

Numerical Simulations of an Axisymmetric Shock-Wave/Boundary-Layer Interaction

David J. Friedlander¹, Nicholas J. Georgiadis², Layton W. Howerton³,
and Heath H. Reising⁴

NASA Glenn Research Center, Cleveland, OH, 44135

A series of Reynolds-averaged Navier-Stokes (RANS) simulations were performed of two Mach 2.5 axisymmetric shock-wave/boundary-layer interactions (SWBLIs) based off the NASA Turbulent Computational Fluid Dynamics Validation Experiment (TCFDVE). The goals of the simulations were to (1) determine how well RANS can predict the flow physics of an axisymmetric SWBLI and (2) generate a new validation case for the NASA Turbulence Modeling Resource (TMR). The simulations were run with two different turbulence models and three different computational fluid dynamics (CFD) codes while looking at two different conical shock generator angles, one of which was shown to cause flow separation in the experiment. The results in this paper show that, as expected, the CFD simulations agreed well with each other and the experiment when SWBLI-induced separation was not present. When separation was present, differences in the predicted separation characteristics were noted in comparison to experimental measurements. These included a stronger interaction and subsequently longer separation extent for the Menter Shear-Stress Transport $k-\omega$ (SST) turbulence model simulations compared to both the Spalart-Allmaras (SA) turbulence model simulations and the experiment.

Nomenclature

$C_{f,x}$	=	skin friction coefficient
H	=	shape factor
k	=	turbulent kinetic energy
L	=	separation length
N	=	number of grid computational points
p, p_t	=	static and total pressure
R	=	test section radius
T, T_t	=	static and total temperature
u	=	streamwise velocity
v	=	radial velocity
x, y, z	=	Cartesian coordinates
y^+	=	non-dimensional wall distance
δ	=	boundary-layer thickness based on $0.995 u_\infty$
δ^*	=	boundary-layer displacement thickness
θ	=	boundary-layer momentum thickness
μ	=	viscosity
τ	=	wall shear stress
ω	=	specific dissipation rate

¹ Aerospace Engineer, Inlets and Nozzles Branch, d.j.friedlander@nasa.gov, AIAA Senior Member

² Aerospace Engineer, Inlets and Nozzles Branch, georgiadis@nasa.gov, AIAA Associate Fellow

³ Student Trainee (Engineering), Inlets and Nozzles Branch, layton.w.howerton@nasa.gov, AIAA Student Member

⁴ Aerospace Engineer, Inlets and Nozzles Branch, heath.reising@nasa.gov, AIAA Senior Member

Subscripts

e	=	boundary-layer edge
end	=	end of separation region
i	=	inviscid shock impingement
$start$	=	start of separation region
t	=	total condition
$turb$	=	turbulent
$wall$	=	test section wall
1	=	experimental incoming boundary-layer station
∞	=	freestream

Special Notations

$\overline{(\quad)}$	=	Reynolds average
$\widetilde{(\quad)}$	=	Favre average
$(\quad)'$	=	Reynolds fluctuation
$(\quad)''$	=	Favre fluctuation

I. Introduction

SHOCK-WAVES are pervasive to all aerospace vehicles flying near or past the speed of sound. The interaction of the shock-waves with the boundary-layer is a complex flow phenomenon and frequently causes separated flow in the vicinity of the shock-wave/boundary-layer interaction (SWBLI). Mixed-compression inlets are often found in supersonic aircraft propulsion systems and utilize shock-waves to slow the air down from supersonic to subsonic speeds. SWBLIs in such inlets and the resultant flow separation causes total pressure loss, distortion, and unsteadiness that is not captured well by standard computational methods and is a concern for supersonic aircraft propulsion system designs. To that end, a workshop was held in conjunction with the 48th AIAA Aerospace Sciences Meeting to gauge the ability of computational fluid dynamics (CFD) simulations to correctly capture nominally 2D SWBLIs. The workshop utilized two experimental datasets [1-2], with the CFD results summarized in Debonis et al. [3] and additional analysis of results documented by Benek [4-5] and Hirsch [6]. One of the key conclusions of the workshop was the need for SWBLI experimental datasets that did not include side walls. The reason for not wanting side walls is because their presence in two-dimensional ducts induce corner flows that influence the centerline SWBLI to become 3D-like instead of 2D-like, as documented in Babinsky et al. [7] and Titchener et al. [8].

With regards to relevant high-quality CFD validation experimental datasets, Settles and Dodson [9] compiled a list covering supersonic and hypersonic SWBLIs. Many of the identified candidate datasets were rejected for one of the following reasons:

- lack of well-defined boundary conditions
- undocumented 3D effects
- inconsistency between repeated measurements
- lack of repeated measurements.

Because of the need for high-quality SWBLI experimental datasets available, researchers at the NASA Glenn Research Center performed a Mach 2.5 SWBLI experiment within a circular test section of the NASA Glenn Research Center's 225 cm² Wind Tunnel [10-12]. The experiment, referred to as the Turbulent Computational Fluid Dynamics Validation Experiment (TCFDVE), followed the guidance as outlined in Ref. [13] for performing CFD code validation experiments. Measurements obtained using Pitot probes, hot-wire probes, and particle image velocimetry (PIV) were used to study SWBLIs generated from three different shock generator half-angles: 10°, 13.5°, and 16°. The experiment utilized a circular test section to eliminate the influence of the corner flows on the centerline SWBLI, thus addressing one of the key conclusions from the CFD workshop.

Given the quality of the TCFDVE dataset, a series of Reynolds-averaged Navier-Stokes (RANS) simulations were performed for the 10° and 16° half-angle shock generator cases with the intention of creating a new validation case for the NASA Turbulence Modeling Resource (TMR) [14]. The TMR provides turbulence model formulations along with verification and validation cases for CFD codes to verify/validate that the turbulence models are implemented correctly. The simulations explored in this paper use two common turbulence models, specifically the one-equation Spalart-Allmaras (SA) [15] and the two-equation Menter Shear-Stress Transport $k-\omega$ (SST) [16] formulations. The

form of the SA turbulence model used in this paper is the standard formulation while the form of the SST model used uses the vorticity-based turbulent production term (SST-V). This work is independent of previous RANS simulations that were performed on this dataset [17] that were run to prepare for large eddy simulations and did not extensively examine the ability of RANS to capture the physics of the SWBLI.

II. Flow Solvers

The requirements from the TMR are that each case requires at least two CFD codes to be run and generate consistent results. The first code that was used for the simulations was the NASA's Fully Unstructured Navier-Stokes (FUN3D) code [18]. FUN3D is a node-based, production-level code developed and maintained at the NASA Langley Research Center. It can solve the 2D/3D Euler and RANS equations for incompressible and compressible flows on unstructured grids. The FUN3D flow solver was chosen because it is one of the solvers historically used when developing a validation case for the TMR.

Another CFD code that has been historically used when developing a validation case for the TMR is the Wind-US code [19]. Wind-US is a node-based, production-level code that can solve the compressible flow 2D/3D Euler and RANS equations on either unstructured or structured grids. For the simulations presented, only the structured grid solver was used.

A third code, VULCAN-CFD [20-21], was also employed. VULCAN-CFD is a cell-center-based, production-level code developed and maintained at the NASA Langley Research Center. It can solve the compressible flow 2D/3D RANS equations on unstructured or structured grids. For the simulations presented, only the structured grid solver was used.

III. Geometry and Numerical Modeling

The geometry chosen for the simulations is based off the axisymmetric configuration from the TCFDVE and is shown in Fig. 1. It consists of a cone attached to a cylinder located within a circular test section. The maximum cone diameter, as well as the cylinder diameter, is 50 mm while the diameter of the test section is 170 mm. Two cone half-angles were modeled, namely 10° and 16° . The cone/cylinder act as the shock generator and is referred to as such throughout the paper. Upstream of the shock generator is an axially symmetric convergent-divergent (CD) nozzle to accelerate the flow to the desired test section supersonic Mach number condition, which in this case is Mach 2.5. The experimental setup also included a bellmouth nozzle upstream of the CD nozzle that took in air from a plenum tank. While the experiment was performed at several Reynolds numbers, the simulations reported in this paper are for a Reynolds number based on the test section diameter of 4.0×10^6 , with the freestream reference conditions listed in Table 1.

Based on the geometry outlined in Fig. 1, a baseline grid was generated using the Pointwise grid generation software [22]. The modeled geometry for the simulations did not include the bellmouth nozzle nor the plenum tank and used a blunt cone tip. The blunt cone tip was defined by ignoring any part of the cone with a radius less than 0.05 mm. The baseline grid consists of structured-like quads on a 2D plane consisting of 556.6×10^3 grid points (554.6×10^3 grid cells) with a nominal wind tunnel wall y^+ of 0.347 and is shown in Fig. 2. This 2D plane was rotated 90° to create a 3D quarter-sector grid totaling 22.6×10^6 grid points (21.9×10^6 grid cells) to run with FUN3D and is shown in Fig. 3. Wind-US and VULCAN-CFD could use the single plane grid and perform axially symmetric calculations, resulting in lower computational costs compared to the FUN3D simulations that required a three-dimensional grid. To avoid pole domains in the 3D grid, an o-grid topology was used to define the blunt cone tip, as shown in Fig. 3b. This topology kept the number of points defining the cone tip equal to the number of azimuthal points. The blunt tip grid spacing was then extruded upstream to the computational inflow plane. The constraint on the number of grid points defining the cone tip, coupled with the radius of the shock generator, resulted in a nominal y^+ of 0.361 of the first point off the wall, based on reference conditions, for the shock generator surface. The boundary conditions for the grids, shown in Fig. 4, included a subsonic inflow (which specified total pressure and total temperature), adiabatic viscous walls for the shock generator and wind tunnel walls, and symmetry on the bottom portion of the grid upstream of the shock generator. The TCFDVE flow conditions were set with a plenum total temperature that would result in a nearly adiabatic wall state throughout the flowpath. The FUN3D simulations employed a back pressure boundary condition at the outflow that would automatically switch to an extrapolation boundary condition when the flow was

supersonic while both the Wind-US and VULCAN-CFD simulations employed an extrapolation boundary condition at the outflow. Due to the 3D-version of the grid representing a quarter sector, the use of symmetry boundary conditions at the 0° and 90° planes was employed for the FUN3D simulations.

With regards to numerical schemes, the FUN3D simulations used the low diffusion flux splitting scheme (LDFSS) [23] in conjunction with the min-mod flux limiter with a heuristic pressure limiter [24]. The Wind-US simulations used the Roe flux construction scheme [25] with the min-mod flux limiter and the VULCAN-CFD simulations used the LDFSS flux construction scheme with the min-mod flux limiter.

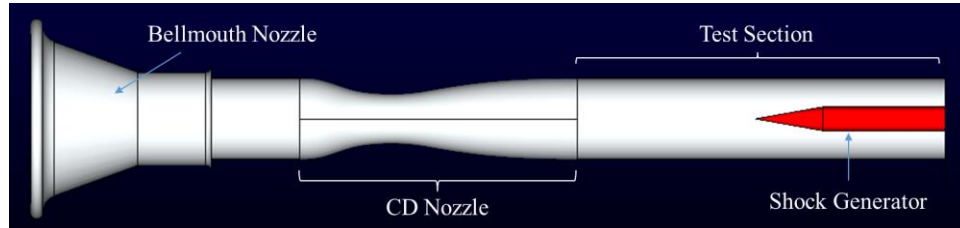


Figure 1. Notional axisymmetric SWBLI experimental setup. For simplicity, the upstream plenum tank is not shown.

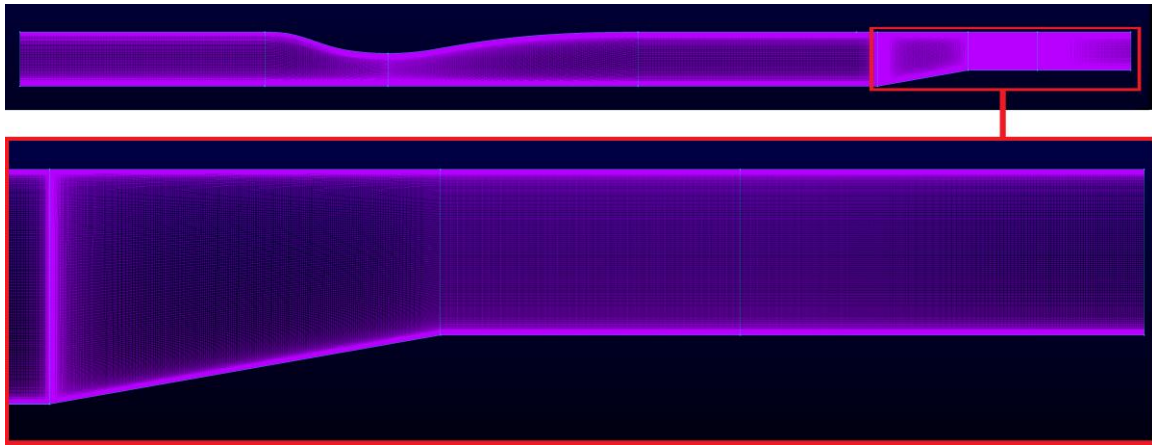


Figure 2. 2D axisymmetric SWBLI baseline grid.

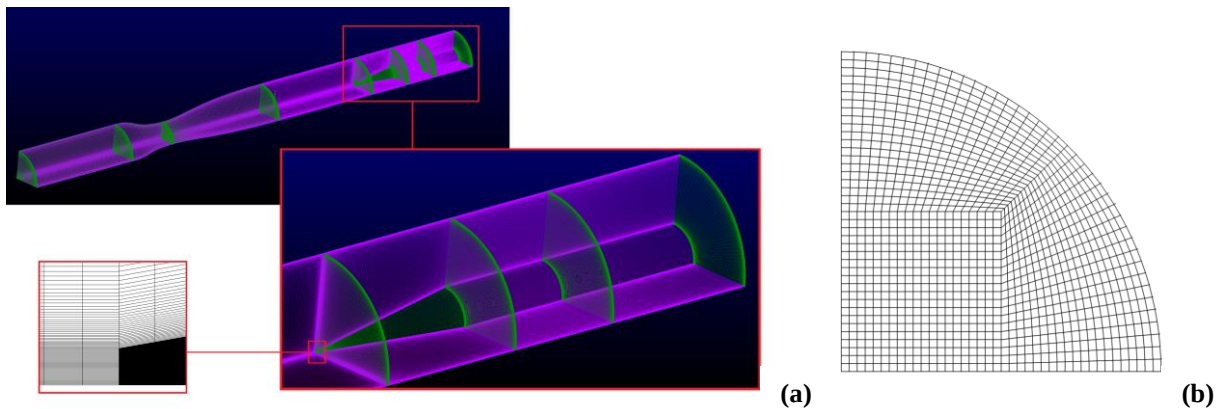


Figure 3. (a) 3D axisymmetric SWBLI baseline grid and (b) corresponding blunt cone tip grid.

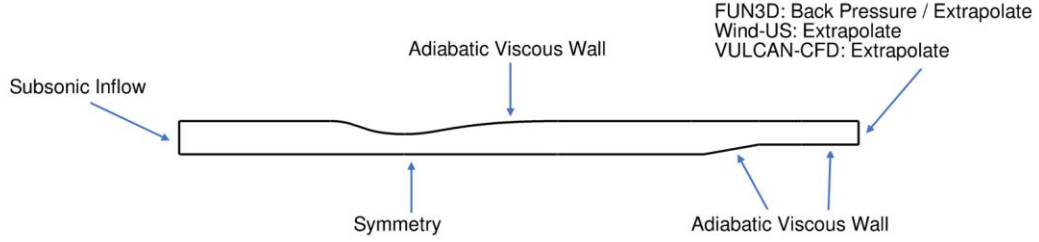


Figure 4. SWBLI grid boundary conditions.

Table 1. Axisymmetric SWBLI freestream reference conditions.

Mach Number	2.5
Reynolds Number	$23.5 \times 10^6 \text{ m}^{-1}$
Static Temperature	138.2 K
Total Temperature	310.9 K
Total Pressure	257.9 kPa

IV. Grid Study

A grid sensitivity study was performed using the Wind-US and VULCAN-CFD flow solvers. The grids used in the study were generated by coarsening the 2D baseline grid by every other point to generate a medium grid and every fourth point to generate a coarse grid. Both flow solvers utilized the grid in axisymmetric mode. The metrics used to determine grid convergence included the axial velocity profile at the experimental incoming boundary-layer station and the static pressure profile along the test section wall in the vicinity of the interaction region. The experimental data used in the comparisons are from the Pitot probe measurements.

A. Experimental Incoming Boundary-Layer Station

The experiment defined the incoming boundary-layer station to be at an axial location 139.7 mm upstream of the inviscid shock impingement location [12]. For the CFD simulations, the nominal inviscid shock impingement location of $x = 495.3 \text{ mm}$ was used, as the experiment showed variation in its location as a function of the shock generator half-angle [26]. The axial velocity profiles for the 16° half-angle shock generator cases are shown in Fig. 5 for the SA turbulence model simulations and Fig. 6 for the SST-V turbulence model simulations. The 10° half-angle shock generator cases showed similar behavior and are not shown in this subsection. From Fig. 5 and Fig. 6 it can be seen qualitatively that the axial velocity profiles appear to be grid converged on the finest grid level.

To quantify the comparisons, the integrated displacement and momentum thicknesses along with the shape factor were computed using Eq. (1) through Eq. (3). These are the forms of the equations that follow the same derivation for 2D/planar boundary-layers, and their simpler functional forms, when accounting for the mass flow and momentum deficit resulting from a boundary-layer growing on the concave surface of an annular flow, as is the case here.

$$\frac{\delta^*}{R} = 1 - \sqrt{1 - 2 \int_0^{\delta/R} \left(1 - \frac{\rho}{\rho_e} \frac{u}{u_e}\right) \left(1 - \frac{y}{R}\right) d\left(\frac{y}{R}\right)} \quad (1)$$

$$\frac{\theta}{R} = 1 - \sqrt{1 - 2 \int_0^{\delta/R} \frac{\rho}{\rho_e} \frac{u}{u_e} \left(1 - \frac{u}{u_e}\right) \left(1 - \frac{y}{R}\right) d\left(\frac{y}{R}\right)} \quad (2)$$

$$H = \frac{\delta^*}{\theta} \quad (3)$$

Figure 7 shows the boundary-layer quantities for the CFD-derived profiles of Fig. 5 and Fig. 6. The figure shows that not all the quantities of interest are converging monotonically, namely the displacement thickness for the Wind-US SA turbulence model simulation, and the momentum thickness for the VULCAN-CFD SA turbulence model simulation and Wind-US SST-V turbulence model simulation. This is due to the sensitivity of where the maximum

axial velocity or the maximum axial momentum value occurs with different grid levels, which is used to define the boundary-layer edge. Unlike an external flow problem where the farfield values are typically constant prescribed freestream levels for all grid levels, for an internal flow problem such as that considered here, differences in the boundary-layer growth from one grid to another results in a different “edge” state for both the velocities and densities used in the integral thickness equations.

Due to the sensitivities caused by variation in the boundary-layer edge quantities, grid convergence of the wall shear stress, shown in Table 2, along with the corresponding skin friction coefficient, shown in Table 3, were examined. The skin friction coefficients were computed using the reference axial velocity of 589.1 m/s and reference density of 0.3806 kg/m³. Table 2 and Table 3 show that both the wall shear stress and the corresponding skin friction coefficient are converging monotonically, except for the Wind-US SA turbulence model case. That case’s convergence shows very low magnitude oscillatory-like behavior with a max oscillation on the order of less than 0.2% (based on the value on the finest grid level). Table 2 and Table 3 also show that the Wind-US SST-V turbulence model case and the VULCAN-CFD SA turbulence model case are still converging on a value for the wall shear stress and the corresponding skin friction coefficient. Quantifiably, the max difference between successive grid levels based on the finest grid level are less than 0.9% for the Wind-US case and less than 2.0% for the VULCAN-CFD case.

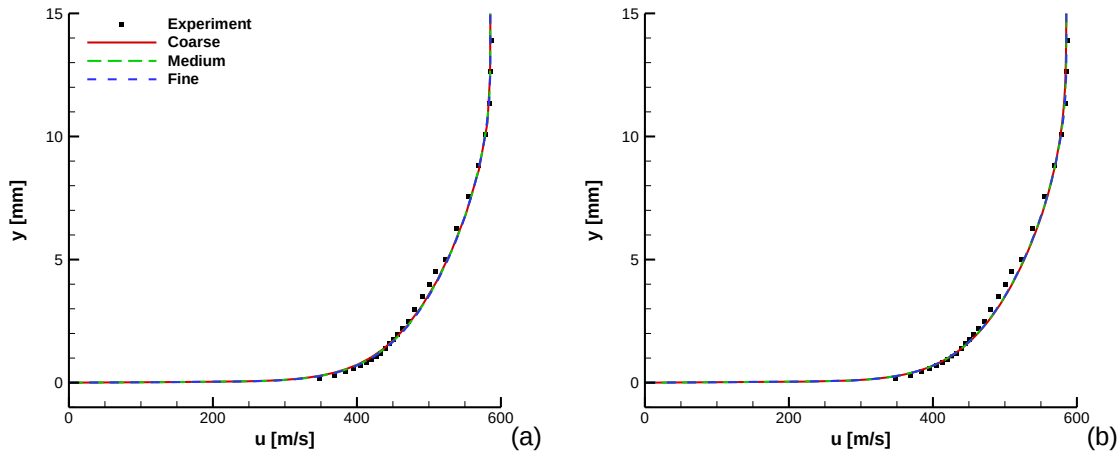


Figure 5. Experimental incoming boundary-layer station axial velocity profiles for the (a) Wind-US and (b) VULCAN-CFD SA turbulence model simulations as a function of the grid density.

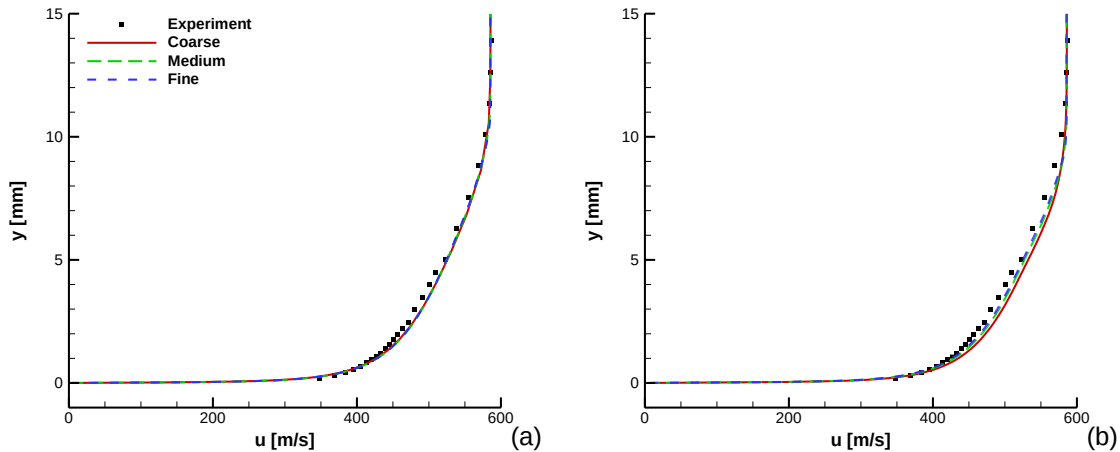


Figure 6. Experimental incoming boundary-layer station axial velocity profiles for the (a) Wind-US and (b) VULCAN-CFD SST-V turbulence model simulations as a function of the grid density.

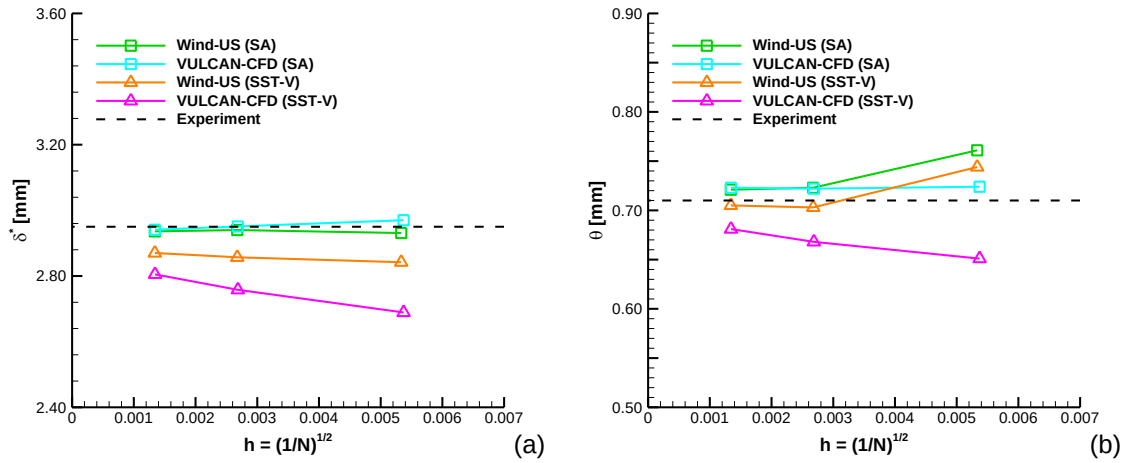


Figure 7. (a) Displacement thickness and (b) momentum thickness as a function of the grid refinement.

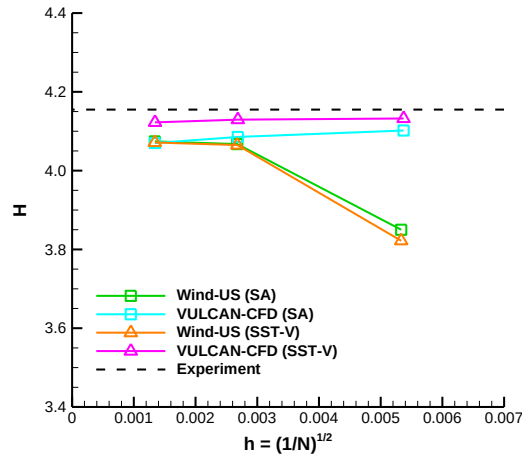


Figure 8. Shape factor as a function of the grid refinement.

Table 2. Wall shear stress convergence, in units of N/m^2 .

Code	Model	Coarse	Medium	Fine
Wind-US	SA	106.6	106.5	106.7
VULCAN	SA	111.5	110.9	108.7
Wind-US	SST-V	102.8	103.7	104.6
VULCAN	SST-V	104.2	104.1	103.6

Table 3. Skin friction coefficient convergence ($\times 10^{-3}$).

Code	Model	Coarse	Medium	Fine
Wind-US	SA	1.615	1.613	1.616
VULCAN	SA	1.688	1.679	1.646
Wind-US	SST-V	1.556	1.570	1.584
VULCAN	SST-V	1.577	1.576	1.569

B. Test Section Wall

The test section wall pressure profiles in the vicinity of the interaction region are shown in Fig. 9 and Fig. 10 for the 10° half-angle shock generator simulations and Fig. 11 and Fig. 12 for the 16° half-angle shock generator simulations. For the four figures shown, 1) the axial distance has been normalized by both the nominal inviscid shock impingement location, x_i , and the experimental incoming boundary-layer station boundary-layer displacement thickness, δ_1^* and 2) the pressure has been normalized by the nozzle exit pressure. Figs. 9 through 12 show that the test section wall pressure profiles are nearly converged on the finest grid level. The exception to this is the VULCAN-CFD SST-V turbulence model simulations which still show a slight change in the peak pressure for the 10° half-angle shock generator case and slight change in the location of the start of the pressure rise for the 16° half-angle shock generator case. This is displayed as differences in the green-dashed to blue-dashed lines in Fig. 10b and Fig. 12b, respectively.

Given the grid convergence behavior of the metrics explored at the experimental incoming boundary-layer station as well as the test section wall pressure profile, the fine grid level was deemed to be sufficiently grid converged and was used for the simulations presented in the remainder of the paper.

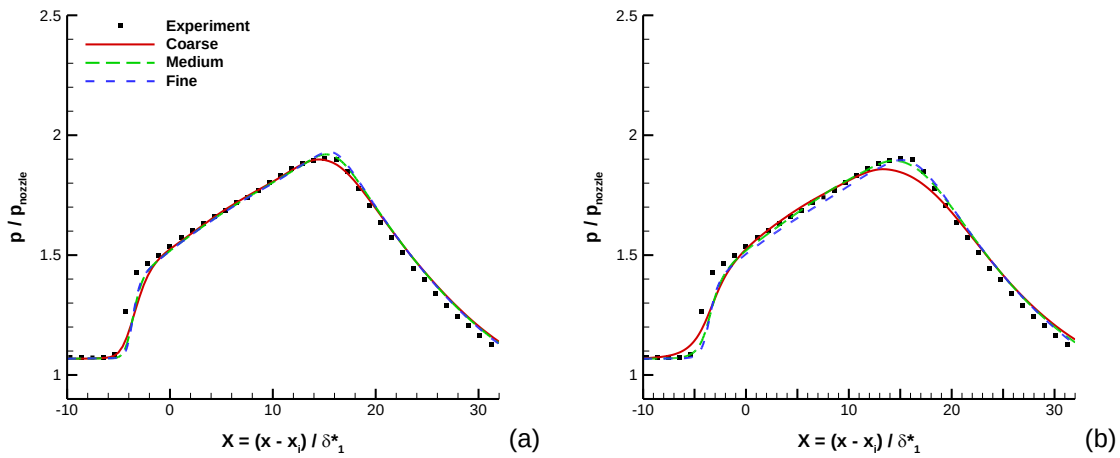


Figure 9. Test section wall pressure profiles for the (a) Wind-US and (b) VULCAN-CFD 10° half-angle shock generator SA turbulence model simulations as a function of the grid density.

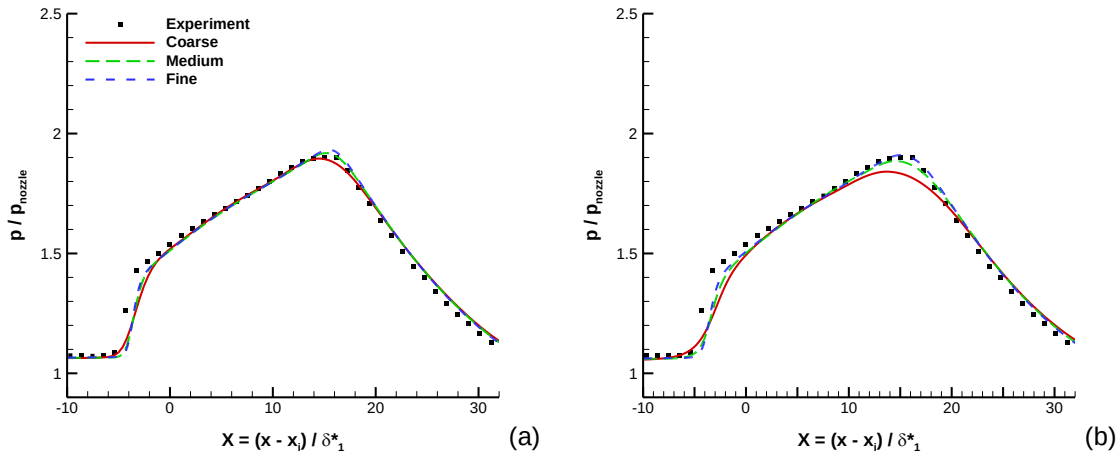


Figure 10. Test section wall pressure profiles for the (a) Wind-US and (b) VULCAN-CFD 10° half-angle shock generator SST-V turbulence model simulations as a function of the grid density.

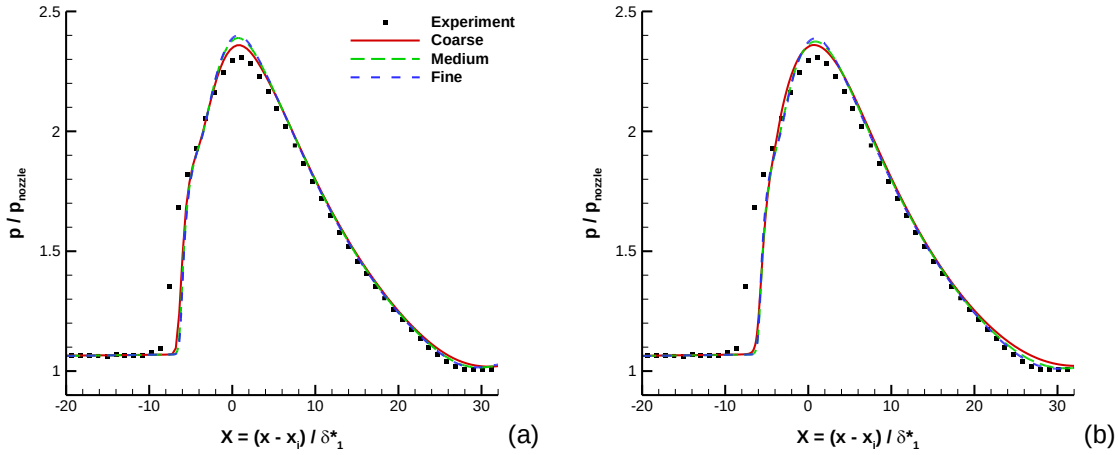


Figure 11. Test section wall pressure profiles for the (a) Wind-US and (a) VULCAN-CFD 16° half-angle shock generator SA turbulence model simulations as a function of the grid density.

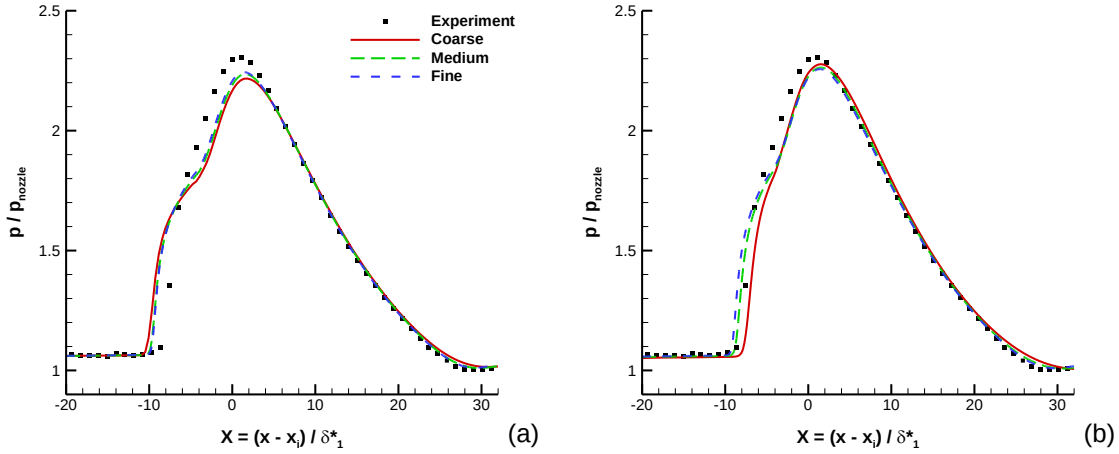


Figure 12. Test section wall pressure profiles for the (a) Wind-US and (a) VULCAN-CFD 16° half-angle shock generator SST-V turbulence model simulations as a function of the grid density.

V. Comparisons with Experimental Measurements

This section contains the fine grid simulation results with more detailed comparisons with the experimental data. The results have been grouped by the test section location, starting upstream.

A. Experimental Incoming Boundary-Layer Station

The axial velocity profiles for the 16° half-angle shock generator cases at the experimental incoming boundary-layer station are shown in Fig. 13a for the SA turbulence model simulations and Fig. 14a for the SST-V turbulence model simulations. The experimental quantities are derived from the Pitot probe measurements. The 10° half-angle shock generator cases showed similar behavior and are not shown in this subsection. From Fig. 13a and Fig. 14a it can be seen that all three CFD codes predict qualitatively similar axial velocity regardless of the turbulence model used. It is also shown that the predicted profiles qualitatively agree with experiment, noting that the SA turbulence model solutions appear to be in slightly closer agreement with the experiment than the SST-V turbulence model solutions.

To quantify the comparisons, the integrated displacement and momentum thicknesses along with the shape factor were computed and are displayed in Table 4 for the SA turbulence model simulations and Table 5 for the SST-V turbulence model simulations. To help quantify the uncertainty in the CFD-derived values of Tables 4 and 5, a percent difference was computed, as defined by Eq. (4).

$$\% \text{ Difference} = \frac{100 \times (CFD_{max} - CFD_{min})}{Experiment} \quad (4)$$

Table 6 shows the computed percent differences, from which it is shown that the uncertainty, or variation from one code's results to another, increases with the use of the two-equation turbulence model compared to the one-equation turbulence model.

To further compare the CFD simulations, the non-dimensional eddy viscosity profiles were plotted, as depicted in Fig. 13b for the SA turbulence model simulations and Fig. 14b for the SST-V turbulence model simulations. The eddy viscosity profiles were made non-dimensional by the reference dynamic viscosity of $9.528 \times 10^{-6} \text{ kg/(m}\cdot\text{s)}$. From a qualitative perspective, the simulations from all three codes predict similar profiles for both turbulence models. To make quantitative observations, the maximum value of the non-dimensional eddy viscosity was computed with the tabulated results in Table 7. From this table it is noted that 1) the FUN3D simulations are predicting a lower maximum non-dimensional eddy viscosity value compared to the other two codes and 2) the spread in the maximum value increases with the use of the SST-V turbulence model relative to the SA turbulence model. This is due to differences in the predicted shear layer behavior between the simulations. Specifically, while all the simulations predict a shear layer starting around the nozzle throat, the SST-V turbulence model simulations predict a second shear layer that starts further downstream. As the second shear layer forms, the initial shear layer disappears, with the CFD codes varying in the prediction of where this “transition” occurs.

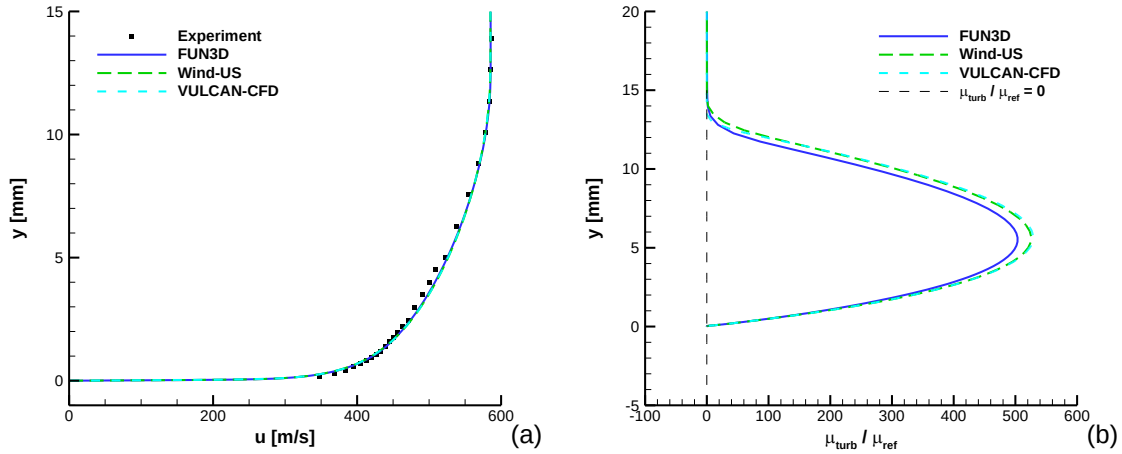


Figure 13. Experimental incoming boundary-layer station (a) axial velocity profiles and (b) non-dimensional eddy viscosity profiles for the SA turbulence model simulations.

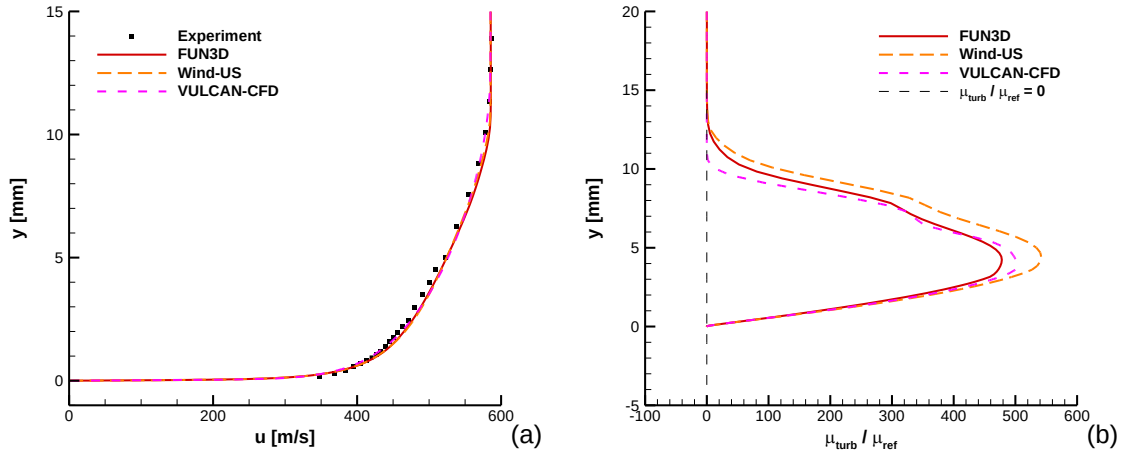


Figure 14. Experimental incoming boundary-layer station (a) axial velocity profiles and (b) non-dimensional eddy viscosity profiles for the SST-V turbulence model simulations.

Table 4. Experimental incoming boundary-layer station boundary-layer quantities for the SA turbulence model simulations.

δ^* [mm]			
FUN3D	Wind-US	VULCAN	Experiment
2.93	2.94	2.94	2.95
θ [mm]			
FUN3D	Wind-US	VULCAN	Experiment
0.727	0.721	0.723	0.710
H			
FUN3D	Wind-US	VULCAN	Experiment
4.026	4.074	4.070	4.155

Table 5. Experimental incoming boundary-layer station boundary-layer quantities for the SST-V turbulence model simulations.

δ^* [mm]			
FUN3D	Wind-US	VULCAN	Experiment
2.83	2.87	2.81	2.95
θ [mm]			
FUN3D	Wind-US	VULCAN	Experiment
0.701	0.705	0.681	0.710
H			
FUN3D	Wind-US	VULCAN	Experiment
4.038	4.072	4.123	4.155

Table 6. Uncertainty in the CFD-derived boundary-layer quantities at the experimental incoming boundary-layer station.

	% Difference	
	SA	SST-V
δ^*	0.4	2.2
θ	0.9	3.7
H	1.2	2.1

$$\% \text{ Difference} = 100 \times (\text{CFD}_{\text{Max}} - \text{CFD}_{\text{Min}}) / \text{Experiment}$$

Table 7. Maximum eddy viscosity at the experimental incoming boundary-layer station.

Model	Code	μ_t / μ_{ref}
SA	FUN3D	503
SA	Wind-US	526
SA	VULCAN-CFD	529
SST-V	FUN3D	478
SST-V	Wind-US	542
SST-V	VULCAN-CFD	503

B. Interaction Region

This subsection explores the results with respect to the interaction region and is broken out by the shock generator geometry.

1. 10° Half-Angle Shock Generator

The test section wall pressure profiles spanning the interaction region are shown in Fig. 15, which shows that all three codes predict nearly identical wall pressure profiles, independent of the turbulence model used. The exception to this is in the prediction of the peak pressure, where the VULCAN-CFD SA turbulence model solution predicts a slightly lower value than the FUN3D and Wind-US solutions. Further, the CFD simulations predict the overall pressure rise and fall and match the experiment reasonably well, noting that the CFD simulations consistently predict the start of the pressure rise downstream of the reported experimental location.

To further investigate what is happening off the wall and towards the core of the wind tunnel, Pitot pressure profiles as well as the corresponding mass flux profiles for the first five upstream profiles from the hot-wire measurements were plotted at several axial stations as shown in Fig. 16. For reference, Fig. 15a has dashed vertical lines to show where these axial stations are in relation to the test section wall pressure profiles. In Fig. 16b, the CFD derived mass flux is defined, within the RANS equations framework, as the averaged value of the product of density and axial velocity while for the experiment it was defined as the averaged value of density times axial velocity. Figure 16 omits the SST-V turbulence model results as they showed very similar results to those of the SA turbulence model, namely that all three CFD codes agree well with each other and the experiment. This is also demonstrated in the axial velocity profiles of Fig. 17a, with the experimental data taken from the PIV measurements. The SST-V turbulence model solutions are included in Fig. 17b as they show marginally better agreement with the experiment towards the core flow (i.e., $y = 7.1$ mm to $y = 11.2$ mm).

Further downstream in the interaction region, only Pitot pressure and PIV measurements were made, whose axial locations are shown in Fig. 18a. The Pitot pressure measurement station locations, represented by the black dashed vertical lines in Fig. 18a, form a subset of the PIV measurement stations. From Fig. 18b it is shown that the CFD solutions from all three codes agree well with each other and the experimental measurements, which is further supported in the corresponding axial velocity profiles of Fig. 19. The good agreement is to be expected as there is a lack of separated flow off the wall, in both the experiment and computations, as evidenced in the wall skin friction coefficient profiles shown in Fig. 20.

To explore turbulence quantities, the turbulent shear stress and turbulent kinetic energy were plotted. Figure 21 depicts turbulent shear stress profiles at the axial stations shown in Fig. 15a, with the experimental data being the PIV data. While the term RANS usually refers to the broad category of turbulent flow simulations from incompressible to compressible regimes, strictly speaking most general-purpose solvers treat the solved equations not as only spatially-averaged (pure Reynolds-averaged), but Favre mass-averaged. If only Reynolds-averaging was assumed, there would be several extra terms involving density fluctuations which would require closure. Assuming Favre-averaging removes the density fluctuations and as a result, the turbulent stress terms imply a density-weighting. Figure 21a shows that the SA turbulence model solutions from all three CFD codes agree well with each other except near the wall, where the FUN3D solution predicts slightly lower magnitude peak values compared to the other two codes. Also, the CFD solutions predict larger turbulent shear stress magnitudes near the wall compared to the experiment. This is due to the PIV technique used in the experiment not being able to pick up near wall gradients [28], which resulted in the PIV data underpredicting the peak turbulent shear stress near the wall. Figure 21b shows that the SST-V turbulence model solutions from all three CFD codes agree well with each other, noting that the VULCAN-CFD solution predicts higher magnitude peak values compared to the other two codes near the wall for the four downstream most stations depicted in Fig. 21b. Compared to the SA turbulence model solutions, the SST-V turbulence model solutions are predicting lower magnitudes of the turbulent shear stress, and thus are in closer agreement with the

experiment. Both the SA turbulence model solutions and SST-V turbulence model solutions show better agreement near the wall between the three codes and when compared to the experiment in the downstream portion of the interaction region. These axial stations are depicted in Fig. 18a, and the corresponding turbulent shear stress profiles are shown in Fig. 22. Closer inspection of Fig. 22 indicates that it takes the SST-V turbulence model solutions longer to resume better agreement compared to the SA turbulence model solutions. This is due to differences in the predicted stress fields between the three codes.

Figure 23 shows the turbulent kinetic energy profiles for the SST-V turbulence model solutions at the stations depicted in Fig. 15a, with the experimental data being the PIV data. This plot shows that the Wind-US and FUN3D solutions are in good agreement with each other. The VULCAN-CFD solution is shown to predict lower values of turbulent kinetic energy towards the core flow and predict higher turbulent kinetic energy near the wall for all stations except for the first two upstream stations. Both these trends are as compared to the other two flow solvers. When compared against the experiment, the CFD predicts lower turbulent kinetic energy near the wall in the first few downstream profiles, or in other words in the portion of SWBLI where the flow begins to recover. Agreement near the wall is shown to improve in locations located further downstream in the interaction region, as evidenced in Fig. 24.

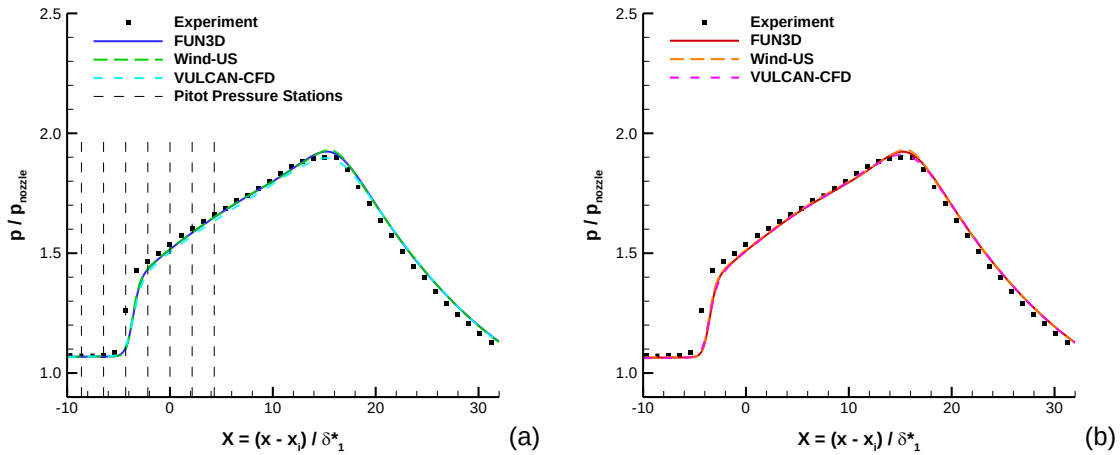


Figure 15. Test section wall pressures for the (a) SA turbulence model solutions and (b) SST-V turbulence model solutions for the 10° half-angle shock generator cases.

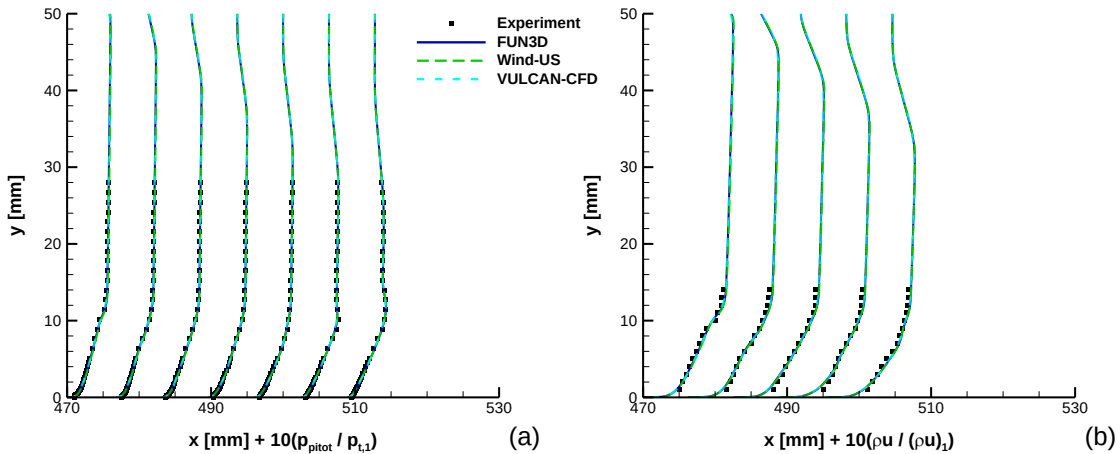


Figure 16. SA turbulence model solutions (a) Pitot pressure profiles and (b) mass flux profiles for the 10° half-angle shock generator cases.

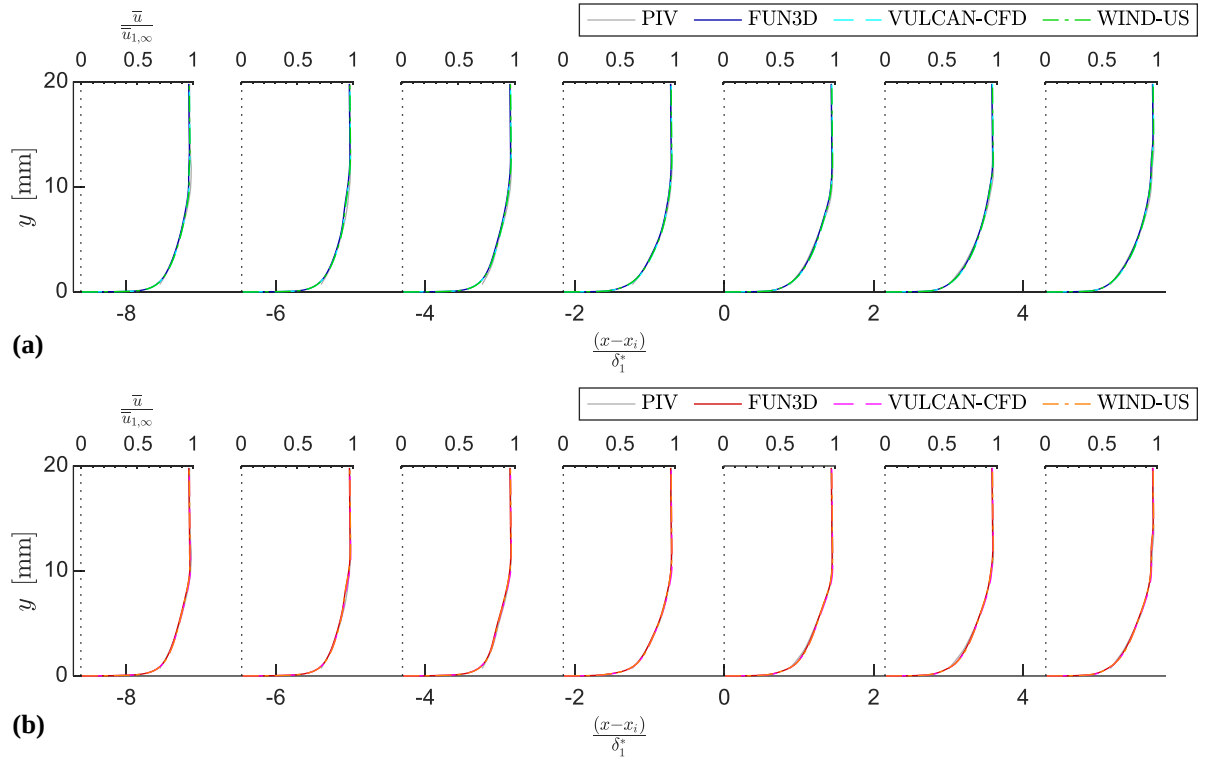


Figure 17. (a) SA turbulence model solutions and (b) SST-V turbulence model solutions axial velocity profiles for the 10° half-angle shock generator case.

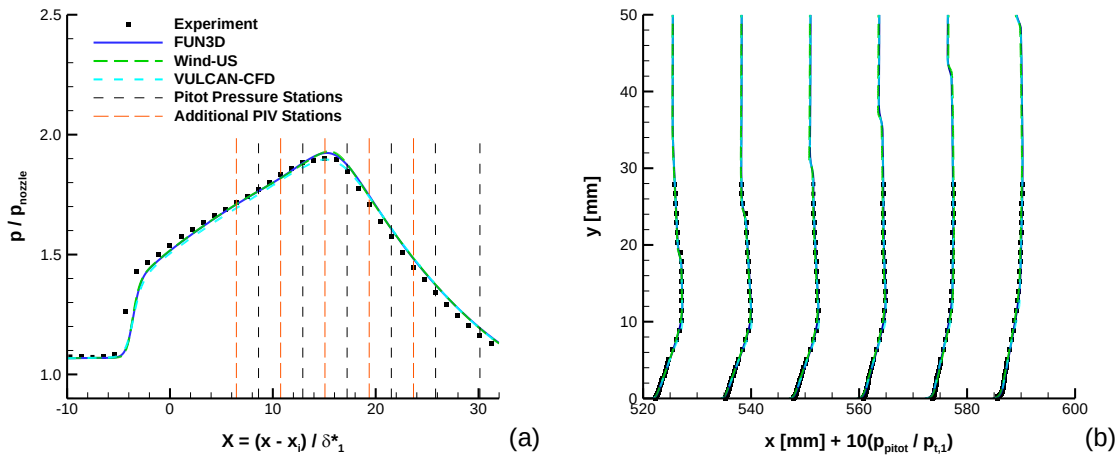


Figure 18. (a) Downstream Pitot pressure and PIV axial stations and (b) SA turbulence model solutions downstream Pitot pressure profiles for the 10° half-angle shock generator case.

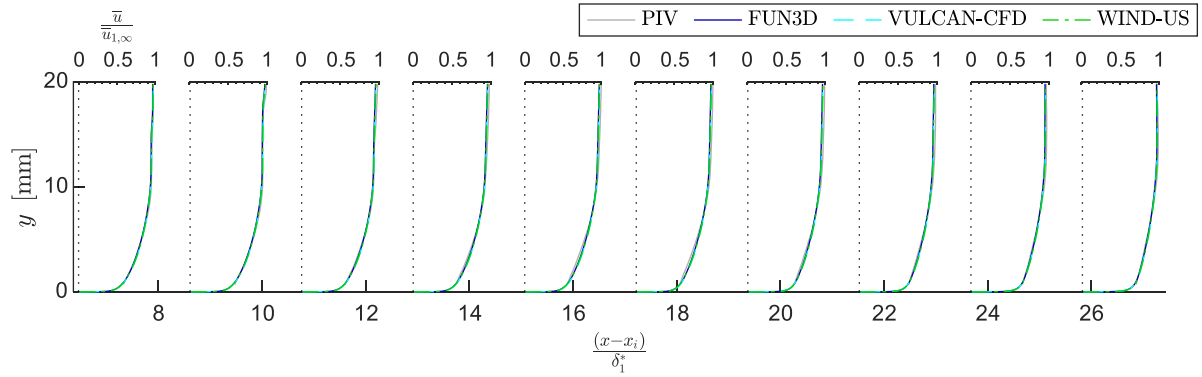


Figure 19. SA turbulence model solutions downstream axial velocity profiles for the 10° half-angle shock generator case.

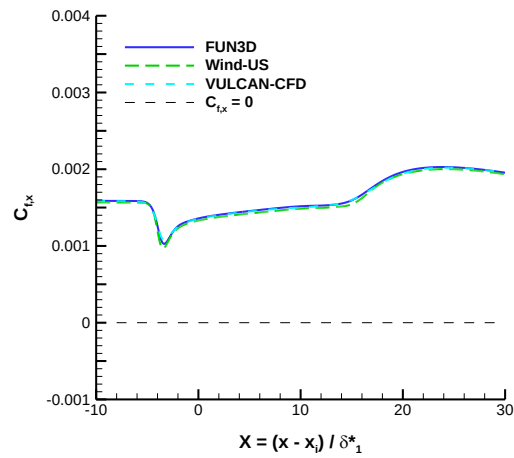


Figure 20. SA turbulence model solutions skin friction coefficient profiles for the 10° half-angle shock generator case.

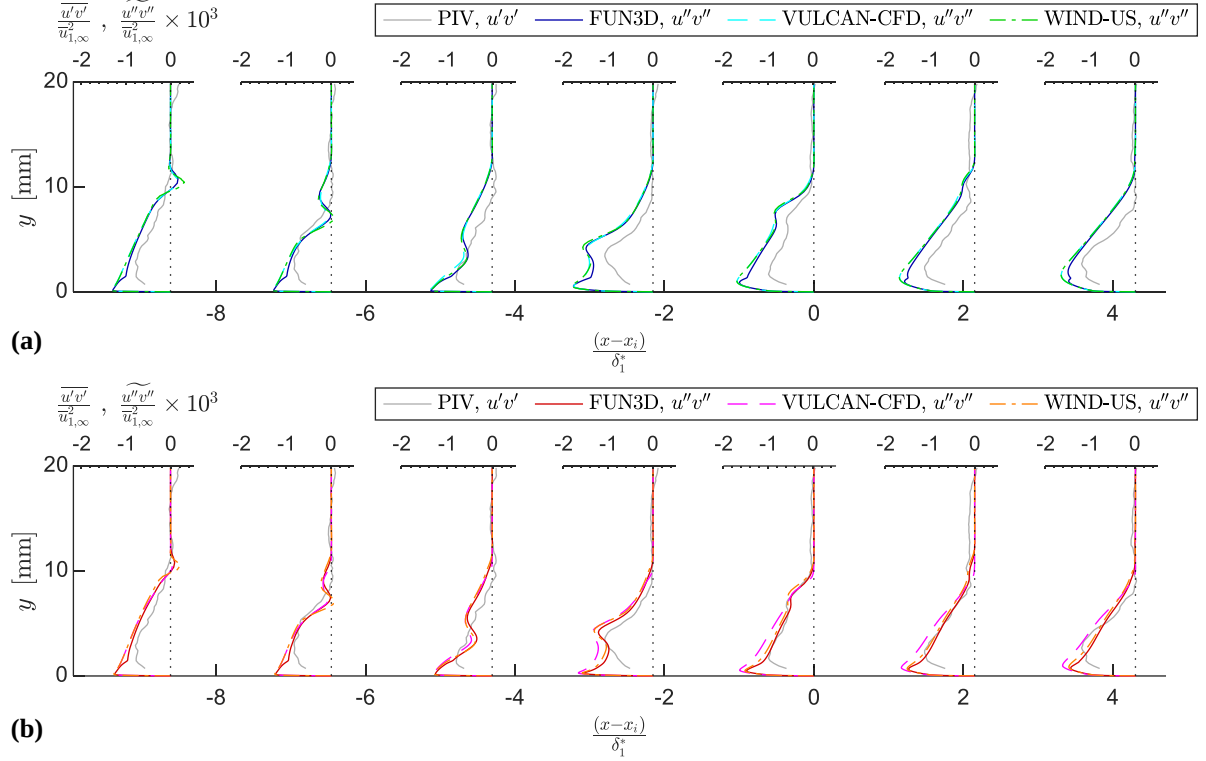


Figure 21. Turbulent shear stress profiles for the (a) SA turbulence model solutions and (b) SST-V turbulence model solutions for the 10° half-angle shock generator cases.

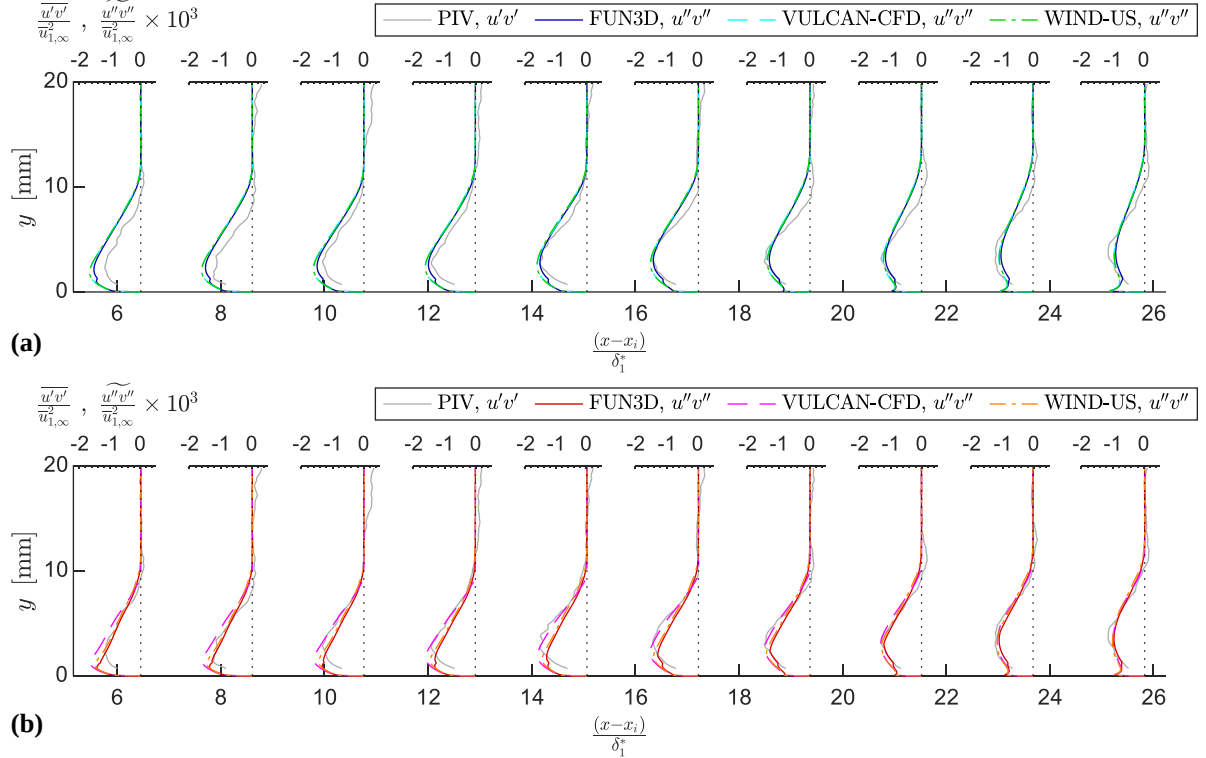


Figure 22. Downstream turbulent shear stress profiles for the (a) SA turbulence model solutions and (b) SST-V turbulence model solutions for the 10° half-angle shock generator cases.

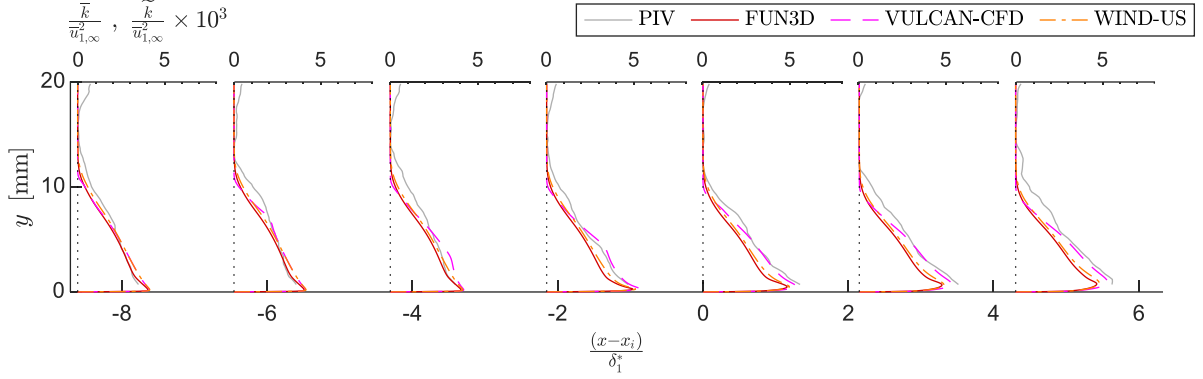


Figure 23. SST-V turbulence model solutions turbulent kinetic energy profiles for the 10° half-angle shock generator cases.

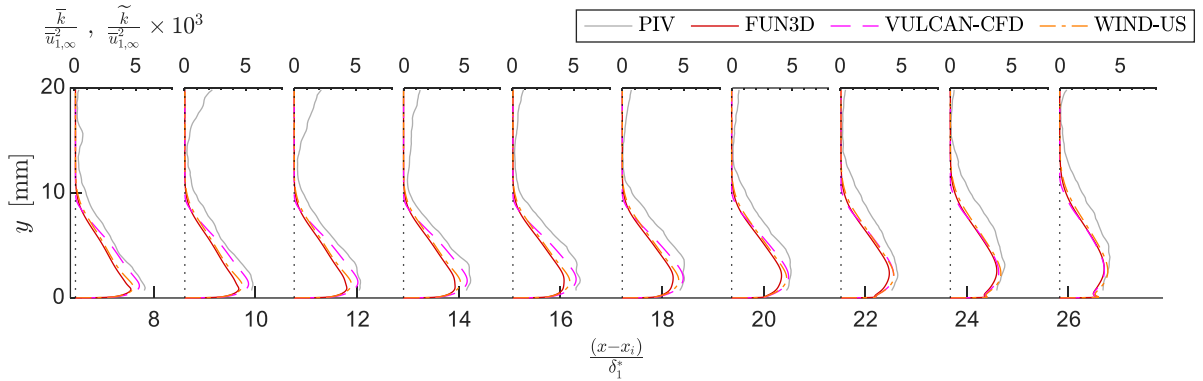


Figure 24. SST-V turbulence model solutions downstream turbulent kinetic energy profiles for the 10° half-angle shock generator cases.

2. 16° Half-Angle Shock Generator

The test section wall pressure profiles spanning the interaction region are shown in Fig. 25. Figure 25a shows that all three codes predict nearly identical wall pressure profiles for the SA turbulence model simulations which consistently 1) overpredict the peak pressure and 2) predict the start of the pressure rise downstream of the reported experimental location. Figure 25b shows that there is a code sensitivity to the prediction of the pressure rise region for the SST-V turbulence model simulations. The furthest upstream location for the start of the pressure rise is predicted by the Wind-US code while the most downstream location is predicted by the FUN3D code. This corresponds to the differences in the predicted start of the separation region, which will be explored later in this subsection. The SST-V turbulence model solutions also slightly underpredict the peak pressure while predicting the start of the pressure rise at a location closer to the reported experimental location compared to the SA turbulence model solutions.

Pitot pressure profiles, along with the corresponding mass flux profiles, were plotted at several axial stations and are shown in Fig. 26 for the SA turbulence model solutions and Fig. 27 for the SST-V turbulence model solutions. For reference, Fig. 25 shows dashed vertical lines to show where these axial stations are in relation to the test section wall pressure profiles. As a reminder, the CFD derived mass flux profiles were defined as the averaged value of density times the averaged value of axial velocity while the experiment mass flux profiles were defined as the averaged value of density times axial velocity. From Fig. 26 and Fig. 27 it is observed that the CFD simulations are in good agreement with each other except for the Wind-US SST-V turbulence model solution, which predicts a slightly different reflected shock position. Additionally, Fig. 26b and Fig. 27b show that the SA turbulence model solutions generally predict the peak mass flux to be closer to the wall compared to the SST-V turbulence model solutions and the hot-wire data. This is due to the SA turbulence model solutions predicting a smaller SWBLI extent than the SST-V turbulence model solutions and the experiment. The trends found in the Pitot pressure and mass flux profiles are shown in the axial velocity profiles of Fig. 28, which are compared to the PIV data. Figure 28b also shows that the Wind-US and VULCAN-CFD SST-V turbulence model simulations predict reversed flow near the wall earlier than

the FUN3D SST-V turbulence model simulation, which is consistent with the predicted start of the wall pressure rise location.

Like the 10° half-angle shock generator case, only Pitot pressure and PIV measurements were obtained in the latter portion of the interaction region, with axial stations downstream of the profiles shown in Fig. 26 through Fig. 28. These additional profiles, shown in Fig. 29 and Fig. 30 for the Pitot pressure and Fig. 31 for the axial velocity, show that the CFD solutions all agree well with each other and reasonably close with the experiment. The good agreement is to be expected as there is a lack of separated flow off the wall in this region (i.e., $X = 8.61$ to $X = 30.14$), as evidenced in the wall skin friction coefficient profiles shown in Fig. 32. While Fig. 32 does show that the flow is separated off the test section wall, the region of separation is upstream of the axial profiles in Fig. 29 through Fig. 31 but encompasses some of the upstream axial stations included in Fig. 26 through Fig. 28 (i.e., $X = -8.61$ to $X = -2.15$). The boundary-layer separation on this portion of the wall is due to the larger shock generator angle and was also seen in the experiment, as noted in Fig. 32. The experimental values for the separation and reattachment locations were estimated from oil flow visualization. Figure 32 shows that the SA turbulence model solutions all agree with each other with respect to the prediction of the separation and reattachment points and more closely matches the experimental values compared to the SST-V turbulence model solutions. The SST-V turbulence model solutions meanwhile show some disagreement in the prediction of the separation point, namely that the Wind-US solution predicts an earlier separation than the FUN3D and VULCAN-CFD simulations. These observations are quantified in Tables 8 and 9 and are consistent with the wall pressure profiles of Fig. 25 and the Pitot pressure and mass flux profiles of Fig. 26. From Fig. 32 it is also noted that the SST-V turbulence model solutions predict a larger separated region compared to the SA turbulence model solutions and the experiment, which has been observed elsewhere such as in Ref. [27].

Profiles of the turbulent shear stress for the upstream portion of the interaction region are plotted in Fig. 33. The profiles are at the axial stations depicted in Fig. 25 and are compared against the PIV data. Figure 33a shows that the SA turbulence model solutions are consistent between the three codes, except in the prediction of the secondary peak at $X = -4.31$. Also, all three codes show the least agreement with the experiment between $X = -6.46$ and $X = -2.15$. This is somewhat to be expected as this is the location that the CFD simulations predict the separated region. From Fig. 33b it is noted that all the SST-V turbulence model simulations agree well away from the wall. Near the wall, the FUN3D simulations are predicting lower peak magnitudes of the shear stress compared to other two codes, with the other two codes agreeing well with each other. Like the SA turbulence model solutions, the SST-V turbulence model solutions show the least agreement with the experiment within the extent of the separation region. Further downstream at the axial stations denoted by vertical dashed lines in Fig. 29a, the SA turbulence model simulations agree with well each other and underpredict the magnitude of the peak turbulent shear stress compared to the experiment, as evidenced in Fig. 34a. A similar trend is noted for the SST-V turbulence model solutions, depicted in Fig. 34b. Unlike the SA turbulence model solutions, the disagreement near the wall between the three codes for the SST-V turbulence model solutions lasts until $X = 17.22$, noting that the SA turbulence model solutions show good agreement well upstream of that location (i.e., by $X = 4.31$). This trend between the different turbulence model solutions is consistent with the skin friction coefficient plots of Fig. 32.

Figure 35a shows the turbulent kinetic energy profiles for the SST-V turbulence model solutions at the stations depicted in Fig. 25b, with the experimental data being the PIV data. From this plot it is shown that near the wall the FUN3D simulations predict the least amount of turbulent kinetic compared to the other two codes (which are in good agreement with each other). The near wall agreement between the three codes improves further downstream within the interaction region, as depicted in Fig. 35b. Figure 35b also shows that the CFD codes agree well with the experiment in the region away from the wall.

To further interrogate the interaction region, the axial and radial component velocity contours are respectively shown in Fig. 36 and Fig. 37. The figures compare the CFD results to PIV data, from which the following is noted. First, the predicted CFD velocity fields appear to be qualitatively similar among the three codes when using the same turbulence model, with the SST-V turbulence model solutions predicting a larger extent of lower velocity near the wall. Second, the CFD predicted velocity fields qualitatively agree well with the experiment, with the radial velocity contours more clearly showing the shock structure. Third, the initial shock from the cone tip along with the reflected shock off the wind tunnel wall are both visible in the CFD solutions and the PIV data. This is most evident when looking at the radial velocity contours in Fig. 37. Fourth, the CFD simulations are picking up a secondary SWBLI generated by the reflected shock-wave off the wind tunnel wall interacting with the shock generator boundary-layer. This is most evident in the upper right portion of the SST-V turbulence model simulation contours and is not shown in the experiment. Its absence in the experimental data is because the focus of the experiment was the SWBLI generated from the shock off the cone tip interacting with the test section wall and not the subsequent downstream

SWBLI(s). Fifth, the CFD solutions show much sharper shocks than the PIV data. The shock smoothness seen in the PIV data is due to the inherent way PIV data is post-processed, i.e., via window-averaging. For those interested, additional information on the PIV setup and data acquisition can be found in Ref. [28] and Ref. [29] while additional comparisons to the PIV data can be found in Ref. [26].

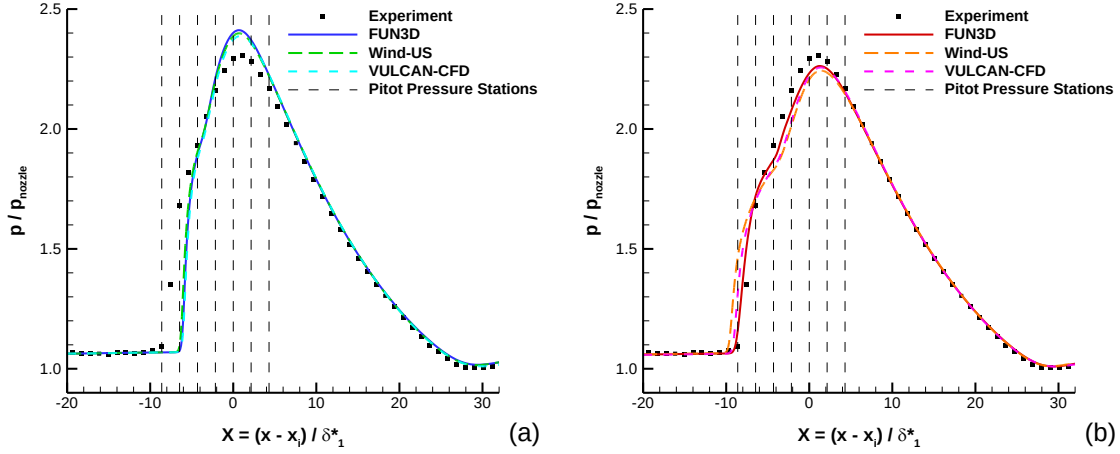


Figure 25. Test section wall pressures for the (a) SA turbulence model solutions and (b) SST-V turbulence model solutions for the 16° half-angle shock generator cases.

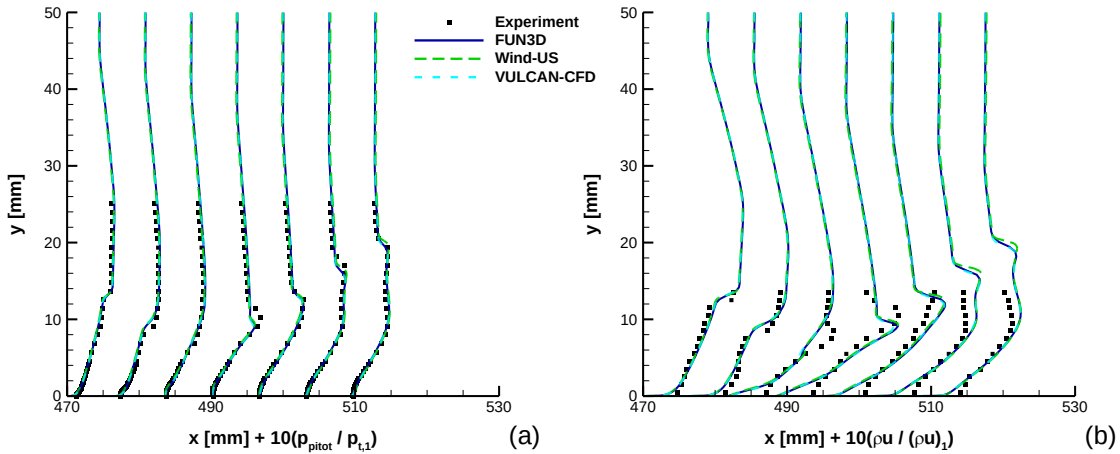


Figure 26. SA turbulence model solutions (a) Pitot pressure profiles and (b) mass flux profiles for the 16° half-angle shock generator cases.

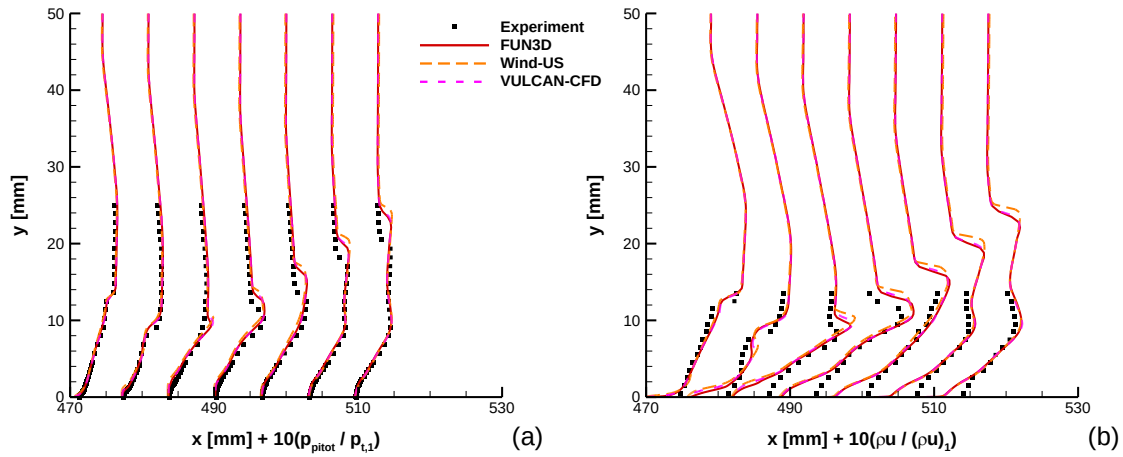


Figure 27. SST-V turbulence model solutions (a) Pitot pressure profiles and (b) mass flux profiles for the 16° half-angle shock generator cases.

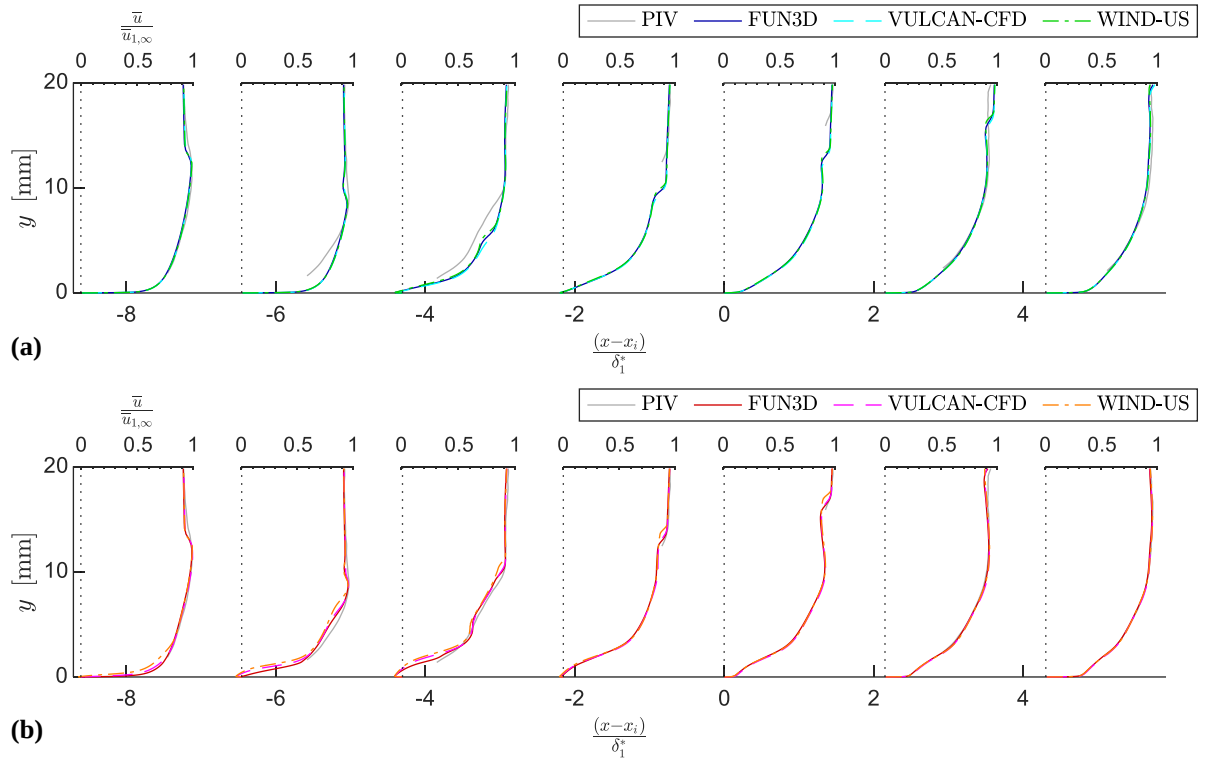


Figure 28. (a) SA turbulence model solutions and (b) SST-V turbulence model solutions axial velocity profiles for the 16° half-angle shock generator case.

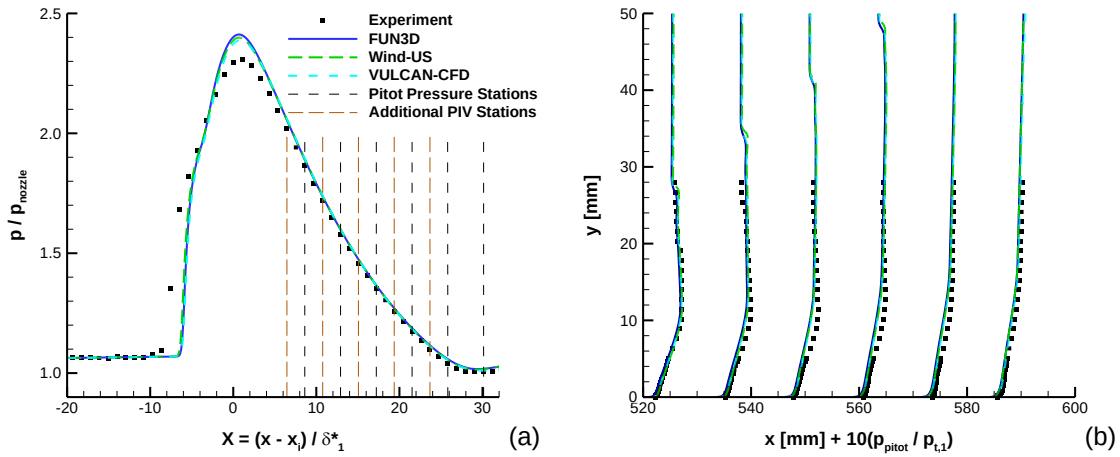


Figure 29. (a) Downstream Pitot pressure stations and (b) SA turbulence model solutions downstream Pitot pressure profiles for the 16° half-angle shock generator case.

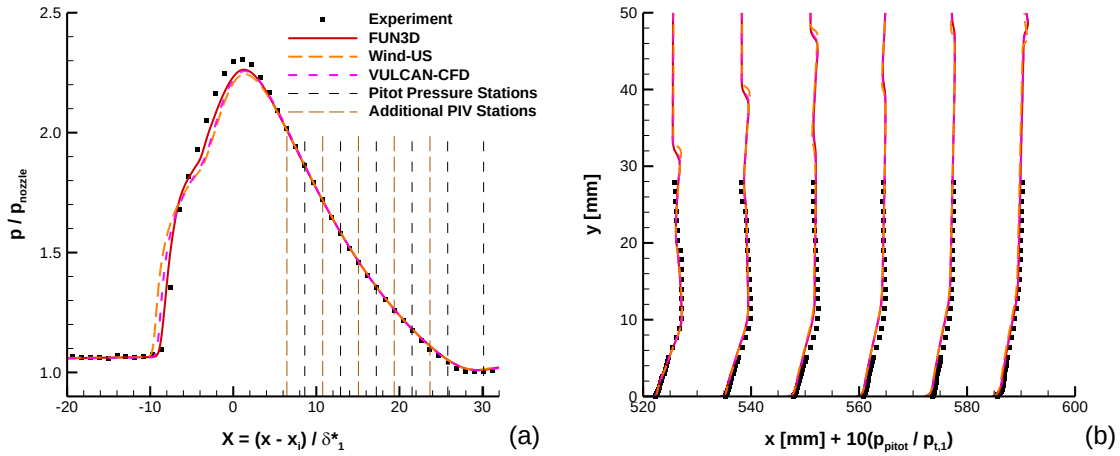


Figure 30. (a) Downstream Pitot pressure stations and (b) SST-V turbulence model solutions downstream Pitot pressure profiles for the 16° half-angle shock generator case.

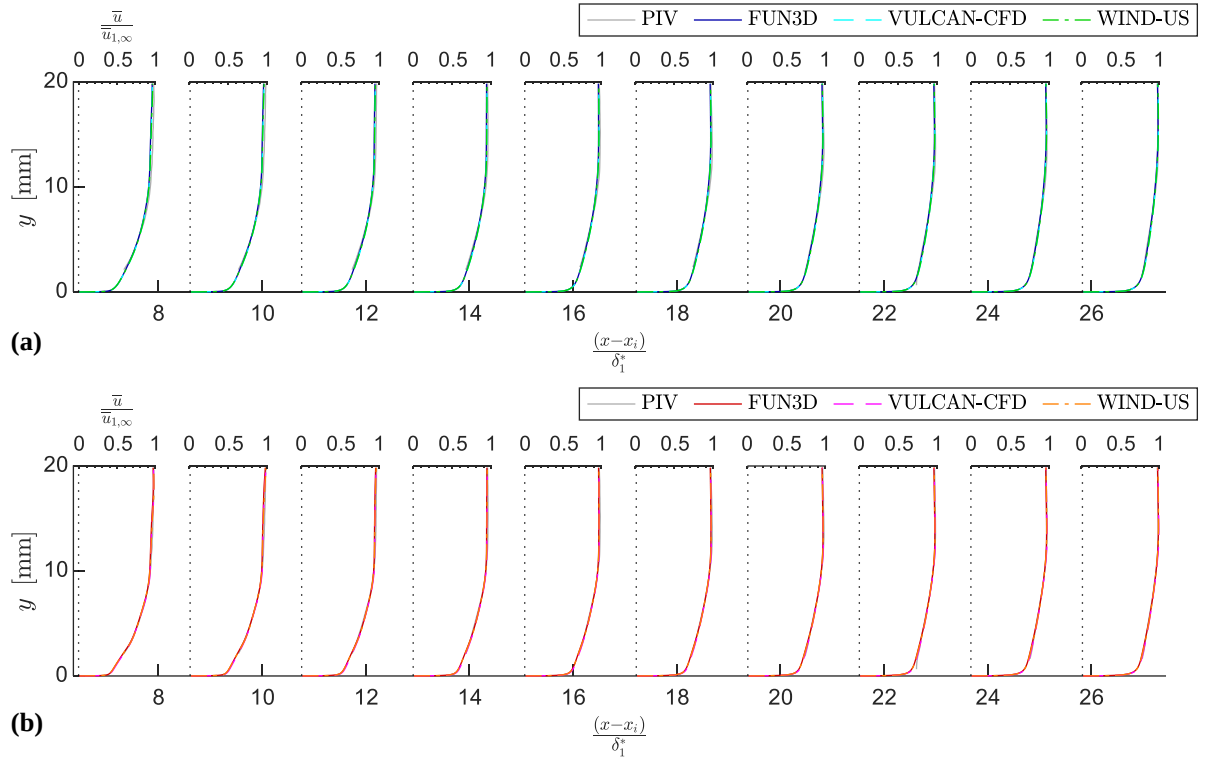


Figure 31. (a) SA turbulence model solutions and (b) SST-V turbulence model solutions downstream axial velocity profiles for the 16° half-angle shock generator case.

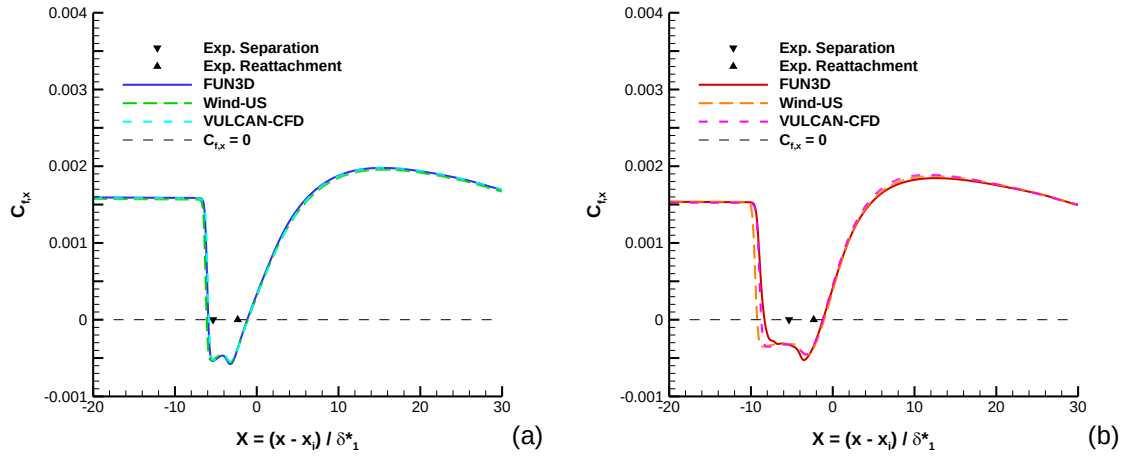


Figure 32. (a) SA turbulence model solutions and (b) SST-V turbulence model solutions skin friction coefficient profiles for the 16° half-angle shock generator case.

Table 8. SWBLI-induced separation extent (mm) for the SA turbulence model simulations.

Code	x_{start}	x_{end}	L
Experiment	479.6	488.4	8.9
FUN3D	477.8	491.9	14.0
Wind-US	477.3	492.1	14.9
VULCAN-CFD	478.0	492.1	14.0

Table 9. SWBLI-induced separation extent (mm) for the SST-V turbulence model simulations.

Code	x_{start}	x_{end}	L
Experiment	479.6	488.4	8.9
FUN3D	470.7	491.8	21.1
Wind-US	468.1	492.2	24.2
VULCAN-CFD	469.5	491.7	22.1

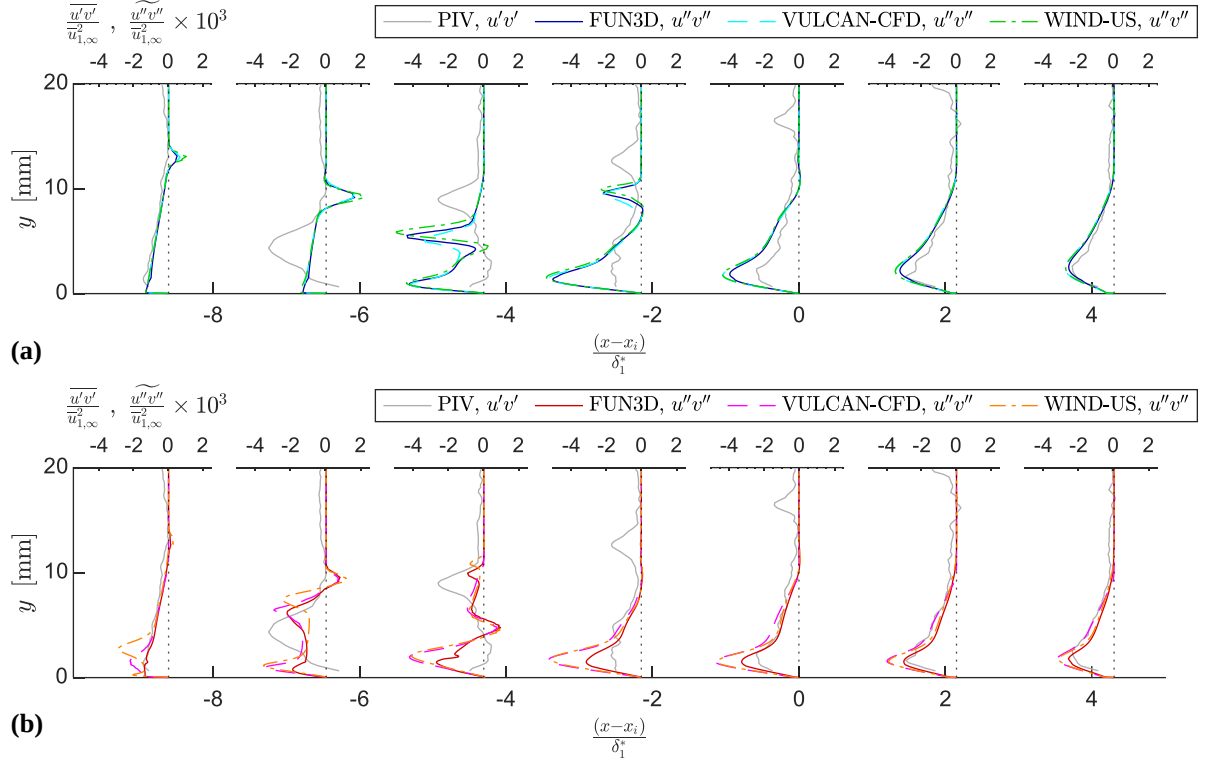


Figure 33. Turbulent shear stress profiles for the (a) SA turbulence model solutions and (b) SST-V turbulence model solutions for the 16° half-angle shock generator cases.

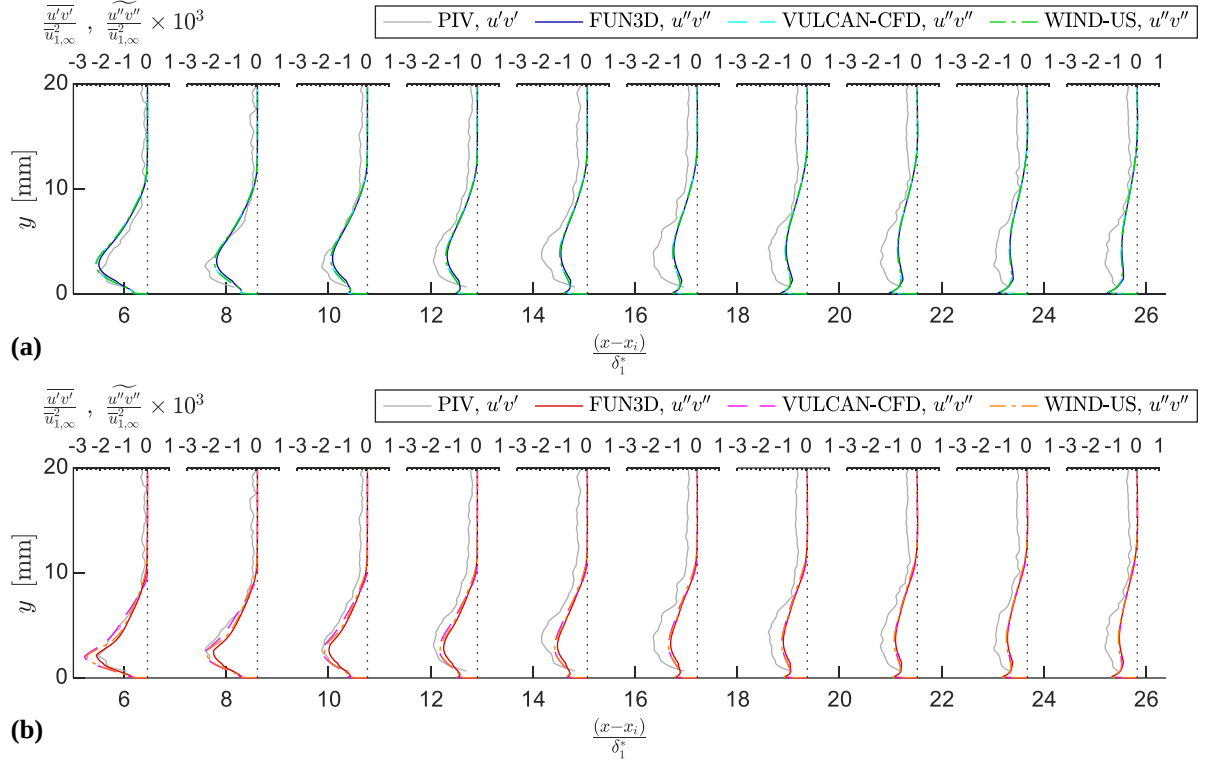


Figure 34. Downstream turbulent shear stress profiles for the (a) SA turbulence model solutions and (b) SST-V turbulence model solutions for the 16° half-angle shock generator cases.

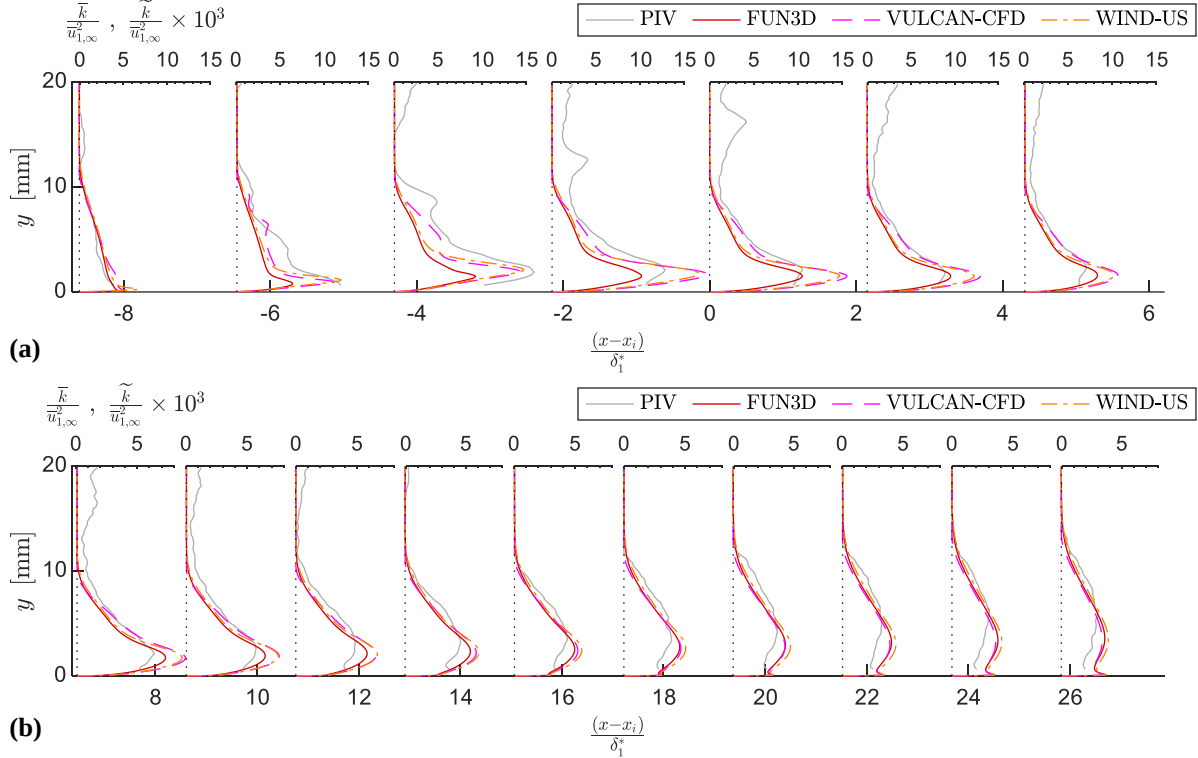


Figure 35. SST-V turbulence model solutions (a) turbulent kinetic energy profiles and (b) downstream turbulent kinetic energy profiles for the 16° half-angle shock generator cases.

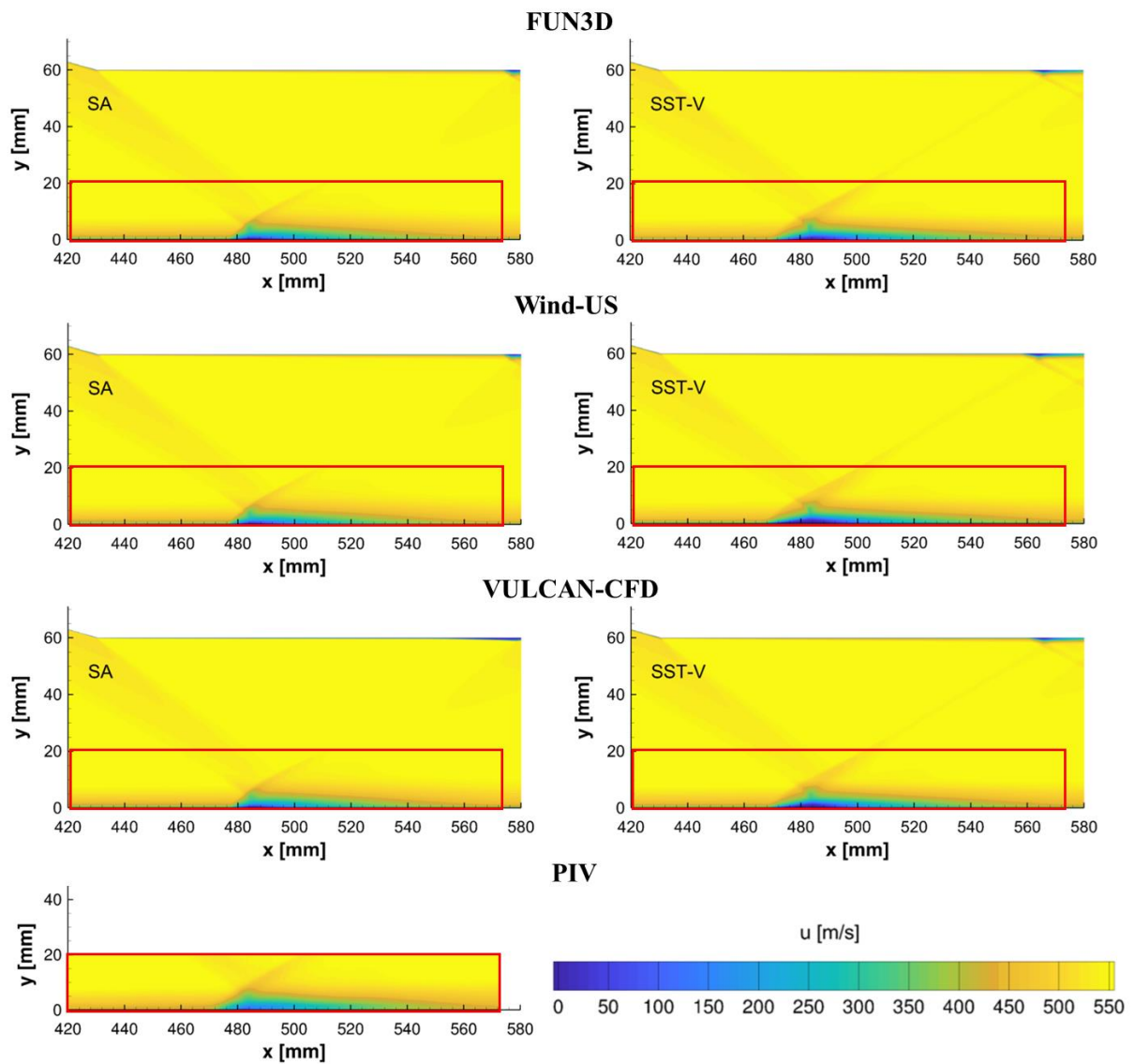


Figure 36. CFD test section axial velocity contours compared to PIV data.

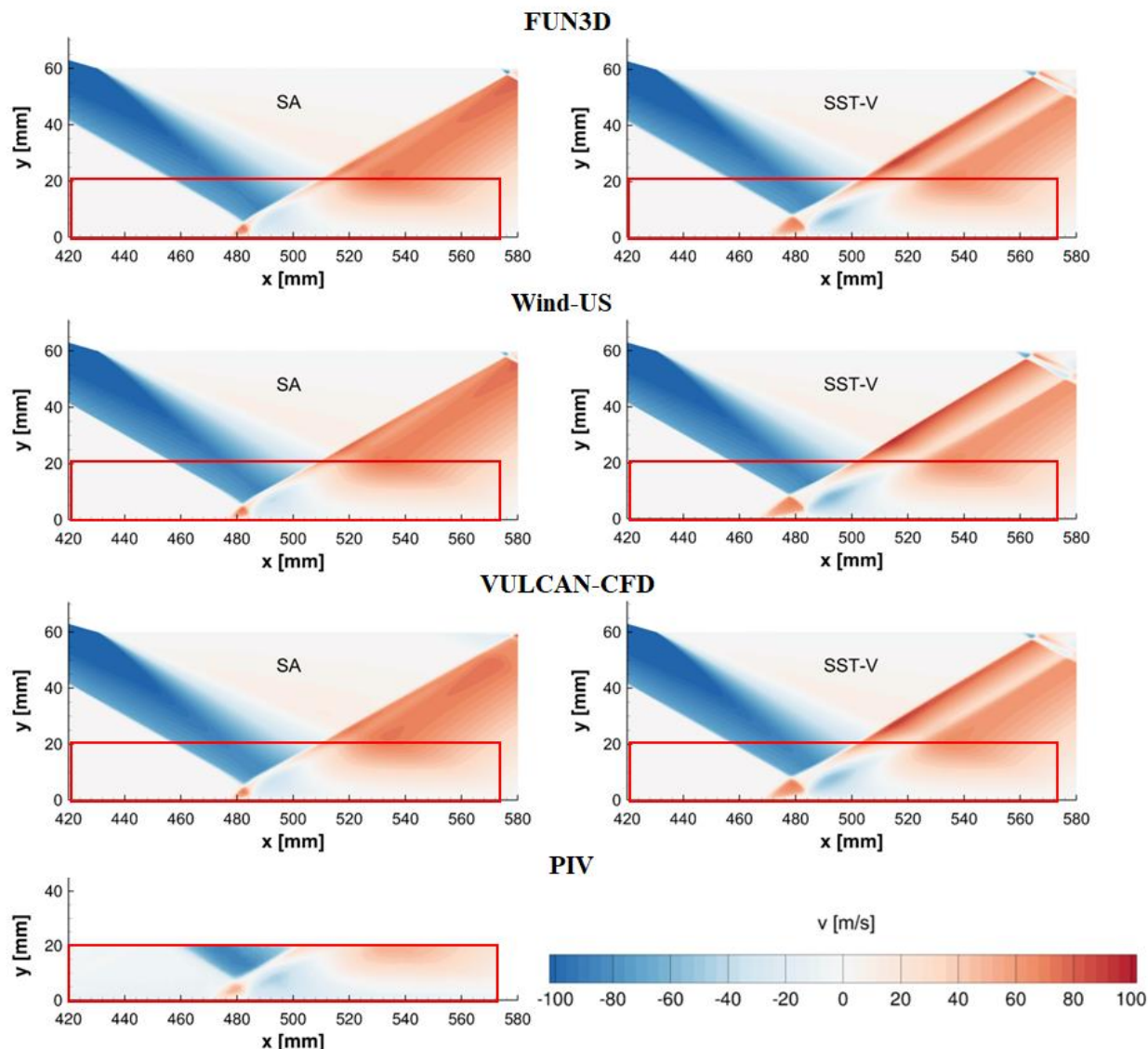


Figure 37. CFD test section radial velocity contours compared to PIV data.

VI. Conclusions

A study, based on the NASA Turbulent Computational Fluid Dynamics Validation Experiment (TCFDVE), was performed to 1) assess the capability of Reynolds-averaged Navier-Stokes (RANS) techniques to predict the physics of two of the Mach 2.5 axisymmetric shock-wave/boundary-layer interactions (SWBLIs) and 2) create a new validation case for the NASA Turbulence Modeling Resource (TMR). The study explored two shock generator half-angles: 10° and 16° . The RANS simulations were run with two of the more common eddy-viscosity turbulence models: the one-equation Spalart-Allmaras (SA) turbulence model and the two-equation Menter Shear-Stress Transport $k-\omega$ (SST) turbulence model. The formulation of the SST turbulence model with the vorticity-based turbulent production term (SST-V) was used. Three widely used CFD codes were used to verify the trends: FUN3D, Wind-US, and VULCAN-CFD. The Wind-US and VULCAN-CFD simulations utilized a 2D grid run in axisymmetric mode while the FUN3D simulations utilized a 3D quarter-sector grid.

A grid sensitivity study was performed that utilized the 2D axially symmetric grid and generated coarser grids by skipping every other point and every fourth point in each direction. The grid study demonstrated qualitative grid

convergence of the incoming boundary-layer profile and test section wall static pressure, independent of the turbulence model and flow solver used. The incoming boundary-layer integral quantities exhibited some minor non-monotonic convergence, owing to a sensitivity of the boundary-layer growth and resultant core flow edge quantities with respect to the grid density for internal flows. The wall shear stress values at the incoming boundary-layer station were shown to converge monotonically.

Code-to code comparisons were conducted on the fine mesh. From this it was noted that the CFD predicted boundary-layers qualitatively agreed well with each other and with the experimental Pitot probe measurements. Quantifying the comparison using the boundary-layer integral quantities demonstrated a larger spread in values for the SST-V turbulence model simulations compared to the SA turbulence model simulations. As expected, these trends were independent of the shock generator angle as the station is upstream of the shock generator in supersonic flow.

For the weaker SWBLI interaction of the 10° half-angle shock generator case, the predicted wall pressure, Pitot pressure, and mass flux profiles through the interaction region were relatively insensitive to the turbulence model and code used. The wall pressure profiles agreed decently well with the experimental measurements, with the CFD simulations predicting the start of the pressure rise downstream of the experiment location. Further, it was shown that CFD derived Pitot pressure profiles and mass flux profiles compared well against the experiment, with the experimental mass flux profiles being derived from the hot-wire probe measurements. The CFD also showed no SWBLI-induced separation, as evident by the wall skin friction coefficient profiles and in agreement with the experiment.

For the stronger SWBLI interaction of the 16° half-angle shock generator case, the predicted wall pressure for the SA turbulence model simulations were also relatively insensitive to the code used. When compared against the experiment, the SA turbulence model simulations 1) predicted the start of the pressure rise downstream of the experiment location and 2) overpredicted the peak pressure. The SST-V turbulence model simulations showed more of a code sensitivity to the prediction of the start of the pressure rise location, with Wind-US predicting the most upstream location and FUN3D predicting the most downstream location (which was in better agreement with the experiment). From the SST-V turbulence model simulation-derived wall pressure profiles it was also noted that the simulations underpredicted the peak pressure when compared to the experiment.

When looking at the skin friction coefficient profiles for the 16° half-angle shock generator case, the SST-V turbulence model simulations predicted a larger separation extent (and therefore a stronger SWBLI) compared to both the SA turbulence model simulations and the experiment. It was noted that 1) the experiment determined the separation and reattachment points from observing oil flow visualizations and 2) the discrepancy in the CFD predicted separation extent was due to the differences in the predicted separation point, as the predicted reattachment point was shown to be turbulence model insensitive. Evidence of the stronger SWBLI predicted by the SST-V turbulence model simulations was also seen in the velocity flow field around the interaction region, which was compared against the PIV experimental data.

The overall conclusions are 1) RANS can correctly predict the axisymmetric SWBLI flow physics when SWBLI-induced separation is not present, as evidenced by the good agreement between the CFD simulations and the experiment for the 10° half-angle shock generator case and 2) when SWBLI-induced separation is present, RANS is less able to accurately predict the axisymmetric SWBLI flow physics. The inability to correctly predict the flow physics when separation is present stems from an inability to correctly predict the turbulent stresses through the separated interaction region, which is a known deficiency of RANS modeling.

Acknowledgments

The authors would like to thank the NASA Transformational Tools and Technologies (TTT) Project for supporting this work and the NASA's High-End Computing (HEC) Program for providing super-computing resources.

References

- [1] Dupont, P., Piponniau, S., Sidorenko, A., and Debieve, J., "Investigation by Particle Image Velocimetry Measurements of Oblique Shock Reflection with Separation," *AIAA Journal*, Vol. 46, No. 6, 2008, pp. 1365-1370. doi: 10.2514/1.30154
- [2] Lapsa, A. P., and Dahm, W. J. A., "Stereo Particle Image Velocimetry of Nonequilibrium Turbulence Relaxation in a Supersonic Boundary Layer," *Experiments in Fluids*, Vol. 50, No. 1, 2010, pp. 89-108.

- doi: 10.1007/s00348-010-0897-x
- [3] Debonis, J. R., Oberkampf, W. L., Wolf, R. T., Orkwis, P. D., Turner, M. G., Babinsky, H., and Benek, J. A., "Assessment of Computational Fluid Dynamics and Experimental Data for Shock Boundary-Layer Interactions," *AIAA Journal*, Vol. 50, No. 4, 2012, pp. 891-903.
doi: 10.2514/1.J051341
 - [4] Benek, J. A., "Overview of the 2010 AIAA Shock Boundary Layer Interaction Prediction Workshop," AIAA 2010-4821, June 2010.
doi: 10.2514/6.2010-4821
 - [5] Benek, J. A., "Lessons Learned from the 2010 AIAA Shock Boundary Layer Interaction Prediction Workshop," AIAA 2010-4825, June 2010.
doi: 10.2514/6.2010-4825
 - [6] Hirsch, C., "Lessons Learned from the first AIAA-SWBLI Workshop CFD Simulations of Two Test Cases," AIAA 2010-4824, June 2010.
doi: 10.2514/6.2010-4824
 - [7] Babinsky, H., Oorebeek, J., and Cottingham, T. G., "Corner effects in reflecting oblique shock-wave/boundary-layer interactions," AIAA 2013-0859, January 2013.
doi: 10.2514/6.2013-0859
 - [8] Titchener, N., Bruce, P., and Babinsky, H., "An Experimental Investigation of Corner Bleed Applied to a Normal Shock-Wave/Boundary-Layer Interaction and Diffuser," AIAA 2011-69, January 2011.
doi: 10.2514/6.2011-69
 - [9] Settles, G. S., and Dodson, L. J., "Supersonic and hypersonic shock/boundary-layer interaction database," *AIAA Journal*, Vol. 32, No. 7, 1994, pp. 1377-1383.
doi: 10.2514/3.12205
 - [10] Davis, D. O., "CFD Validation Experiment of a Mach 2.5 Axisymmetric Shock/Boundary-Layer Interaction," *Proceedings of the ASME/JSMA/KSME 2015 Joint Fluids Engineering Conference*, Fluids Engineering Division Summer Meeting, Vol. Volume 1: Symposia, 2015.
 - [11] Sasson, J., "Conical Shock Wave Turbulent Boundary Layer Interactions in a Circular Test Section at Mach 2.5," Ph.D. thesis, Case Western Reserve University, 2022. URL: http://rave.ohiolink.edu/etdc/view?acc_num=case1648825433614363.
 - [12] Sasson, J., Reising, H. H., Davis, D. O., and Barnhart, P. J., "Summary of Shock Wave Turbulent Boundary Layer Interactions in a Circular Test Section," AIAA 2023-0442, January 2023.
doi: 10.2514/6.2023-0442
 - [13] Aeschliman, D. P., and Oberkampf, W. L., "Experimental Methodology for Computational Fluid Dynamics Code Validation," *AIAA Journal*, Vol. 36, No. 5, 1998, pp. 733-741.
doi: 10.2514/2.461
 - [14] Rumsey, C. L., Smith, B. R., and Huang, G. P. "Description of a Website Resource for Turbulence Modeling Verification and Validation," AIAA 2010-4742, June 2010.
doi: 10.2514/6.2010-4742
 - [15] Spalart, P. R., and Allmaras, S. R., "A One-Equation Turbulence Model for Aerodynamic Flows," *Recherche Aerospaciale*, No. 1, 1994, pp. 5-21.
 - [16] Menter, F. R., "Improved Two-Equation k-omega Turbulence Models for Aerodynamic Flows," NASA/TM-103975, October 1992.
 - [17] Mosele, J.-P., Gross, A., Slater, J. W., Davis, D. O., and Vyas, M. A., "Axisymmetric and Asymmetric Turbulent Shock Wave Boundary Layer Interaction at Mach 2.5," AIAA 2021-2763, August 2021.
doi: 10.2514/6.2021-2763
 - [18] Anderson, W. K., Biedron, R. T., Carlson, J.-R., Derlaga, J. M., Diskin, B., Druyor Jr., C. T., Gnoffo, P. A., Hammond, D. P., Jacobson, K., Jones, W. T., et al., "FUN3D Manual: 14.0.2," NASA/TM-20230004211, November 2023.
 - [19] Yoder, D. A., "Wind-US User's Guide Version 4.0," NASA/TM-2016-219145, September 2016.
 - [20] Baurle, R. A., White, J. A., O'Connell, M. D., Drozda, T. G., and Norris, A. T., "VULCAN-CFD User Manual: Ver. 7.2.0," NASA/TM-2022-0008781, November 2022.
 - [21] Baurle, R. A., White, J. A., Drozda, T. G., and Norris, A. T., "VULCAN-CFD Theory Manual: Ver. 7.2.0," NASA/TM-2022-0008776, November 2022.
 - [22] Pointwise, Software Package, Version 18.4 Release 4, Pointwise Inc., Fort Worth, TX, 2021.
 - [23] Edwards, J. R., "A Low-Diffusion Flux-Splitting Scheme for Navier-Stokes Calculations," *Computers & Fluids*, Vol. 26, No. 6, 1997, pp. 635-659.
doi: 10.1016/S0045-7930(97)00014-5
 - [24] Spalart, P. R., "Strategies for Turbulence Modelling and Simulations," *International Journal of Heat and Fluid Flow*, Vol. 21, No. 33, 2000, pp. 252-263.
doi: 10.1016/S0142-727X(00)00007-2
 - [25] Roe, P. L., "Approximate Riemann Solvers, Parameter Vectors, and Difference Schemes," *Journal of Computational Physics*, Vol. 43, No. 2, 1981, pp. 357-372.
doi: 10.1016/0021-9991(81)90128-5

- [26] Reising, H. H., Sasson, J., Davis, D. O., Friedlander, D. J., and Howerton, L. W., “Cross-Measurement Comparisons for CFD Validation Dataset on Mach 2.5 Turbulent Shock-Wave/Boundary-Layer Interaction Without Sidewall Effects,” AIAA Paper (to be published January 2025).
- [27] Georgiadis, N. J., Vyas, M. A., and Yoder, D. A., “Wind-US Code Contributions to the First AIAA Shock Boundary Layer Interaction Workshop,” NASA/TM-2013-217837, February 2013.
- [28] Reising, H. H., and Davis, D. O., “PIV Measurements of Shock/Boundary-Layer Interactions at Mach 2.5 in a Circular Test Section,” AIAA 2024-2554, January 2024.
doi: 10.2514/6.2024-2554
- [29] Reising, H. H., and Davis, D. O., “Development and Assessment of a New Particle Image Velocimetry System in the NASA GRC 225 cm² Wind Tunnel,” AIAA 2023-0631, January 2023.
doi: 10.2514/6.2023-0631