer) solution is defined as the solution error of the discrete equations. Iterative convergence error of the steady-state or the transient flow simulations is included in this category. A similar description of the discretization errors can also be found in Roache.\textsuperscript{3,4}

Since the terms \textit{error} and \textit{uncertainty} are commonly used interchangeably in many CFD studies, it will be useful to give a definition for each. Uncertainty, itself, can be defined in many forms depending on the application field as listed in DeLaurentis and Mavris.\textsuperscript{5} For computational simulations, Oberkampf \textit{et al.}\textsuperscript{6-7} described uncertainty as a potential deficiency in any phase or activity of modeling process that is due to the lack of knowledge, whereas \textit{error} is defined as a recognizable deficiency in any phase or activity of modeling and simulation.

Considering these definitions, any deficiency in the physical modeling of the CFD activities can be regarded as \textit{uncertainty} (such as uncertainty in accuracy of turbulence models, uncertainty in the geometric dimensions, uncertainty in thermophysical parameters \textit{etc.}), whereas the deficiency associated with the discretization process can be classified as \textit{error}.\textsuperscript{7}

Discretization errors can be quantified by using methods like Richardson’s extrapolation or grid-convergence index (GCI), a method developed by Roache\textsuperscript{4} for uniform reporting of grid-convergence studies. However, these methods require fine grid resolution in the asymptotic range, which may be hard to achieve in the simulation of flow-fields around complex geome-
tries. Also non-monotonic grid convergence, which may be observed in many flow simulations, prohibits or reduces the applicability of such methods. That is, it is often difficult to estimate errors in order to separate them from uncertainties. Therefore, for the rest of the paper, the term uncertainty will be used to describe the inaccuracy in the CFD solution variables originating from discretization, solution, or physical modeling errors.

In this extended abstract, the results of a steady, two-dimensional, turbulent, transonic flow in a converging-diverging channel obtained with the General Aerodynamic Simulation Program (GASP) are presented. Runs were performed with different turbulence models, grid levels and flux-limiters in order to see the effect of each to the CFD simulation uncertainties. In the proposed paper, additional cases will be included to generalize the results. Also, the uncertainty levels will be compared to the magnitude of uncertainties due to typical manufacturing errors in the geometry of the system.

2 Description of the Simulation Case

The test case presented in this extended abstract is the simulation of steady, 2-D, turbulent, transonic flow in a converging-diverging channel, which is also known as the Sajben Transonic Diffuser in the CFD validation studies. Figure 1 shows a schematic of the geometry and one of the grids used in the computations. The flow is from left to right, in the positive $x$-direction, and $y$-direction is normal to the bottom wall. All the dimensions are scaled by the throat height, $h_t$. The throat section, which is the minimum cross-sectional area of the channel, is located at $x/h_t = 0.0$. The bottom wall of the channel is flat and the converging-diverging section of the top wall is described by a function of $x/h_t$ defined in Bogar et al. Although the geometry is relatively simple, the flow has a complex structure. The exit-pressure-to-inlet-total pressure ratio $P_e/P_{0i}$ sets the strength and the location of the shock. In our studies, we term $P_e/P_{0i} = 0.72$ as the strong shock case and $P_e/P_{0i} = 0.82$ as the weak shock case. A separated flow region exists just after the shock at $P_e/P_{0i} = 0.72$. A large set of experimental data for a range of $P_e/P_{0i}$ values is available. In our study, top and bottom wall pressure values were used for the comparison of CFD results with the experiment.

GASP is a three-dimensional, finite-volume, Reynolds-averaged Navier-Stokes code, which is capable of solving steady-state and time-dependent problems. For this problem, the inviscid fluxes were calculated by upwind-biased 3rd-order spatially accurate Roe flux scheme. The minimum modulus (Min-Mod) and Van Albada’s flux limiters were used in order to prevent non-physical oscillations in the solution. All the viscous terms were included in the solution.
and two turbulence models, Spalart-Almaras (Sp-Al) and $k - \omega$ (Wilcox, 1998 version), were used for modeling the viscous terms.

The iterative convergence of each solution is examined by monitoring the overall residual, which is the sum (over all the cells in the computational domain) of the L2 norm of all the governing equations solved in each cell. In addition to this overall residual information, the individual residual of each equation is also monitored.

Four different grids were used in the simulations: Grid 1 (g1), Grid 2 (g2), Grid 3 (g3), and Grid 4 (g4). The finest mesh is grid 4 and the other grids are obtained by reducing the number of divisions by a factor of 2 in both $x$ and $y$-directions at each consecutive level (grid halving). The number of mesh points for each grid can be listed as: Grid 1, $(41 \times 26 \times 2)$, Grid 2, $(81 \times 51 \times 2)$, Grid 3, $(161 \times 101 \times 2)$, and Grid 4, $(321 \times 201 \times 2)$. Grid 2 is shown in Figure 1. In order to resolve the flow gradients due to viscosity, the grid points were clustered in the $y$-direction near the top and the bottom walls. In wall bounded turbulent flows, it is important to have sufficient number of grid points in the wall region, especially in the laminar sublayer, for the resolution of the near wall velocity profile, when turbulence models without wall-functions are used. A measure of grid spacing near the wall can be obtained by examining the $y^+$ values defined as

$$y^+ = \frac{y \sqrt{\tau_w/\rho}}{\nu},$$

where $y$ is the distance from the wall, $\tau_w$, the wall shear stress, $\rho$, the density of the fluid, and $\nu$, the kinematic viscosity. In turbulent boundary layers, a $y^+$ value between 7 and 10 is considered as the edge of the laminar sublayer. General CFD practice has been to have several mesh points in the laminar sublayer with the first mesh point at $y^+ = O(1)$. In our study, the maximum value of $y^+$ values for Grid 2 and Grid 3 at the first cell center locations from the bottom wall were found to be 0.53 and 0.26 respectively. The grid points were also stretched in the $x$-direction to increase the grid resolution in the vicinity of the shock wave. The center of the clustering in the $x$-direction was located at $x/h = 2.24$. At each grid level, except Grid 1, the initial conditions were obtained by interpolating the primitive variable values of the previous grid solution to the new cell locations. This method, known as grid sequencing, was used to reduce the number of iterations required to converge to a steady state solution at finer mesh levels.

It should be noted that Grid 4 is highly refined, beyond what is normally used for such two-dimensional problems. A single solution on Grid 4 required approximately 363 hours of total node CPU time on an SGI Origin2000 with 6 processors, when the L2 norm of the overall residual was allowed to drop 5 orders of magnitude. If we consider a three-dimensional case,
with the addition of another dimension to the problem, Grid 2 would usually be regarded as a fine grid, whereas Grid 3 and Grid 4 would generally not be used.

3 Results and Discussion

For the transonic flow in the converging-diverging channel, the uncertainty of the CFD simulations are investigated by examining the nozzle efficiency \( n_{eff} \) as a global output quantity obtained at different \( P_e/P_{0i} \) ratios with different grids (g1, g2, g3 and g4), flux limiters (Min-Mod and Van Albada), and turbulence models (Sp-Al and \( k-\omega \)). \( n_{eff} \) is defined as

\[
n_{eff} = \frac{H_{0i} - H_e}{H_{0i} - H_{es}},
\]

where \( H_{0i} \) is total enthalpy at the inlet, \( H_e \), the enthalpy at the exit, and \( H_{es} \), the exit enthalpy at the state that would be reached by isentropic expansion to the actual pressure at the exit. Since the enthalpy distribution at the exit was not uniform, \( H_e \) and \( H_{es} \) were obtained by integrating the cell-averaged enthalpy values across the exit plane. Besides \( n_{eff} \), wall pressure values from the CFD simulations are compared with experimental data. In addition to the visual assessment of the graphs, the comparison with the experiment is also performed quantitatively by introducing a measure of the error between the experiment and the curve representing the CFD results, the orthogonal distance error

\[
E_n = \frac{\sum_{i=1}^{N_{exp}} d_i}{N_{exp}},
\]

where

\[
d_i = \min_{x_{inlet} \leq x \leq x_{exit}} \left\{ \sqrt{(x - x_i)^2 + (P_e(x) - P_{exp}(x_i))^2} \right\}.
\]

In equations (3) and (4), \( d_i \) represents the orthogonal distance between the \( i \)th experimental point and the \( P_e(x) \) curve (the wall pressure obtained from the CFD calculations), \( P_{exp} \) is the experimental wall pressure value, and \( N_{exp} \) is the total number of experimental data points. Pressure values are scaled by \( P_{0i} \) and the \( x \) values are scaled by the length of the channel.

3.1 Comparison of the results with the experiment

The quantitative comparison of CFD simulation results with the experiment can be achieved by considering different measures of error. In this example case, \( E_n \) was used to approximate the difference between the wall pressure values obtained from the numerical simulations and the experimental data. Figure 2 shows the top wall \( E_n \) values of the strong and the weak shock case.
calculated from the results obtained with different grids, turbulence models, and flux-limiters. The maximum orthogonal distance error \((E_n)_{\text{max}}\) is seen at the strong shock case with Grid 4, Min-Mod limiter, and \(k-\omega\) model. \(E_n\) takes its minimum value at the weak shock case with Grid 2 and Sp-Al model. By scaling \(E_n\) as

\[
\hat{E}_n = \frac{E_n}{(E_n)_{\text{max}}} \times 100, \tag{5}
\]

we can replace the orthogonal distance error with its percentage value of \((E_n)_{\text{max}}\) for each case. These values are listed in Table 1. Each row of Table 1 gives a measure of the difference between CFD results and the experiment for the cases with different limiters and turbulence models at each grid level.

\[
\begin{array}{|c|c|c|c|c|c|}
\hline
\frac{P_e}{P_0} & \text{Grid} & \text{Sp-Al, Min-Mod} & \text{Sp-Al, Van Albada} & k-\omega, \text{Min-Mod} & k-\omega, \text{Van Albada} \\
\hline
0.72 & g1 & 85.3 & 75.1 & 90.3 & 84.0 \\
0.72 & g2 & 63.2 & 61.6 & 91.8 & 87.2 \\
0.72 & g3 & 52.1 & 51.9 & 92.0 & 91.4 \\
0.72 & g4 & 47.1 & 47.2 & 100.0 & 98.2 \\
0.82 & g1 & 50.8 & 47.7 & 41.3 & 43.5 \\
0.82 & g2 & 36.7 & 36.7 & 40.5 & 40.6 \\
0.82 & g3 & 42.9 & 42.3 & 39.3 & 39.7 \\
0.82 & g4 & 45.9 & 45.5 & 37.3 & 37.2 \\
\hline
\end{array}
\]

At the strong shock case, the minimum difference between the numerical results and the experimental data is obtained with Sp-Al model and Min-Mod limiter on Grid 4, which is 47.1% of the \((E_n)_{\text{max}}\). The maximum difference between CFD results and the experiment for the weak shock case is much smaller than that of the strong shock case. It is 50.8% of the \((E_n)_{\text{max}}\) and obtained on Grid 1 with Sp-Al model and Min-Mod limiter. In contrast to the strong shock case, the minimum value of the difference is seen on Grid 2, not at the finest mesh level, however it is again obtained with Sp-Al turbulence model. This minimum value, 36.7% of \((E_n)_{\text{max}}\), is also the smallest difference between the CFD results and the experiment among all the cases.

In this extended abstract, the difference in the wall pressure values between CFD and the experiment is not considered as a measure of the total uncertainty in CFD simulations due to existence of the uncertainty in the experimental data. This uncertainty in the experimental
data may originate from many factors such as geometric irregularities, difference between the
actual \( P_e/P_{in} \) and its intended value, measurement errors, and other factors, such as heat
transfer to the fluid. For example, any difference between the actual shape of the channel
upper wall and its analytical description, which is also used in the grid generation, may cause
changes in the wall pressure values. With the existence of uncertainty due to different factors,
the experimental results represent a fuzzy band, and the difference between the numerical
results and the experiment originate both from the inaccuracy of the CFD simulations and the
uncertainty of the experimental data. In the proposed paper, we will try to approximate the
order of the experimental uncertainties by studying the effect of different sources. With this
information, the quantification of the total uncertainty in CFD simulations would be possible
and the contribution of each source to the overall uncertainty would be determined.

The relative uncertainty between each case with different grids, limiters, and turbulence mod-
els can be examined in three groups: (1) uncertainty due to the discretization error, (2) the
uncertainty due the turbulence model, and (3) uncertainty due to the iterative convergence.

3.2 Uncertainty due to the discretization error

In order to investigate the uncertainty due to the discretization error, we study the Sp-Al and
\( k - \omega \) cases separately. For each case with a different turbulence model, grid level and the
flux-limiter affect the magnitude of the discretization error. Grid level determines the spatial
resolution, and the limiter is part of the discretization scheme which reduces the spatial accuracy
of the method to first order in the vicinity of shock waves.

In Table 1, we examine each individual column to assess the contribution of grid resolution
to the discretization uncertainty. For the strong shock case with Sp-Al model, the numerical
wall pressure values approach the experiment as the mesh is refined. This can also be seen by
visual inspection of the wall pressure distributions given in Figure 4. However, the difference
between each grid level is significant. For example, in Sp-Al model and Min-Mod limiter case,
the difference in \( \hat{E}_n \) between Grid 2 and Grid 4 is 16% percent of \((E_n)_{max}\). In contrast to
Sp-Al model, the difference between the numerical wall pressure values and the experimental
data increase with the refinement of mesh size in the \( k - \omega \) case. However, the difference in
\( \hat{E}_n \) between Grid 2 and Grid 4 is smaller. For the \( k - \omega \) and Min-Mod case, this difference
is 8% of the \((E_n)_{max}\). Similar observations can be made on nozzle efficiency results shown in
Figure 3. For each turbulence model, the relative uncertainty between the results of Grid 2 and
Grid 4 are significant. The largest value of this difference is observed for the Sp-Al case with
the Min-Mod limiter, which is approximately 6%.
For the weak shock case with the Sp-Al model, the smallest $\hat{E}_n$ is obtained with Grid 2, and results tend to deviate from the experiment as the mesh is refined (Table 1 and Figure 5). The difference in $\hat{E}_n$ between Grid 2 and Grid 4, which is $\%9$ of $(E_n)_{max}$, is still significant. For the $k - \omega$ model, the lowest $\hat{E}_n$ value is obtained with Grid 4, the finest mesh level. However, the difference between each grid level is not as big as that of the Sp-Al case. In fact, the wall pressure distribution of each grid level is very close to each other (Figure 5). The $\hat{E}_n$ difference between Grid 2 and Grid 4 is $3\%$ percent of $(E_n)_{max}$.

Figure 6 again shows the significance of the discretization uncertainty between each grid level. In this Figure, the noisy behavior of $n_{eff}$ results obtained with Grid 1 can be seen for both turbulence models. The order of the noise error is much smaller than the discretization error between each grid level, however this can be a significant source of uncertainty if the results of grid 1 are used in a gradient-based optimization.

The uncertainty due to the choice of the limiter can be seen in the results of Grid 1 and Grid 2 for the strong shock case and Grid 1 for the weak shock case. In Table 1, the maximum difference in $\hat{E}_n$ between the Min-Mod limiter and Van Albada limiter occurs on Grid 1 with Sp-Al model. This difference is $10.2\%$ of $(E_n)_{max}$ for the strong shock case, and $3.1\%$ for the weak shock case. In nozzle efficiency results (Figure 3), similar effect of the limiter can be observed. A relative uncertainty of $2\%$ in $n_{eff}$ results between the Min-Mod and Van Albada limiter at Grid level 2 with Sp-Al model can be detected. The uncertainty due to the choice of the limiter is more significant for the strong shock case. For both pressure ratios, the results obtained with different limiters have almost the same value, when Grid 3 and Grid 4 are used, which would hardly be the case in practice due their computational expense.

From nozzle efficiency results, it can be seen that the asymptotic grid convergence is not achieved at any case, although the difference between each grid level is reduced with the mesh refinement except the $k - \omega$, strong shock case, which exhibits a non-monotonic behavior. The lack of grid resolution in the asymptotic range and the non-monotonic behavior of the solution variables at different grid levels prohibit the use of conventional error estimate methods for the quantification of the discretization error.

When we look at Mach number values at two points; one, upstream of the shock ($x/h_t = -1.5$) and the other, downstream of the shock ($x/h_t = 8.65$, the exit plane), both of which are located at the mid point of the local channel heights (Figure 7), we see the convergence of Mach number upstream of the shock for all the cases. However, for the strong shock case, the lack of convergence downstream of the shock at all grid levels with the $k - \omega$ model can be observed. For the Sp-Al case, we see the convergence only at Grid levels 3 and 4. For the weak
shock case, downstream of the shock, the convergence at all grid levels with the $k - \omega$ model is also seen. At this pressure ratio, Sp-Al model results do not seem to converge, although the difference between each grid level is small. These results may indicate the effect of the complex flow structure downstream of the shock, especially the separated flow region seen in the strong shock case, on the grid convergence.

### 3.3 Uncertainty due to the turbulence models

Since the asymptotic grid convergence is not achieved in any case, isolating the uncertainty due to the turbulence model from the discretization uncertainty is a difficult task. However, nozzle efficiency results and the orthogonal distance error values can be used to approximate the relative uncertainty at each grid level due to the selection of the turbulence model.

From the nozzle efficiency results, for the strong shock case with flow separation, it can be seen that the relative uncertainty originating from the selection of turbulence models is much bigger than the uncertainty due to the discretization errors (Figure 3). At $P_e/P_{bi} = 0.72$, the difference in $n_{eff}$ between Sp-Al and $k - \omega$ turbulence models is around 9%, when grid 4 solutions are compared. As $P_e/P_{bi}$ is increased and the strength of the shock is reduced, the difference between the results of each turbulence model get smaller (Figure 6) and relative uncertainty due to the discretization error becomes comparable to the one due to the selection of turbulence model. Similar conclusions can be drawn from the orthogonal distance error distributions. In table 1, the difference in $\hat{E}_n$ between Sp-Al and $k - \omega$ model is approximately 50% of $(E_n)_{max}$ at Grid level 4. This is much bigger than the discretization uncertainty between Grid 2 and 4 obtained with each turbulence model. On the other hand, at $P_e/P_{bi} = 0.82$, $\hat{E}_n$ difference is 8% of $(E_n)_{max}$ between Sp-Al and $k - \omega$ models at Grid level 4, which is much smaller than that of the strong shock case. This value is again comparable to the discretization uncertainty obtained between Grid levels 2 and 4 at this pressure ratio.

### 3.4 Uncertainty due to the iterative convergence

Figures 8 and 9 show the convergence history of the L2 norm residual of the energy equation for the strong shock case obtained with Sp-Al and $k - \omega$ turbulence models. The convergence history of the residual, normalized by its initial value, is presented for all the limiters and the grids used in the study. By examining these figures, it can be seen that the main parameter that affects the residual convergence of a solution is the flux-limiter. With the Min-Mod limiter, the residuals of Grid 2, Grid 3, and Grid 4 do not reach even one order of magnitude reduction while
the same grid levels show much better residual convergence when the Van Albada limiter is used. For example, the residual of the Sp-Al case with Grid 3 was reduced more than seven orders of magnitude and the residual of the same grid with the $k - \omega$ model was dropped approximately nine orders of magnitude when 10000 cycles were run with the Van Albada limiter. The same convergence behavior of the Min-Mod and the Van Albada limiter was observed for the residual of the other equations and the weak shock case.

Although the use of Min-Mod limiter causes poor L2 norm residual convergence, this does not seem to effect the final results, such as the wall pressure values and the nozzle efficiencies. When we compare the orthogonal error values of the Min-Mod and the Van Albada limiters obtained with the same turbulence model at the grid level g3 in Figure 2, we see that the error values are almost the same even though the residual convergence between each case is significantly different. The visual comparison of the wall pressure distributions with the experimental data also verifies this result. Furthermore, if we compare the nozzle efficiency results (Figure 3) obtained at grid levels g3 and g4 with the same turbulence model, the values for different limiters are again very close to each other, suggesting no effect of the difference observed in the residual convergence.

4 Conclusions

In this extended abstract, different sources of uncertainty in the CFD simulations are demonstrated by examining a steady, 2-D, turbulent, transonic flow in a converging-diverging channel at various $P_e/P_{in}$ ratios by using the commercial CFD code GASP. Runs were performed with different turbulence models (Sp-Al and $k - \omega$), grid levels (g1, g2, g3, and g4) and flux-limiters (Min-Mod and Van Albada) in order to see the effect of each to the CFD simulation uncertainties.

The difference in the wall pressure values between CFD and the experiment was determined by calculating the orthogonal distance error $E_n$ for each case. The maximum difference is seen at the strong shock case with Grid 4, Min-Mod limiter, and $k - \omega$ model. The minimum difference was obtained at the weak shock case with Grid 2 and Sp-Al model. In this extended abstract, the difference in the wall pressure values between CFD and the experiment is not considered as a measure of the total uncertainty in CFD simulations due to existence of the uncertainty in the experimental data. With the uncertainty due to different factors, the experimental results represent a fuzzy band, and the difference between the numerical results and the experiment originate both from the inaccuracy of the CFD simulations and the uncertainty of the experi-
mental data. In the proposed paper, we will try to isolate the experimental uncertainty in the wall pressures by studying the effect of different factors. With this information, we will be able to approximate the magnitude of total simulation uncertainty in the wall pressure results.

The nozzle efficiencies and the orthogonal distance error values were used to examine the relative uncertainties in the results obtained with different grids, turbulence models, and limiters. For low pressure ratios, the flow-field has a complex structure with a fairly large separation region and the dominant source of relative uncertainty is the turbulence model. For high pressure ratios, the flow-field is simpler and in this case relative uncertainty originating from grid selection becomes comparable to the one due to the selection of the turbulence models.

The effect of the limiter on the discretization uncertainty is seen at Grid levels 1 and 2, especially for the strong shock case. For both pressure ratios, the results obtained with different limiters have almost the same value, when Grids 3 and 4 are used. However these grid levels are highly refined, and would generally not be considered in practice.

Although the use of Min-Mod limiter causes poor L2 norm residual convergence, this does not seem to effect the final results, such as the wall pressure values and the nozzle efficiencies.

In the final version of the paper, additional test cases will be included in order generalize the conclusions for different geometries and flows. Also, the effect of the geometric imperfections on the output variables will be examined to simulate the manufacturing uncertainties and the results will be contrasted with the magnitude of the uncertainty sources considered here.

Acknowledgements

This research was supported by the National Science Foundation grant DMI-9979711.
References


Figure 1: Sajben Transonic Diffuser geometry and Grid2 (g2) used in the computations.

Figure 2: $E_n$ for the top wall pressure distributions obtained at different grid levels with Sp-Al and k-w turbulence models, Min-Mod and Van Albada limiters.

Figure 3: $n_{eff}$ obtained at different grid levels with Sp-Al and k-w turbulence models, Min-Mod and Van Albada limiters.
Figure 4: Top wall $P/P_0$ vs. $x/h_t$ distributions obtained with Sp-Al and k-w turbulence models, Van Albada limiter, and grids g2 and g4 at $P_e/P_{0i} = 0.72$.

Figure 5: Top wall $P/P_0$ vs. $x/h_t$ distributions obtained with Sp-Al and k-w turbulence models, Van Albada limiter, and grids g2 and g4 at $P_e/P_{0i} = 0.82$. 
Figure 6: $n_{eff}$ vs. $P_e/P_{0i}$ for different grids obtained with Sp-Al and k-w turbulence models, and Min-Mod limiter.

Figure 7: Mach number values at the upstream of the shock ($x/h_t = -1.5$), and downstream of the shock ($x/h_t = 8.65$, the exit plane) for different grids obtained with Sp-Al and k-w turbulence models, Min-Mod and Van Albada limiters. The values of $y/h_t$ correspond to the mid points of the local channel heights.
Figure 8: Normalized L2 Norm residual of the energy equation for the case with Sp-Al turbulence model, Van Albada, and Min-Mod limiters at $P_e/P_{0i} = 0.72$. Normalization is done with the initial value of the residual.

Figure 9: Normalized L2 Norm residual of the energy equation for the case with $k - \omega$ turbulence model, Van Albada, and Min-Mod limiters at $P_e/P_{0i} = 0.72$. Normalization is done with the initial value of the residual.