

Application of shock-unsteadiness model to interaction of transverse sonic jet and supersonic crossflow

Jagadish Babu Vemula[†], Tanisha Kishor Joshi[†] and Krishnendu Sinha[†]

[†]Department of Aerospace Engineering, Indian Institute of Technology Bombay, Mumbai, 400076

Transverse injection of an under-expanded sonic jet into a supersonic crossflow generates a complex flow field. The interaction of the turbulence in the incoming boundary layer and the jet shear layer with the shock waves lies at the heart of the phenomena. In this paper, we study these aspects using the shock-unsteadiness turbulence model that is developed using the physics of shock-turbulence interaction. Reynolds Average Navier-Stokes computations are performed for a sonic jet in a Mach 3.75 crossflow, and the results are compared with the experimental data available in the literature. The effect of the SU correction on the size of the separation bubble, surface pressure, and skin friction coefficient are presented. It is shown to improve results significantly compared to the standard $k - \omega$ model, without compressibility correction. The implementation details of the SU model parameters are also discussed.

Nomenclature

δ_0	Boundary-layer thickness upstream of interaction, m
∞	Free-stream condition
ω	Specific dissipation rate of turbulent kinetic energy, s^{-1}
ρ, p, T	Density, pressure, and temperature
a	Speed of sound
b'_1	Shock-unsteadiness damping parameter
c_f	Skin friction coefficient
k	Turbulent kinetic energy, m^2/s^2
S_{ii}	Mean dilatation
S_{ij}	Symmetric part of mean strain rate tensor
M	Mach number

I. Introduction

The injection of a sonic jet into a supersonic crossflow (JICF) is seen in air/fuel injection techniques used in scram-jet combustors and thrust vectoring of supersonic aircraft and rockets. The resulting flow field consists of complex flow features of shockwave/turbulent boundary-layer interaction including lambda shock, separation bubble, barrel shock, Mach disk, and secondary recirculation regions (see Fig 1). It becomes essential to numerically predict such features with reasonably good accuracy to determine their sensitivity to the geometric design and the extent of the jet mixing and penetration into a supersonic crossflow. The problem has simple geometric requirements but with a complicated flow structure, making it an excellent choice to test different turbulence models behaviour at high flow speeds.

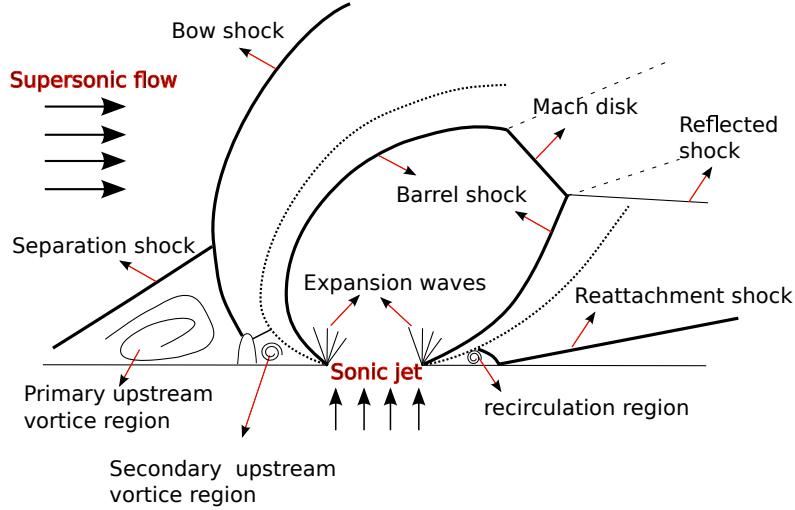


Figure 1: Schematic of sonic jet in supersonic cross flow with barrel shock formation within the jet boundary.

The injected jet, being under-expanded, expands into the crossflow and simultaneously deflected downstream by it. On the other hand, the crossflow sees this jet as an obstruction - this leads to the formation of a bow shock just upstream of the injection slot. Due to this strong jet induced bow shock, the incoming boundary layer is subjected to a high adverse pressure gradient causing it to separate and form a recirculation bubble. Two-three vortices are typically seen within this bubble, termed as primary, secondary, and tertiary vortices based on their relative sizes. There is also a separation shock present before the bow shock as a result of this recirculation zone.

The bow shock and the separation shock form a lambda shock pattern, visualized in Fig 1. Post bow shock, we have the free shear layer of the jet curving into the flow. The expansion fans arising within the under-expanded jet hit the jet boundary (shear layer) and reflect as compression waves. These compression waves merge and create a barrel-shock inside the jet boundary and a Mach disk terminates this barrel shock. The Mach disk is a normal shockwave like structure that will be seen as a plane in two-dimensional flow and a circular disk in three-dimensional flow. Downstream of the jet, there is again a recirculation zone and the supersonic flow over the downstream recirculation zone turns itself by formation a reattachment shock. The vortices (located upstream and downstream) have a more complex structure in the three-dimensional flow.

Several experiments have been conducted in the past for 2D & 3D cases,¹⁻⁵ and various RANS models have been assessed by many researchers using the data set from Aso *et al.*^{1,2} Some analytical calculations for the effective thrust force were also done. The primary challenge in getting experimental data is that the supersonic flow structure is highly susceptible to minute changes in the geometry and flow conditions. Thus, care must be taken that the measuring instruments cause minimum interference with the actual flow. Aso *et al.*^{1,2} carried out experiments for two-dimensional and three-dimensional mixing flow fields in supersonic flow induced by injected secondary flows through slot perpendicularly. Schlieren visualization of the shock structure and oil flow visualization on the plate are obtained to understand the shock and flow topology. Surface pressure distribution was measured from ports located along several lines at constant span-wise distances from the injection port.

Gaseous nitrogen jet is injected normally into the external flow of Mach 3.75 through a transverse slot nozzle mounted on a flat plate model. The freestream total pressure was 1.2 MPa, and the Reynolds number was 2×10^7 . The experiments were conducted with different widths of the slot and varying total pressure ratios of the jet to the free stream. The experimental setup was similar to that of Spaid and Zukoski.^{3,4} The set of experiments was performed with and without aerodynamic fences. It was observed that the fences had a significant impact on maintaining the two-dimensionality of the flow, thus improving the data set for validation with the numerical simulations.

In 1962, Rizzetta⁶ used an explicit MacCormack scheme²⁰ and $k-\epsilon$ equations for investigating the flowfield.

By using an explicit compressibility modeling, significant improvements were seen in matching upstream pressure rise and peak pressure region with experimental data. However, the reference data itself had shortcomings due to three-dimensionality effects because of aerodynamic fences absence. Also, there were a limited number of pressure tabs, and hence the experimental data was highly discontinuous. The scheme had no inclusion of the turbulent kinetic energy in the total energy term, and hence there would be a considerable error with this data.

Gerlinger *et al.*⁷ used a $q - \omega$ (where $q = k$) turbulence model because previous $k - \omega$ simulations had not been stable. Transport equations for q and ω were decoupled from the other transport equations for stability. Three types of corrections to the standard turbulence model were assessed. One is a limit on ω , the second is assigning a different value of a model constant in the ω -equation, and the third is the compressibility correction (same as that used in Rizzeta's model⁶). The effect of the first two modifications applied together was approximately the same as that due to the third alone. Only the first two modifications were retained. A central differencing scheme with a matrix for artificial dissipation was used and velocity magnitude scaling was done to keep dissipation low.

The injector region has several flow structures that cause Reynolds stresses to develop in different ways. Since two-equation eddy viscosity models cannot consider this anisotropy, Chenault and Beran⁸ examined whether improvements can be realized by using Reynolds stress transport models (RSTM). A Roe scheme with MUSCL (Monotone Upwind Scalar Conservation Law) was used, and there was no explicit compressibility modeling included. Like Gerlinger,⁷ they too used the old results of Aso *et al.*,¹ which has several drawbacks. The authors reflected that a better turbulence model is necessary and external factors such as the difference of boundary conditions in simulations versus experiments are to be blamed for the disparity in results at high injection pressure ratios.

Mathew and Sriram^{9,10} attempted to improve the results at high-pressure ratios by including low Reynolds number corrections and increasing the grid resolution. The necessity of the Reynolds number correction was explained due to the presence of recirculation zones, which are typically having a low Reynolds number. The grid refinement was made at the boundary layer and up to the Mach disk since that region eventually affects the upstream separation. They used a $k - \omega$ model with explicit compressibility corrections, as suggested by Wilcox.¹¹ At high-pressure ratios, the inclusion of this correction had stabilizing effects.

The Roe scheme calculated the inviscid fluxes, and a MUSCL scheme was used for extrapolating variables to get a higher-order accuracy. The Favre-averaged Navier-Stokes Equations were integrated to steady states using a six-stage, low storage Runge-Kutta method. Good results were achieved, even at large pressure ratios, leading to the conclusion that a simple two-equation turbulence model, when given corrections for compressibility and low Reynolds number, and a proper grid resolution at the boundary layer and the Mach disk, will give a reasonably good solution. $k - \omega$ results of Mathew and Sriram⁹ compared with the previous computation results were close to the RSTM solution, but show mismatch between the experimental data set and the numerical results.

The upstream separation length was usually under-predicted,^{6,8,9,12} and the peak pressures were higher in the simulations compared to the experiment.^{1,2} In all the previous simulations, it was seen that the prediction errors of upstream separation location increased at higher pressure ratios. An example is shown in Fig 2. The disparity increased with slot width also. The location and size of the upstream separation bubble have a strong influence on the flow pattern and altering its characteristics, thereby change surface properties like pressure, skin friction, and heat transfer.

The majority of improvements in predictions reported above are using compressibility corrections. These corrections to standard turbulence models are found to be useful in free-shear layers, for example, the one bounding the separation bubble. However, they can have an adverse effect in the undisturbed boundary layer upstream of the interaction. Coakley and Huang¹³ found that the model predictions deteriorate in a turbulent boundary layer by applying the compressibility corrections to the turbulence model. Furthermore, the turbulent kinetic energy in the boundary layer reduces by using the compressibility corrections in dilatational-dissipation. It thus decreases the skin friction coefficient compared to well-established correlations for zero pressure-gradient turbulent boundary layers.

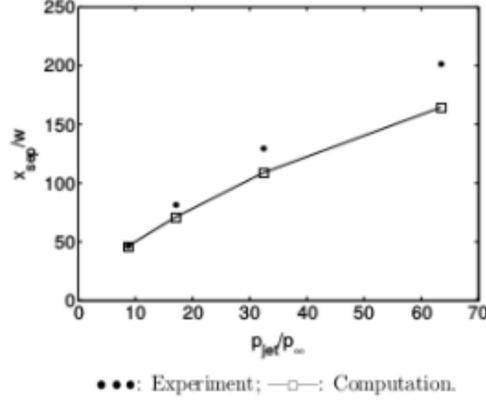


Figure 2: Increase in the error of simulations at higher pressures as studied by Mathew and Sriram.⁹

The discrepancies between turbulence model predictions and experimental measurements can be due to the physics of shock-turbulence interaction, which is not accounted for by standard RANS turbulence models, like $k-\epsilon$ and $k-\omega$.¹⁴ This leads to unsatisfactory prediction of the separation bubble size and peak heat transfer rates at reattachment. This drawback is seen in the current model problem of JICF, where the standard $k-\omega$ model highly under-predicts the upstream separation length and peak pressures. Sinha *et al.*^{14,15} point out that such variations from experimental results are due to the over-amplification of turbulent kinetic energy (k) across the shockwave by standard models. The overestimation of k leads to a more energized boundary layer, which can sustain the adverse pressure gradient created by the shock waves for longer, thus delaying flow separation. The over-amplification of k is partly because the standard models do not consider the unsteady nature of shockwave while interacting with turbulence. Upstream turbulence makes the shock oscillate about a mean position, and the amplification of turbulence across an unsteady shock is less compared to its amplification across a steady shockwave.

Sinha *et al.*¹⁵ presents a detailed study of shock-turbulence interaction (STI) and identify the significant physical mechanisms that determine the turbulence amplification at a shockwave. A new shock-unsteadiness (SU) effect is identified and modeled based on linear inviscid theory for STI. The shock-unsteadiness modified turbulence models significantly improve the flow topology, thereby giving better predictions for surface pressure and separation bubble size in flowfields with shock-shock and shock-boundary layer interactions.^{14,17,18}

In this study, we apply the SU $k-\omega$ model to the JICF flowfield and see the effect of SU correction in predicting the size of the separation bubble formed upstream of the injector. This is essential to study because the separated, recirculating region leads to high-pressure peak and high heat transfer rates near the injector. We present a thorough investigation in terms of SU-model parameter b'_1 , and evaluation of shock function f_s and upstream normal Mach number $M_{u,n}$ that are essential components of how the SU-model works in a complex flow.

II. Reference Experiments

Aso *et al.*¹ carried out experiments for two-dimensional mixing flowfield. Reynolds numbers based on the distance between the leading edge of the flat plate and the slot exit are 1.03 to 2.07×10^7 under the almost adiabatic wall condition. Experimental details from Aso *et al.*¹ are tabulated in table 1.

Table 1: Geometry and flow parameters of Aso *et al.*¹ experiments.

Geometry parameter	Value
Slot width	0.5 mm, 1 mm, 2 mm
Flow conditions	Value
Freestream Mach number, M_∞	3.75
Freestream total pressure, $P_{o\infty}$	1.2 MPa
Freestream total temperature, $T_{o\infty}$	289 – 299 K
Ratio jet pressure to free stream pressure, P_j/P_∞	8.57, 17.72 & 25.72

III. Methodology

A. Numerical scheme

The sonic jet interacting supersonic crossflow-field is simulated by solving the three-dimensional Reynolds-averaged Navier-Stokes (RANS) equations. Here Favre decomposition is used for all terms except density and pressure, as presented in Wilcox.¹¹ The standard $k - \omega$ model is used as the baseline model for turbulence closure. In addition, we use a well-validated in-house CFD code that has been employed in a variety of high-speed flow simulations in the past.^{14,17,18} The governing equations are discretized in a finite volume formulation where the convective fluxes are computed using a modified, low-dissipation form of the Steger-Warming flux splitting approach.^{16,20} Central differencing is used for the viscous fluxes and turbulent source terms. The turbulence models are fully coupled with the mean flow equations, and the CFD solver has been implemented in parallel using the Message Passing Interface (MPI). The numerical method is second-order accurate in both space and time. A full implicit matrix Data-Parallel Lower-Upper Relaxation method²¹ is used to integrate the equations until the steady-state is reached.

B. Shock-Unsteadiness (SU) correction

The modeling insights obtained in the canonical STI configuration have proven helpful in modeling the SBLI flows. For example, Sinha *et al.*¹⁵ implemented the SU modified turbulence model in a RANS simulation of a high-speed flow over a compression corner. The production term of the k equation was modified in three different turbulence models, $k - \epsilon$, $k - \omega$, and Spalart-Allmaras, based on the shock-unsteadiness factor. The damping effect of this added flow physics on the k amplification led to a correct prediction of the separation region size, wall pressure, and skin friction coefficient.

Here, the SU correction is incorporated in the $k - \omega$ turbulence model by multiplying the eddy viscosity term in the production term of the standard $k - \omega$ model by the factor,

$$c'_\mu = 1.0 - f_s \left(1.0 + \frac{2b'_1\omega}{\sqrt{3}S^*} \right) \quad (1)$$

where

$$S^* = \left[2S_{ij}S_{ji} - \frac{2}{3}S_{ii}^2 \right]^{1/2} \quad (2)$$

and f_s is an empirical shock function that locates the region of shock wave in a given flow field and is given by¹⁸

$$f_s = 0.5 - 0.5 \tanh \left(12 \frac{S_{ii}\delta_0}{U_\infty} + 4 \right) \quad (3)$$

where the mean dilatation S_{ii} is normalized by the freestream velocity U_∞ and incoming boundary layer thickness δ_0 .

The shock function f_s takes a value of one in shockwaves and high compression regions of the flow field, such that $c'_\mu = -2b'_1\omega/\sqrt{3}S^*$ and we get the modified production term that matches the production term in standard turbulence model. Otherwise, $f_s = 0$ is zero, and the standard turbulence model is recovered. Here the model parameter $b'_1 = 0.4(1 - e^{1-M_{u,n}})$, where $M_{u,n}$ is the shock-normal Mach number in the upstream flow. The shock-unsteadiness modification given by c'_μ is integrated into the standard $k - \omega$ framework by using a shock-identifying function (f_s).

C. Computational grid and boundary conditions

The computational domain is divided into three blocks: block 1- upstream, block 2- injector, and block 3- downstream. The x -axis lies along the freestream, the transverse direction is along y , and the z -axis is in the spanwise direction. Figure 3 shows the various zones (note that the leading edge is at $x = 0$ and the domain extends to 0.495m). The y -axis extends from 0 to 0.03m for all zones. The domain also spreads in the three-dimensional space with a z -axis length of about 10 mm, and five cells along the positive and negative z -direction. All dimensions are in meters unless mentioned otherwise.

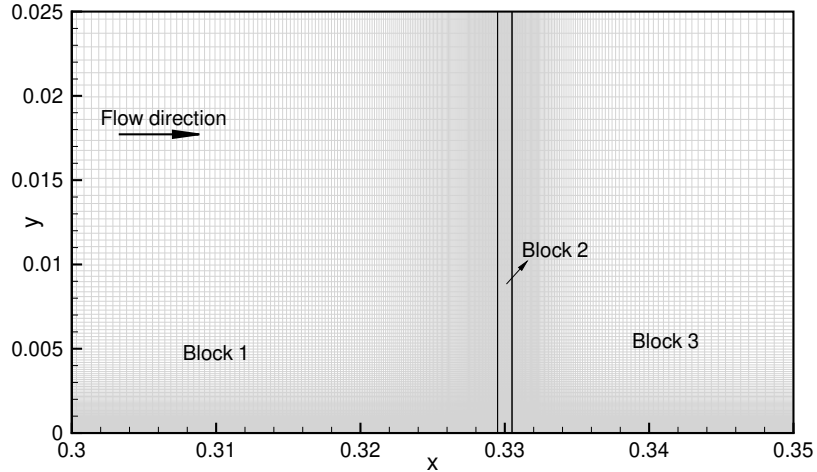


Figure 3: Grid showing multi-block used for the numerical simulations.

On the surface, no-slip condition for the velocity and constant wall temperature boundary conditions are specified. At the inflow boundary upstream of block 1, uniform inflow conditions are prescribed. The internal nozzle flow is not solved. Instead, the nozzle exit flow conditions are derived from one-dimensional isentropic flow corresponding to the stagnation pressure and temperature. The flow properties at the top and exit boundaries are extrapolated from those of the interior domain. Input conditions are given in Table 2. Since we have a three-dimensional domain, a periodic boundary condition at the z planes ensures the simulated flow's two-dimensionality. The wall is isothermal at 300K.

D. Grid requirement and convergence

It can be seen from Fig. 3 that the grid is clustered around the injector location and near the flat plate boundary. Almost 44% of the cells lie within 10% of the height above the plate. The cell height (Δy) near the wall is a minimum equaling $2.5\text{E-}07$ and exponentially increases to $5\text{E-}04$ at the top boundary. Along the freestream, almost 30% of the cells are clustered within 5mm upstream and downstream of the injector. The grid size along x (Δx) is a minimum of $2.5\text{E-}05\text{m}$ near the injector. Upstream it increases up to 0.006m near the leading edge and 0.005m

downstream at the flow exit boundary. Such a refinement is essential to capture the boundary layer accurately, flow separation zones, Mach disc, and barrel shock inside the jet boundary. Based on a grid convergence study

Table 2: Inflow parameters for the computation

Flow conditions	Value
Freestream Mach number, M_∞	3.75
Freestream static pressure, P_∞	11.09 kPa
Freestream static temperature, T_∞	78.43 K
Freestream total pressure, P_{o_∞}	1.2 MPa
Freestream total temperature, T_{o_∞}	299 K
Jet Mach number, M_j	1
Jet static pressure, P_j	196.52 kPa
Jet static temperature, T_j	249 K
Jet total pressure, P_{o_j}	0.372 MPa
Jet total temperature, T_{o_j}	298.8 K

(see Fig. 4), the solution obtained on a 490×120 grid is considered grid converged and is used to generate all the computations.

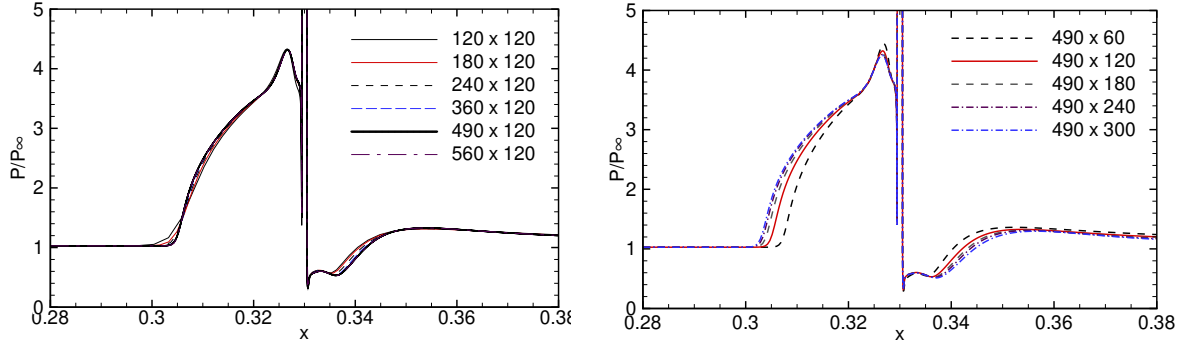


Figure 4: Grid independency study.

IV. Simulation results

For the jet pressure to free stream pressure ratio (NPR) of 17.72, $M_\infty = 3.75$ and 1mm slot width case, the Mach contour of the flowfield along with superimposing of surface pressure distribution and the skin friction coefficient is shown in Fig. 5. We see a sharp initial pressure rise due to the separation of the turbulent boundary layer induced by the adverse pressure gradient created by the separation shockwave. Then, a gradual plateau in pressure distribution follows it; this is the region of the primary upstream vortex locates. Next, another peak is seen in the wall static pressure due to the secondary upstream vortex downstream of the jet injection. Recorded peak pressure is 45% greater than the experimental value. As predicted, the pressure downstream of the jet is low, and it rises when the reattachment shock is encountered.

The separation length is highly underpredicted by standard $k - \omega$, 40% less than the actual length. Adverse pressure gradient of the separation shock decreases the skin friction coefficient from its undisturbed boundary layer value. C_f is negative in the recirculation bubble and attains high values in the jet interaction region. Additionally, downstream pressure distribution also has slight over predictions. Therefore the standard $k - \omega$ turbulence model fails to capture the essential physics of the shock-turbulence interaction. Despite this failure, the standard $k - \omega$

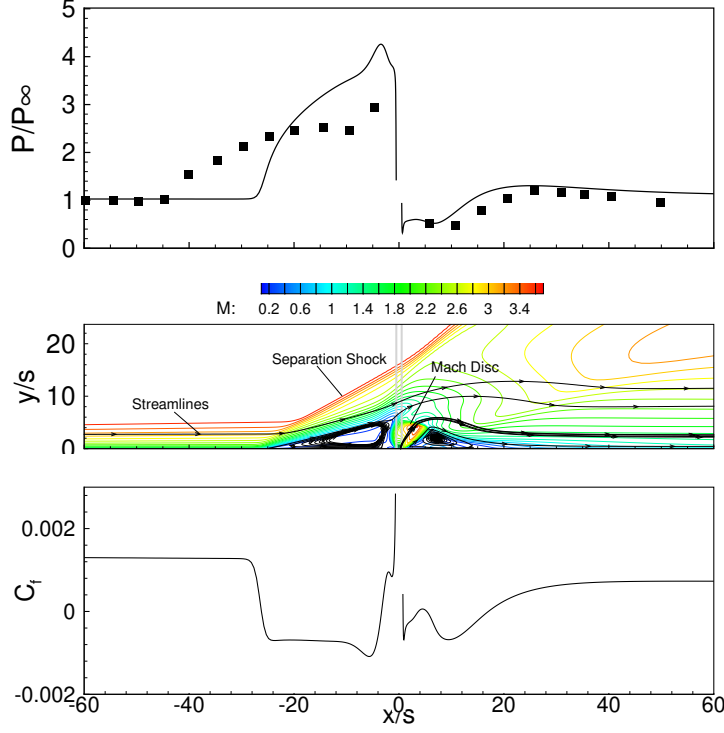


Figure 5: Numerical computed results - surface pressure distribution compared with experimental data^{1,2} (top), Mach contour of the flowfield (middle), and skin friction coefficient (bottom).

turbulence model can identify the different compression regions. It captures the lambda shock formations, barrel shock, Mach disk, and reattachment shock as shown by Mach contour in Fig. 5.

Thus, to improve the predictions, we implemented the SU modification to the standard $k - \omega$ model. Shock-unsteadiness correction is implemented with the model parameter b'_1 , which as a function of upstream normal Mach number to shock-waves $M_{u,n}$ and it is integrated using a shock identifying function f_s .

A. SU 2005 correction

The values of $M_{u,n}$ at different shocks are calculated manually by identifying an upstream location is very cumbersome.¹⁷ Applying the average values of b'_1 depending on the shock strength appeared in the flow field is a general practice used to apply the shock-unsteadiness modification.¹⁷ Similarly approach is used in the present work. The effect of the damping parameter, b'_1 on the surface pressure distribution is shown in Fig. 6. Here we used the empirical function f_s that locates the region of shock wave in terms of the ratio S_{ii}/S^* .¹⁴ It is defined as

$$f_s = \frac{1}{2} - \frac{1}{2} \tanh \left[5 \frac{S_{ii}}{S^*} + 3 \right] \quad (4)$$

The shock damping parameter, b'_1 is varied from 0.1 to 0.4. The upstream separation length slightly increases due to the impact of the SU modification. However, peak pressures remain same as that predicted by standard $k - \omega$ turbulence model. There is no noticeable effect of varying the shock damping parameter. Further the shock function f_s contour with $b'_1 = 0.28$ is shown in Fig. 7. The SU model is used in the regions where f_s is non-zero, and the baseline standard $k - \omega$ is used elsewhere. Depending on the level of compression, it smoothly varies between 0 and 1. The f_s contour do not accurately capture the shape of the upstream shocks.

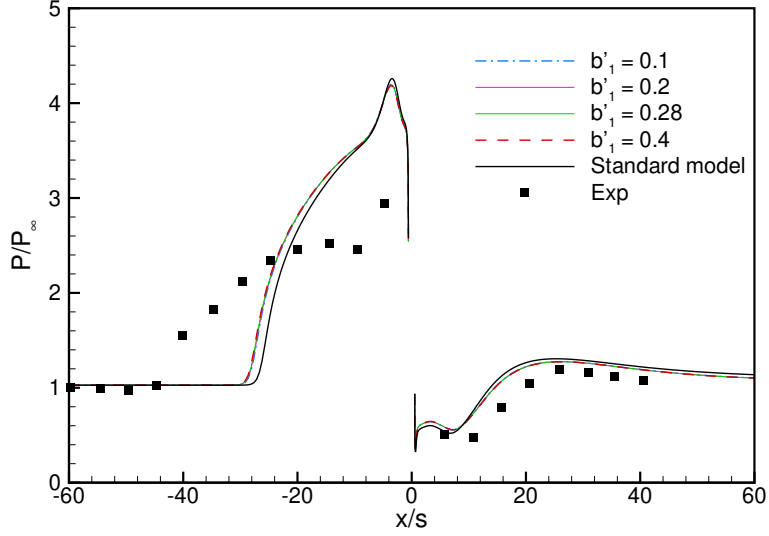


Figure 6: Wall pressure ratio plots - comparing 2005 model with standard model and experimental data.

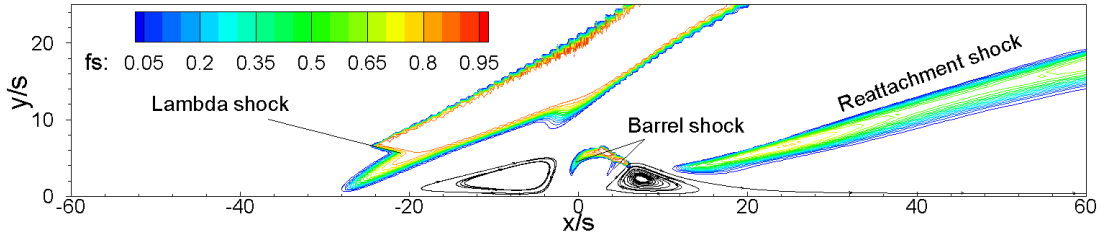


Figure 7: Shock function contour for SU 2005 model ($b'_1 = 0.28$).

B. SU 2008 correction

The original form of f_s proposed by Sinha et al.¹⁴ is a function of S_{ii}/S^* . Inside the boundary layer, S^* attains very high values reducing the magnitude of the ratio S_{ii}/S^* . Thus the function applied to the current configuration did not capture the separation shock accurately (see Fig. 7). In an alternative way, Pasha and Sinha¹⁷ replaced S_{ii}/S^* by $S_{ii}\delta_0/V$, where V is the magnitude of the local velocity vector. This form of f_s yields better reproduction of the separation shock. However, very close to the wall, it causes some difficulties in numerical evaluation of f_s as the magnitude of velocities are small. Next, the maximum magnitude of mean dilatation over the computational domain is used to normalize S_{ii} in the definition of f_s . Although this function identifies the shockwaves well, it still requires additional coding to identify the maximum value, especially for parallel simulations run on multiple processors. Finally, the ratio of S_{ii} to U_∞/δ_0 is used to define a function of the form given below.

$$f_s = \frac{1}{2} - \frac{1}{2} \tanh \left[12 \frac{S_{ii}\delta_0}{U_\infty} + 4 \right]. \quad (5)$$

Here, the characteristic length δ_0 and velocity U_∞ are a priori known for a given simulation and can be specified as user input.¹⁴ The constants chosen are such that f_s takes values close to one in strong shock regions and goes to zero outside shock waves. Depending on the level of compression, it smoothly varies between 0 and 1. The shock function (SU 2008)¹⁸ able to capture the shocks better than SU 2005 model.¹⁴ The shock topology computed using SU 2008 model is shown in Fig. 9.

Similar comparison in variation of shock damping parameter, b'_1 is done for SU 2008 f_s shock identifying function with two values of b'_1 (0.1 and 0.4). The results obtained using SU 2008 f_s are shown in Fig. 8 and compared with the previous models. There is a significant improvement in capturing the upstream separation location. The percentage error in its prediction is 20% lesser than the error arising from the standard $k - \omega$

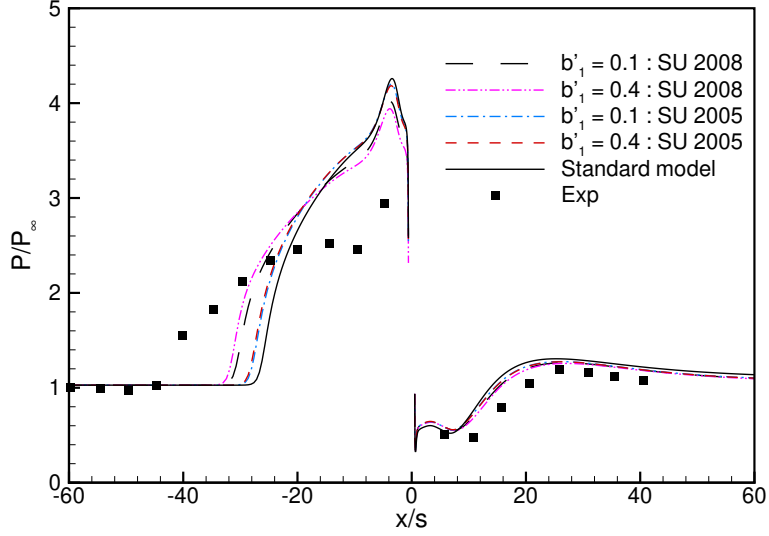


Figure 8: Wall pressure ratio plot for 2008 SU model ($b'_1 = 0.1$ and 0.4), compared with previous models.

model. Predictions in the peak pressures also considerably improve. The SU 2008 model is more sensitive to the variation of b'_1 as compared to the SU 2005 model. A higher value of shock damping parameter, b'_1 , results in earlier separation of the boundary layer. This phenomenon is consistent with the physics involved in the SU modification. Although, the SU 2005 model does not produce much effect of b'_1 on the peak pressures.

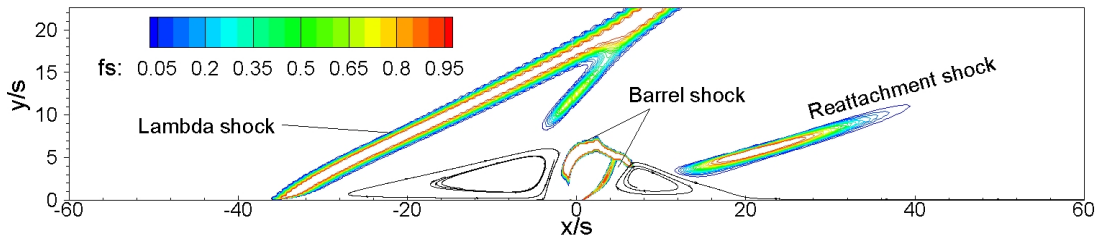


Figure 9: Shock function contour for 2008 SU model ($b'_1 = 0.4$).

C. Modified SU 2008 correction

From the above results, we observe that S_{ii} plays a significant role in identifying the shock structure. To quantify its effect, we examine the values of S_{ii} at the shock wave using SU 2008 shock function. See contour of Mach and S_{ii} in Fig. 10. We see that separation shock spreads within a region where S_{ii} varies from 30,000-60,000 s^{-1} . Therefore for the present case, this range of S_{ii} is important to capture the shock structure.

In the present section we investigate the impact of the ratio of $\frac{S_{ii}\delta_0}{U_\infty}$ on the shock structure by modifying the coefficients in the original SU 2008 shock function, such as,

$$f_s = \frac{1}{2} - \frac{1}{2} \tanh \left[a \frac{S_{ii}\delta_0}{U_\infty} + b \right]. \quad (6)$$

Here, we replaced the coefficients 4 and 12 of the f_s function with the variables a and b . First, a is held constant at 12, and b is varied to 3 and 5. Then, b is held constant at 4, and a is varied to 20 and 7. Here, the values of $\delta_0 = 0.0056$ m and $U_\infty = 665$ m/s. The effect of varying coefficients on the shock structure is shown in figure 11 using f_s contours. We understand that a higher value of coefficient a amplifies compression regions, leading to thicker shocks and earlier separation. A larger value of b results in thinner shocks, leading to the extended region of f_s with zero (lower compression). The values of $a = 12$ and $b = 3$ shows the better shock structure.

Comparison of the surface pressure distribution for the standard $k - \omega$ turbulence model, SU 2005, SU-2008

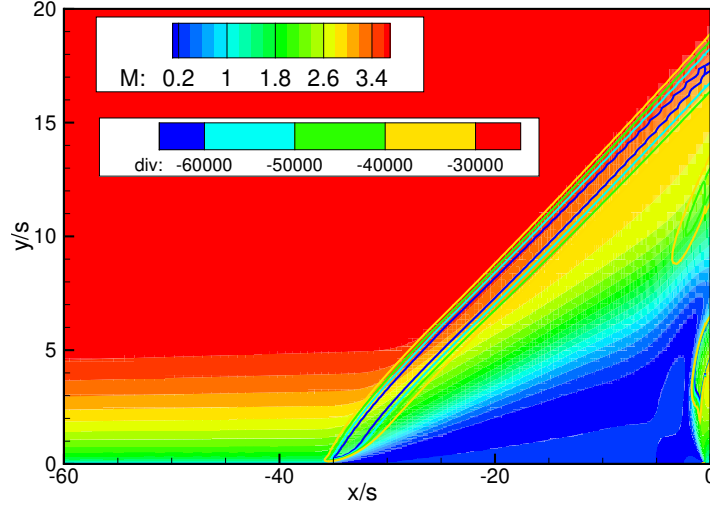


Figure 10: S_{ii} line contour superimposed over mach contour (SU 2008)

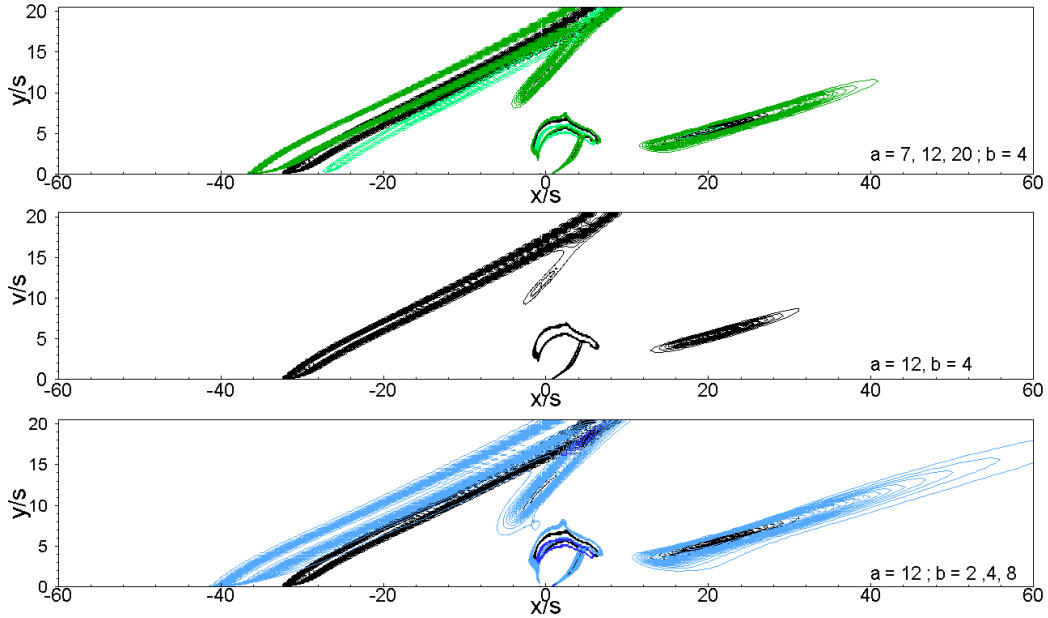


Figure 11: Effect of changing variables a and b on shock function contours (lighter shade = lower value)

with the variation of coefficients is shown in Fig. 12. The shock-unsteadiness modification appreciably improves the flow predictions in the interaction region compared to the standard turbulence models. The location of the separation and the pressure rise is much closer to the experiment for SU 2008 model than other models.

A quantitative comparison of the upstream separation location obtained with a modified version of the SU 2008 shock function with other shock functions is tabulated in Table 3. It lists the percentage error values with respect to the experimental data and the improvement obtained in predicting the upstream separation location for the SU corrections. Compared to the standard $k - \omega$ turbulence model, the model with the SU 2008 correction shows a good improvement in capturing the upstream separation location, which shows a 21 % error. The error percentage decreases from 40 % to 21 % compared to the standard $k - \omega$ model. Moreover, the modified SU 2008 shock function reduces the error percentage further to 16%. This significant improvement is obtained by varying the coefficients of a and b in the f_s shock function.

Figure 13 shows the comparison of the surface pressure distribution of the modified SU 2008 model¹⁷ with previous computations^{6,8,9} and experimental data.¹ All the models show a similar rise in upstream pressure rise

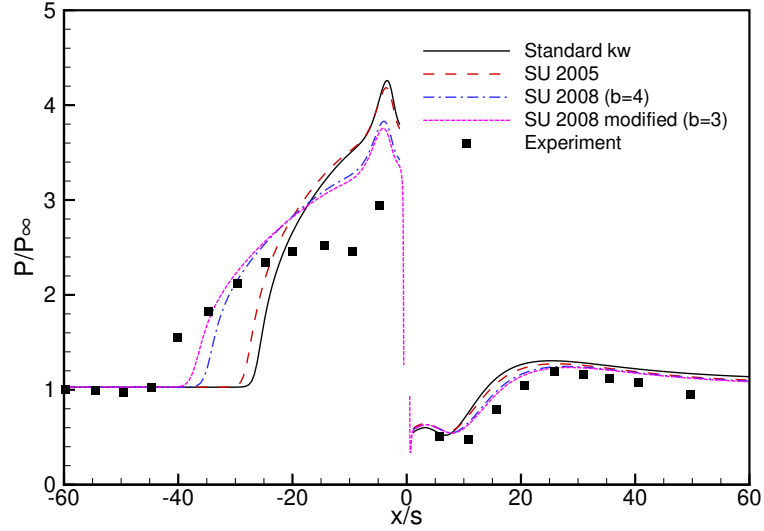


Figure 12: Comparing the modified SU 2008 model with standard, SU 2005, SU 2008 and experimental results.

due to separation shock and a gradual pressure rise over the separation bubble, indicating separation of a turbulent boundary layer. Also, a similar rise in pressure is seen over the secondary upstream vortex downstream of the jet injection and at the reattachment shockwave. The SU modified $k - \omega$ turbulence model captures the upstream separation more accurately than the $k - \epsilon$ simulation results of Chenault and Beran,⁸ and Sriram and Mathew⁹ $k - \omega$. by comparison, the RSTM results are closer to the experimental separation pressure rise location. The current modification shows a gradual rise in surface pressure over the separation bubble, but the peak pressure is comparable to other RANS results in literature. There is also a slight overprediction of the pressure downstream of reattachment. Overall, it is shown that the physics-based shock-unsteadiness correction to the $k - \omega$ turbulence model can give significant improvement in surface pressure prediction.

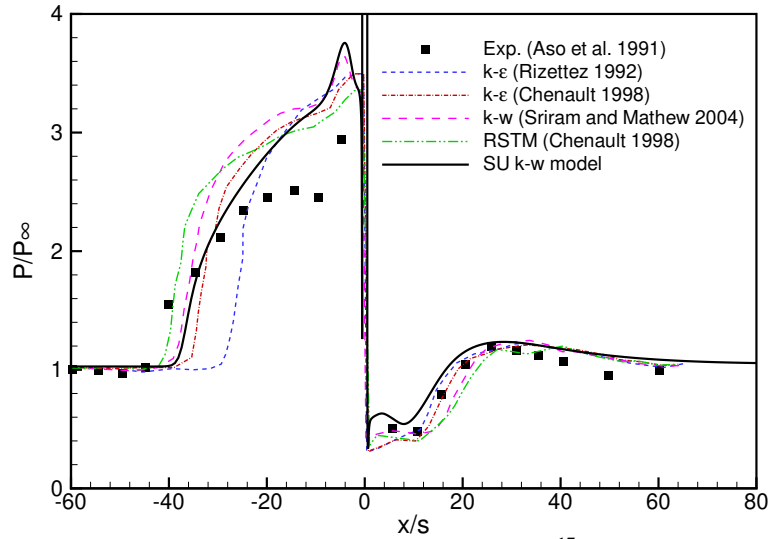


Figure 13: Comparison of surface distribution obtained by SU model¹⁷ with experimental data¹ and different turbulence models.^{6,8,9}

Table 3: Improvement obtained in prediction of upstream separation point

		Standard $k - \omega$	SU 2005	SU 2008	Modified SU 2008
X_{sep}	mm	-26.22	-27.98	-34.59	-36.87
Error	%	40	36	21	16

V. Conclusion

In this paper, we study the interaction of an under-expanded sonic jet with a supersonic cross flow at Mach 3.75, using the shock-unsteadiness (SU) turbulence model of Sinha *et al.*¹⁴ The model is specifically developed for complex turbulent flows with shock waves, and has previously shown significant improvement in shock-boundary layer interaction. It is based on the physics of shock-turbulence interaction, and the SU modification is applied only in the regions of a shock wave. This is in contrast to generic compressibility corrections that are active in the entire flowfield. A detailed comparison of the simulation results for the surface pressure and skin friction coefficient with the available experimental data¹ is presented. The performance of the SU model is gauged by comparing the upstream separation location with those for the standard $k-\omega$ model and other turbulence models in literature. It is found that the physics-based shock-unsteadiness correction significantly improves the separation prediction, and the results are comparable to other two-equation models using compressibility corrections and to the more elaborate Reynolds stress transport models. CFD implementation details of the SU turbulence model to the jet in crossflow problem is systematic evaluated. This includes a study of SU model parameters like shock-unsteadiness correction factor b'_1 , shock function f_s , and upstream shock normal Mach number $M_{u,n}$. It shows that capturing of the separation shock correctly and applying the SU correction appropriately is important to get accurate results.

References

- ¹ Aso, S., Okuyama, S., Kawai, M., and Ando, Y., "Experimental study on the mixing phenomena in supersonic flows with slot injection," AIAA Paper 91-0016, 1991.
- ² Aso, S., Okuyama, S., Ando, Y., and Fujimori, T., "Two-dimensional and three-dimensional mixing flow fields in supersonic flow induced by injected secondary flows through traverse slot and circular nozzle," In 31st Aerospace Sciences Meeting, pp. 489. 1993.
- ³ Spaid, F. W., and Zukoski, E., "Secondary injection of gases into a supersonic flow," AIAA journal, Vol. 2, No. 10, 1964, pp. 1689-1696.
- ⁴ Spaid, F. W., and Zukoski, E., "A study of the interaction of gaseous jets from transverse slots with supersonic external flows," AIAA journal, Vol. 6, No. 2, 1968, pp. 205-212.
- ⁵ Amick J. L., and Hawk, N. E., "Two-dimensional secondary jet interaction with a supersonic stream," AIAA Journal, Vol. 5, No. 4, 1967, pp. 655-660.
- ⁶ Rizzetta, D. P. "Numerical simulation of slot injection into turbulent supersonic supersonic stream". AIAA Journal, Vol. 30, No. 10, 1992, pp. 2434-2439.
- ⁷ Gerlinger, P., Algermissen, J., and Bruggemann, D. "Numerical simulation of mixing for turbulent slot injection". AIAA Journal, Vol. 34, No. 1, 1996, pp. 73.
- ⁸ Chenault, C., and Beran, P. S. " $k - \epsilon$ and Reynolds stress turbulence model comparisons for two-dimensional flows". AIAA Journal, Vol. 36, No.8, 1998, pp. 1401-1412.
- ⁹ Sriram, A., and Mathew, J., "Numerical prediction of two-dimensional transverse injection flows," In 42nd AIAA Aerospace Sciences Meeting and Exhibit, 2004, pp. 1099.

- ¹⁰ Sriram, A., and Mathew, J., "Improved prediction of plane transverse jets in supersonic crossflows," *AIAA journal*, Vol.44, No. 2, 2006, pp. 405-408.
- ¹¹ Wilcox, D. C., "Turbulence Modeling for CFD," DCW, La Canada, CA. 3rd edition, 2006.
- ¹² Sharma, V., Vinayak Eswaran, V., and Chakraborty, D., "Computational Analysis of Transverse Sonic Injection in Supersonic Crossflow Using RANS Models," *Journal of Fluids Engineering*, Volume 142, 2020, 061502-1.
- ¹³ Coakley, T.J., and Huang, P.G., "Turbulence modeling for high speed flows," *AIAA paper*, 1992, 0436.
- ¹⁴ Sinha, K., Mahesh, K. and Candler, G.V., "Modeling the effect of shock unsteadiness in shock/turbulent boundary-layer interactions," *AIAA Journal*, Vol.43, No.3, 2005, pp.586-594.
- ¹⁵ Sinha, K., Mahesh K., and Candler G. V., "Modeling shock unsteadiness in shock/turbulence interaction," *Physics of Fluids*, Vol.15, No.8, 2003, pp. 2290-2297.
- ¹⁶ Candler, G.V., Sinha, K., and Levin, D.A., "Numerical simulations of Atlas-II rocket motor plumes," 39th *AIAA Aerospace Sciences Meeting and Exhibit Reno*, 2001.
- ¹⁷ Pasha, A. A., and Sinha K., "Shock-unsteadiness model applied to oblique shock wave/turbulent boundary-layer interaction," *International Journal of Computational Fluid Dynamics*, Vol.22, No.8, 2008, pp. 569-582.
- ¹⁸ Pasha, A. A., and Sinha K., "Simulation of hypersonic shock/turbulent boundary-layer interactions using shock-unsteadiness model," *Journal of Propulsion and Power*, Vol.28, No.1, 2012, pp.46-60.
- ¹⁹ Roy, S., Pathak, U. and Sinha, K., "Variable turbulent Prandtl number model for shock/boundary-layer interaction," *AIAA J.*, Vol.56, No.1, 2017, pp.342-355.
- ²⁰ MacCormack, R.W., Candler, G.V., "The solution of the Navier-Stokes equations using Gauss-Seidel line relaxation," *Computers & fluids*, Vol.17, No.1, 1989, pp.135-150.
- ²¹ Wright, M. J., Candler, G. V., and Bose, D., "Data-Parallel Line Relaxation Method for the Navier-Stokes Equations," *AIAA Journal*, Vol.36, No.9, 1998, pp.1603-1609.