

Advanced two-equation turbulence models for computing shock-dominated flows in aerospace applications

Submitted in partial fulfillment of the requirements
for the degree of

Doctor of Philosophy

by

Pratikkumar Vikramark Raje
(Roll No. 144010010)

Supervisor:

Prof. Krishnendu Sinha



Department of Aerospace Engineering
Indian Institute of Technology Bombay

2021

Dedicated to my Mother

Acceptance Certificate

**Department of Aerospace Engineering
Indian Institute of Technology Bombay**

The thesis entitled “Advanced two-equation turbulence models for computing shock-dominated flows in aerospace applications” submitted by Pratikkumar Vikramark Raje (Roll No. 144010010) may be accepted for being evaluated.

Date: 2021

Prof. Krishnendu Sinha

Approval Sheet

This dissertation entitled “Advanced two-equation turbulence models for computing shock-dominated flows in aerospace applications” by Pratikkumar Vikramark Raje is approved for the degree of Doctor of Philosophy.

Examiners

Supervisor (s)

Chairman

Date: _____

Place: _____

Declaration

I declare that this written submission represents my ideas in my own words and where others' ideas or words have been included, I have adequately cited and referenced the original sources. I declare that I have properly and accurately acknowledged all sources used in the production of this dissertation. I also declare that I have adhered to all principles of academic honesty and integrity and have not misrepresented or fabricated or falsified any idea/data/fact/source in my submission. I understand that any violation of the above will be a cause for disciplinary action by the Institute and can also evoke penal action from the sources which have thus not been properly cited or from whom proper permission has not been taken when needed.

Date: 26th May, 2021

Pratikkumar Vikramark Raje
(Roll No. 144010010)

Abstract

Compressible turbulent flows are of primary interest in many of the technological applications including aerospace-related high-speed vehicles. Such flows often involve shock waves interacting with turbulence which pose significant challenges in terms of their physical modeling and numerical computation. Numerical predictions of these flows in practical engineering applications are carried out using Reynolds-averaged Navier-Stokes (RANS) framework. In this approach, the mean flow field is computed and the effect of turbulent fluctuations on the mean flow is introduced using a turbulence model. However, conventionally used turbulence models, like $k-\epsilon$, $k-\omega$, Menter SST $k-\omega$, etc., do not consider the physics of shock/turbulence interaction and generally give poor predictions of mean and turbulence fields for the shock-dominated flows. Such models are incapable to accurately resolve flows with large separation and recirculation and give inaccurate estimations of position of flow structures such as shock waves and surface quantities of engineering interest like surface pressure, skin friction, and base pressure distributions. In this work, two advanced turbulence models are developed to address the issues with the conventional models discussed above. Extensive CFD validation studies are performed to test the performance of these two models across a wide range of high-speed flow configurations involving separation and recirculation.

The shock-unsteadiness (SU) model proposed by Sinha et al. (Phys. Fluids, 15(8), 2003) is used widely in shock-dominated turbulent flow computations. However, the numerical implementation of this model is not straight-forward because of the non-local nature of the model parameters which stems from their dependence on the upstream mean flow Mach number. For the first advanced turbulence model, a physically consistent and numerically robust alternative to the SU model is developed. The model development finds its basis in the canonical shock-turbulence interaction problem and is based on the physical ideas of shock-unsteadiness damping. For the turbulent kinetic energy, a new model for the shock-unsteadiness source term that is physically and dimensionally consistent with the earlier formulation is proposed. An alternate form of the dissipation rate equation is also used to eliminate upstream dependence of the model parameter in

the production term. Numerical error in the model equations is eliminated by a transformation of variables. The new model is able to match the accuracy of the physics based shock-unsteadiness model for a range of canonical shock/turbulence interaction and shock/turbulent boundary layer interaction flows.

Menter SST $k-\omega$ is another widely popular RANS-based two-equation turbulence model routinely used in industries for predicting aerodynamic flows. It shows excellent performance for low-speed flows, but gives inconsistent predictions for high-speed shock-induced separated flows. The model assumption of using a constant value of 0.31 for the structure parameter contradicts experimental observations. The model is also unable to predict Reynolds stress anisotropy generated by shock waves. For the second advanced turbulence model development, the standard SST model is augmented with quadratic eddy viscosity formulation of an explicit algebraic Reynolds stress model. A new relation for the structure parameter is proposed, making it a function of the local mean flow velocity gradients and is no longer a constant in the regions of shock/turbulence interaction. Additional shock-physics is introduced using Sinha et al. (Phys. Fluids, 15(8), 2003) shock-unsteadiness model and an upper limit to the value of structure parameter is set in regions of shock waves.

The second advanced model, termed as SUQ-SST, is validated using a number of supersonic and hypersonic test cases including compression corner, impinging/reflected shock, and base flow configurations involving attached to fully-separated flows. Results show that the modifications do not alter the boundary layer prediction capability of the SST model. On the other hand, the new model gives significant improvement over the standard SST model in predicting mean flow properties, Reynolds stress anisotropy, flow separation and recirculation, and surface properties across the range of flow configurations considered. The model also shows a significant improvement in predicting the mean flow fields for a supersonic underexpanded jet. The model is finally applied to a realistic geometric configuration of four-nozzle clustered rocket engine configuration with multi-jet interaction typical of launch vehicles. Two different flow conditions, including a low and a high altitude case, are considered and the model predicts base pressure distribution reasonably well for both the cases.

Keywords : RANS; turbulence model; shock wave; shock/turbulence interaction; shock boundary layer interaction; shock-unsteadiness; turbulence kinetic energy; SST $k-\omega$ model; structure parameter; EARSM; backward-facing step; base flow; underexpanded jet; rocket nozzle

Table of Contents

List of Figures	xix
-----------------	-----

List of Tables	xxvii
----------------	-------

1	Introduction	1
1.1	Motivation	1
1.1.1	Shock-dominated flows : Practical importance	1
1.1.2	Numerical predictions : Issues with popular turbulence models	5
1.2	Objective	9
1.3	Scope	11
1.4	Organization	14
2	Background and Literature Review	17
2.1	Background	17
2.1.1	Navier-Stokes equations	17
2.1.2	Reynolds Averaging	18
2.1.3	Consequences of Reynolds averaging procedure	18
2.1.4	The closure problem	20
2.1.5	Favre-averaging	21
2.1.6	RANS equations for compressible flows	22
2.2	Linear eddy viscosity turbulence models	25
2.2.1	Boussinesq eddy viscosity hypothesis	27
2.2.2	Turbulence kinetic energy equation	28
2.2.3	Standard k - ϵ model	29
2.2.4	Standard k - ω model	30
2.2.5	Shock-unsteadiness k - ϵ and k - ω models	30
2.2.6	Menter's analysis : SST correction to eddy viscosity	31
2.2.7	Standard SST k - ω model	33
2.3	Explicit algebraic Reynolds stress models	34

2.3.1	EARSM by Wallin and Johansson [70]	36
2.3.2	EARSM by Rung et al. [73]	37
2.4	Literature Review	38
2.4.1	Canonical shock/turbulence interaction	39
2.4.2	Shock/turbulent boundary layer interaction	43
2.4.3	Supersonic flow over a backward-facing step	49
2.4.4	Supersonic axisymmetric base flow	50
2.4.5	Underexpanded supersonic jet	52
2.4.6	Multi-jet interaction and the reverse flow	54
2.5	Summary	58
3	Simulation methodology and validation cases	59
3.1	Canonical shock/turbulence interaction simulations	59
3.2	Two-dimensional and three-dimensional mean flow simulations	62
3.2.1	Finite volume formulation	63
3.2.2	Numerical method	65
3.2.3	Boundary condition formulation	71
3.3	UFAST impinging shock simulations	74
3.4	Schulein's impinging shock simulations	75
3.5	Holden's compression corner simulations	76
3.6	Backward-facing step simulations	77
3.7	Axisymmetric base flow simulations	78
3.8	Axisymmetric underexpanded jet computations	80
3.9	Multi-jet interaction simulations	82
3.10	Summary	88
4	Modeling of shock-turbulence interaction	89
4.1	Physics based shock-unsteadiness modification to $k-\epsilon$ model	89
4.2	A new $k-\epsilon$ model	91
4.2.1	Turbulence Kinetic Energy	91
4.2.2	Turbulent dissipation rate	94
4.2.3	Non-conservative errors at a shock wave	96
4.2.4	Conservative formulation	97
4.3	Comparisons with DNS	99
4.4	Extension to realistic flows	101
4.5	Local shock-unsteadiness $k-\epsilon$ model	105
4.6	Local shock-unsteadiness $k-\omega$ model	106

4.7	Numerical error at shock waves	107
4.8	2D supersonic compression corner simulations	108
4.8.1	Numerical implementation	109
4.8.2	Comparison with experiment	111
4.9	Impinging oblique SBLI simulations	115
4.9.1	CFD implementation	116
4.9.2	Comparison with experiment	117
4.10	Comparison of the computational time required	122
4.11	Summary	124
5	Development of an anisotropic SST turbulence model for shock-dominated flows	127
5.1	Menter SST $k-\omega$ model	127
5.2	Quadratic eddy viscosity formulation	128
5.3	Log layer analysis of EARSM	129
5.4	Modified Structure parameter	130
5.5	Analysis at a shock wave : SU limiter	132
5.6	A new shear stress transport model	133
5.7	The blending function F'_2	134
5.8	Zero-pressure-gradient flat plate boundary layer	135
5.9	Summary	137
6	Application of SUQ-SST model to shock- turbulent boundary layer interaction	139
6.1	UFAST Mach 2.3 impinging shock case	139
6.1.1	Mean velocity and Reynolds stresses variations	140
6.1.2	Structure parameter distribution	143
6.2	Mach 5 impinging shock cases	143
6.3	Mach 8 compression corner cases	146
6.4	Sensitivity to the SU limiter	148
6.5	Computational time comparisons	150
6.6	Summary	151
7	Application of SUQ-SST model to base flows	153
7.1	Backward facing step	153
7.1.1	Incoming boundary layer profile	154
7.1.2	Comparison of pressure distribution	155

7.1.3	Comparison of temperature distribution	156
7.1.4	Comparisons of mean quantities at different streamwise locations	158
7.2	Supersonic base flow	160
7.2.1	Comparison of Mach number distribution	162
7.2.2	Comparison of TKE distribution	163
7.2.3	Variation of mean velocities at different streamwise locations	164
7.2.4	Normal Reynolds stress variations at different streamwise locations	166
7.2.5	Base pressure and centerline velocity comparisons	168
7.3	Summary	169
8	Application of SUQ-SST model to multi-jet interaction	171
8.1	Supersonic underexpanded jet	171
8.1.1	Flowfield comparisons	172
8.1.2	Shock structures captured by the SUQ-SST model	173
8.1.3	Comparison of TKE distribution in the flowfield	173
8.1.4	Comparison of mean quantities along the jet centerline	175
8.2	Multi-jet interaction	177
8.2.1	Flowfield at two different altitudes	177
8.2.2	Flow evolution in the axial direction	179
8.2.3	Flow near the base shoulder	181
8.2.4	Flow near interacting jets	183
8.2.5	Distribution of turbulence kinetic energy in the flowfield	185
8.2.6	Centerline variations of mean velocity and pressure	185
8.2.7	Base pressure comparisons	185
8.3	Summary	188
9	Summary and Future work	189
9.1	Summary	189
9.2	Contributions	191
9.3	Future Work	192
A	Inflow conditions for canonical shock/turbulence interaction computations	195
B	Rotational invariance of local SU model	197
C	Standard SST model : k and ω transport equations	201
D	Performance of linear eddy viscosity based SST k-ω model variants	203

E	SUQ-SST model for two- and three-dimensional mean flows	205
E.1	Two-dimensional mean flows	205
E.2	Three-dimensional mean flows	207
F	Grid convergence studies	213
F.1	Canonical STI case	213
F.2	Two-dimensional SBLI cases	215
F.3	Back-facing step case	216
F.4	Axisymmetric base flow case	218
F.5	Supersonic underexpanded jet case	219
	References	220
	List of Publications	237
	Acknowledgements	239

List of Figures

1.1	Examples of SBLI in a typical scramjet vehicle with consequences. . . .	2
1.2	(a) Falcon 9 during ascent phase [Source : <i>Google</i>] and (b) RANS simulation of a four-nozzle clustered rocket engine configuration showing multi-jet interaction and reverse flow towards base [8].	3
1.3	RANS simulation of Saturn V performed by Gusman et al. [15] on the left with actual flight picture of Saturn V [Source : <i>Google</i>] on the right. .	4
2.1	A snapshot of transverse vorticity contours from DNS of a Mach 2 canonical shock/turbulence interaction showing turbulence amplification and shock wave undulations (Reproduced from [33]).	39
2.2	Streamwise evolution of TKE in a Mach 3 shock/turbulence interaction obtained using DNS and different turbulence models (Reproduced from [30]).	41
2.3	Variation of TKE amplification across a normal shock with upstream Mach number obtained from LIA [33] and DNS [58]; subscript ‘1’ denotes the upstream value and ‘2’ denotes the value downstream of the shock.	41
2.4	Schematic of (a) compression ramp SBLI and (b) impinging oblique shock/boundary layer interaction configuration	43
2.5	Comparison of (a) wall pressure and (b) skin friction distribution obtained using standard $k-\omega$ model and experimental data of Schulein (2006) for shock deflector angle of 14° and freestream Mach number of 5 (Reproduced from [35]).	45
2.6	The shock-impingement case with shock deflector angle of 14° and freestream Mach number of 5 shows that the standard $k-\omega$ model overpredicts the heat transfer-rate by about two times at the reattachment point compared with the experimental data of Schulein [93].	46
2.7	Schematic of a supersonic flow over a backward facing step [102]	49

2.8	Schematic of a supersonic axisymmetric base flow [109].	51
2.9	(a) Structure of a supersonic underexpanded jet exhausted into still air (Adapted from Pack 1948) (b) Schlieren image of a Mach 2.53 jet with NPR (P_j/P_∞) = 3.12, where P_j and P_∞ are the nozzle exit and freestream static pressure values, respectively (Taken from [36]).	53
2.10	A schematic showing the multi-jet interaction flowfield and the associated base flow for moderately underexpanded plumes.	55
3.1	(a) Two-dimensional control volume used in finite volume formulation, where the grid points are represented by black solid circles with capital indices of I and J and cell centers are shown by crosses represented by small indices of i and j and (b) Calculation of normal surface vector $\vec{S}_{i,j-1/2}$ [150].	64
3.2	The conservative variable $U_{i,j}$ at cell center calculated from the fluxes F and G passing through the cell boundaries in the x - and y -directions [150].	65
3.3	Physical interpretation for evaluation of conservative variable at cell-center, using fluxes at cell-interfaces, for the first-order method [150]. . .	69
3.4	Physical interpretation for evaluation of fluxes for the second-order method [150].	71
3.5	Computational domain along with boundary conditions for a flat plate configuration.	72
3.6	Schematic of (a) incident oblique shock wave/turbulent boundary layer interaction on a flat plate and (b) two-dimensional compression corner, with the computational domain marked by dashed line for the two configurations.	74
3.7	Structured 3-block computational Grid for the BFS simulations.	78
3.8	Computational domain and the structured 4-block grid used for axisymmetric base flow simulations.	79
3.9	Computational grid for the axisymmetric base flow computations as seen in different planes.	79
3.10	Structured 2-block grid used for the underexpanded jet simulations along with the boundary conditions used.	81
3.11	(a) Base model used for the experiments conducted by Musial and Ward [131], and (b) Base plate with four nozzles showing the symmetry lines where base pressure and heat flux were measured [131] (c) Geometric configuration showing the nozzle extension considered.	83

3.12	Surface grids showing the computational domain used for the multi-jet simulations : two views	85
3.13	Comparison of the curve fit and co-ordinates given by Musial and Ward [131].	86
3.14	Grid showing bell nozzle used for the multi-jet simulations.	87
3.15	Cylindrical body along with the bell-shaped nozzle used for the multi-jet computations.	87
3.16	3D view of the multi-block grid used for the multi-jet computations along with the boundary conditions employed.	88
4.1	Variation of the model parameter c with upstream Mach number.	92
4.2	TKE amplification for varying upstream Mach number as obtained for the present model with $c = 0.194$, shock-unsteadiness model [30] with exact b'_1 and $b'_1 = 0.28$	93
4.3	Amplification in the turbulent dissipation rate for different upstream Mach numbers as obtained using Eqs. (4.16) and (4.17).	95
4.4	Streamwise variation of TKE and turbulent dissipation rate in the Mach 1.5 shock/homogeneous turbulence interaction shows that all model predictions are comparable to the DNS data.	99
4.5	Spatial evolution of TKE and turbulent dissipation rate in the shock/homogeneous turbulence interaction for $M_1 = 2.5$	100
4.6	Streamwise variation of TKE and turbulent dissipation rate in the shock/homogeneous turbulence interaction for $M_1 = 3.5$	101
4.7	Amplifications in (a) TKE and (b) turbulent dissipation rate across weak oblique shocks obtained using the present model (solid lines) and the shock-unsteadiness $k-\epsilon$ model (dashed lines).	103
4.8	Comparison between the $k-\omega$ model solution and the approximated logarithmic function velocity and temperature profiles just upstream of the separation shock shows a good match.	109
4.9	Variation of (a) normalized TKE production and (b) normalized eddy viscosity for different versions of the $k-\omega$ model along a streamline originating at $y = 0.2\delta_0$ in the incoming boundary layer for the 24° case.	110
4.10	Comparison of wall pressure obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 24° case.	112
4.11	Comparison of skin friction coefficient obtained using the (a) $k-\epsilon$ and (b) $k-\omega$ model variants with the experimental data for 24° case.	112

4.12	Comparison of wall pressure obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 20° case.	113
4.13	Comparison of skin friction coefficient obtained using the (a) $k-\epsilon$ and (b) $k-\omega$ model variants with the experimental data for 20° case.	114
4.14	Comparison of wall pressure obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 16° case.	114
4.15	Comparison of skin friction coefficient obtained using the (a) $k-\epsilon$ and (b) $k-\omega$ model variants with the experimental data for 16° case.	115
4.16	Distribution of normalized mean dilatation in the computed flow-field for 14° shock generator angle. Here, δ_0 is the incoming boundary layer thick- ness and U_∞ is the free-stream velocity. Points are identified in the shock waves and upstream Mach number is obtained along the corresponding streamlines. (Taken from [150]).	116
4.17	(a) Normalized pressure and (b) shock detecting function f_s contours computed using the local shock-unsteadiness $k-\epsilon$ model for 14° shock generator angle.	118
4.18	Comparison of wall pressure as obtained using the different variants of (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 14° case.	119
4.19	Comparison of skin friction coefficient as obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 14° case.	119
4.20	Comparison of wall pressure as obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 10° case.	120
4.21	Comparison of skin friction coefficient as obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 10° case.	120
4.22	Comparison of wall pressure as obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 6° case.	121
4.23	Comparison of skin friction coefficient as obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 6° case.	122
5.1	Variation of the model parameters s_u and a_1'' with upstream shock-normal Mach number M_{1n}	133
5.2	Comparison of (a) Van Driest transformed mean velocity, (b) streamwise mean velocity, (c) mean density, and (d) mean temperature obtained using the standard SST and SUQ-SST models with the DNS data.	137

6.1	Contours of Mach number superposed upon normalized pressure to show the flow structure in the UFAST SBLI case	139
6.2	Comparison of the streamwise velocity, the streamwise, transverse, and shear components of Reynolds stresses at different streamwise positions : — Standard SST ; — SSG/LRR- ω [95]; — EARSM; — SUQ-SST; ○ Experiment.	141
6.3	Distribution of the structure parameter in the vicinity of shock/boundary layer interaction region : (a) Experimental data [51] and (b) SUQ-SST simulation	142
6.4	Distribution of the structure parameter in the Schulein's 14° SBLI flow-field obtained using using (a) standard SST k - ω model and SUQ-SST model (b) Eq. (5.13) and (c) Eq. (5.17).	144
6.5	Comparison of (a) wall pressure and (b) skin friction coefficient obtained using different k - ω models with the experimental data for varying shock generator angles : — Standard SST k - ω ; — EARSM; — SUQ-SST; ○ Experiment.	145
6.6	Distribution of the effective structure parameter a_1^* in the Mach 8 SBLI flowfield for different ramp angles	147
6.7	Comparison of wall pressure obtained using different k - ω models with the experimental data for the Mach 8 SBLI cases at varying ramp angles. . . .	148
6.8	Distribution of the effective structure parameter a_1^* in the Holden's 36° SBLI case obtained using SUQ-SST model with (a) SU limiter = 0.4; (b) SU limiter = 0.51; (c) SU limiter = 0.62	149
6.9	Sensitivity of wall pressure distribution to the SU limiter values for the SUQ-SST model for Holden 36° compression corner SBLI case	150
7.1	Mach number contours showing the supersonic flowfield over the BFS geometry obtained using the SUQ-SST model.	154
7.2	Comparison of the incoming boundary layer profile ($x/h = -1$) obtained using the standard SST, SUQ-SST turbulence models and the experimental measurements.	155
7.3	Comparison of mean pressure distribution in the BFS flowfield obtained using the standard SST, SUQ-SST turbulence models, and the experimental measurements.	156
7.4	Comparison of mean temperature distribution in the BFS flowfield obtained using the standard SST, SUQ-SST turbulence models, and the experimental measurements.	157

7.5	Comparison of the standard SST and SUQ-SST model predictions with the experiments at $x/h = 1.75$ (inside the separation bubble).	158
7.6	Comparison of the standard SST and SUQ-SST model predictions with the experiments at $x/h = 3$ (just after the experimentally observed reattachment).	159
7.7	Comparison of the standard SST and SUQ-SST model predictions with the experiments at $x/h = 6.67$ (downstream of the experimentally observed reattachment).	159
7.8	Mach contours showing the flowfield behind a cylindrical blunt base afterbody obtained using the SUQ-SST model.	161
7.9	Comparison of Mach number distribution in the axisymmetric base flowfield obtained using the standard SST, SUQ-SST turbulence models, and the experimental measurements.	162
7.10	Comparison of TKE distribution in the axisymmetric base flowfield obtained using the standard SST, SUQ-SST turbulence models, and the experimental measurements.	163
7.11	Normalized streamwise velocity variation at different axial locations in the Mach 2.46 axisymmetric base flowfield.	165
7.12	Normalized radial velocity variation at different axial locations in the Mach 2.46 axisymmetric base flowfield.	166
7.13	Normalized streamwise Reynolds stress τ_{xx} variation at different axial locations in the Mach 2.46 axisymmetric base flowfield.	167
7.14	Normalized radial Reynolds stress τ_{yy} variation at different axial locations in the Mach 2.46 axisymmetric base flowfield.	167
7.15	Comparison of the standard SST and SUQ-SST model predictions for the base pressure coefficient and centerline velocity with the experiments. . .	169
8.1	Comparison of the flowfield shown in terms of Mach number distribution obtained using different turbulence models for the Mach 2 underexpanded jet.	172
8.2	Distribution of the (a) mean dilatation and (b) boundary layer and shock detecting function F'_2 in the supersonic underexpanded jet flowfield. . . .	174
8.3	Comparison of TKE distribution obtained using different turbulence models in the Mach 2 underexpanded jet flowfield.	174
8.4	Comparison of TKE variation along the jet centerline obtained using different turbulence models for the supersonic underexpanded jet.	175

8.5	Variation of (a) normalized pressure and (b) Mach number along the jet centerline as obtained using different turbulence models and the experimental measurements of Seiner and Norum [155, 156] for the Mach 2 underexpanded jet.	176
8.6	Mach number contours showing the multi-jet interaction flowfield at two different altitudes : (a) low altitude case with NPR = 990 and (b) high altitude case with NPR = 3760. The flowfield is shown in a plane passes through the centers of the diametrically opposite nozzles.	178
8.7	Density gradient magnitude showing different flow-features in the multi-jet interaction flowfield captured by the simulations using SUQ-SST model.	179
8.8	(a), (c), (e), and (g) Mach number contours and (b), (d), (f), and (h) pressure contours at different xy cross-sections along different axial locations in the multi-jet interaction flowfield (Refer Fig. 8.7 for their exact locations).	180
8.9	(a), (c) Mach number and (b), (c) Pressure (Pa) contours showing the expansion at the base shoulder in the yz symmetry plane. Here, (a) and (b) correspond to the low altitude case, and (c) and (d) show the multi-jet interaction flowfield at the simulated high altitude.	182
8.10	(a), (c) Mach number and (b), (d) Pressure (Pa) contours showing the interacting jet-boundary expansion in the yz symmetry plane for the multi-jet interaction cases. Here, (a), (b) represent the low altitude case, and (c), (d) correspond to the high altitude case.	183
8.11	Comparison of the distribution of TKE in the multi-jet interaction flowfields as obtained using the SUQ-SST model at (a) low altitude and (b) high altitude. Shocks are identified using the negative dilatation contours shown in black color.	184
8.12	Comparison of normailzed (a) streamwise velocity and (b) static pressure along the vehicle centerline obtained using SUQ-SST model for two different altitude conditions.	186
8.13	Base pressure distribution obtained using SUQ-SST model at two different altitudes : (a) low altitude case with NPR = 990 and (b) high altitude case with NPR = 3760.	186
8.14	Comparison of radial variation of normalized base pressure between the SUQ-SST model and measurements for the two altitude cases. The results are shown along a symmetry plane passing midway between the centers of two adjacent nozzles.	187

B.1	Original XY and rotated $X'Y'$ co-ordinate system with shock orientation.	198
D.1	Comparison of wall pressure obtained using different variants of SST $k-\omega$ model with the experimental data for (a) Schulein's 14° impinging shock and (b) Holden's 36° compression corner cases.	204
F.1	Effect of grid refinement on (a) TKE and (b) turbulent dissipation rate evolution for upstream Mach number of 2.5.	213
F.2	Amplification in (a) ϵ and (b) k computed using the non-conservative $k-\epsilon$ equations on successively refined grids for a range of upstream Mach number.	214
F.3	The conservative $k-\epsilon$ equations show a systematic grid convergence in (a) ϵ and (b) k amplifications for all Mach numbers.	215
F.4	Results obtained on three different grids using SUQ-SST model for the UFAST SBLI case.	216
F.5	Grid convergence results for back-facing step case	217
F.6	Results obtained using the different grids for the axisymmetric base flow case.	219
F.7	Centerline pressure variation for two finer grids used in the supersonic underexpanded jet computations.	219

List of Tables

2.1	Basic flow structure interactions with their salient features observed in a multi-jet interaction flowfield	56
3.1	Freestream conditions for the SBLI test cases as reported in the experiments.	75
3.2	Geometry and flow parameters for the multi-jet interaction cases.	84
3.3	Chamber pressure to ambient static pressure ratio (P_c/P_∞), Nozzle exit pressure to ambient static pressure ratio (P_j/P_∞), and freestream conditions	84
3.4	Table of co-ordinates for Bell-shaped nozzle contour [131].	86
4.1	Error (in %) in TKE amplification for the present model with $c = 0.194$ and Sinha et al. [30] with $b'_1 = 0.28$, computed with respect to the shock-unsteadiness model presented in [30].	93
4.2	Normalized values of k and ϵ at the inlet station for different Mach number cases.	98
4.3	Comparison of the computational time (on 32 processors) required by the different variants of $k-\epsilon$ and $k-\omega$ turbulence models for the 14° and 10° incident oblique shock wave turbulent boundary layer interaction cases. .	123
5.1	Free stream and wall temperature conditions for the DNS case	136
F.1	Comparison of grids used for computing the supersonic flow over the BFS geometry.	217
F.2	Comparison of grid sizes in meters in the important regions in the structured grid used for BFS simulations.	217
F.3	Comparison of grids for the supersonic base flow case.	218
F.4	Comparison of cell sizes in meters in the crucial regions in the structured grid used for base flow computations.	218

Chapter 1

Introduction

1.1 Motivation

Interaction of shock waves with turbulence is important in many technological applications such as the high-speed flight of aerospace vehicles like scramjets, launch vehicles, etc. [1–8] and Inertial Confinement Fusion in fusion energy research [9–11]. The interaction is also of importance in various astrophysical systems, e.g. supernova explosions [12–14]. Such phenomenon can be seen in the form of shock/turbulent boundary layer interaction (SBLI) or shock/turbulent free-shear layer interaction (SSLI). If we focus our attention on the aerospace-related applications, shock waves usually interact with turbulence present in the highly sheared parts i.e. the boundary layers on the surface of the component/vehicle body and the wakes that combine the boundary layers following their separation from vehicle trailing edges. The rocket exhaust jets, especially underexpanded supersonic jets, also exhibit interaction between the shocks and turbulence in the jet boundary/mixing layers.

1.1.1 Shock-dominated flows : Practical importance

Shock-turbulent boundary layer interaction (SBLI) in case of high-speed aerospace vehicles can be found over aircraft wings, deflected control surfaces, compressor passages, around turbomachinery blades, wing-body/fin-body junctions, and in air-breathing engine inlet ducts. Figure 1.1 shows schematic of a typical scramjet vehicle along with the regions where SBLIs occur. Typical SBLIs in such a scramjet configuration involve shock waves generated over inlet ramps interacting with turbulent boundary layer over the surface, cowl shock impinging on the turbulent boundary layers in the isolator region, recompression shock wave interacting with the turbulent boundary layers over a backward-facing step in the combustor regions, etc. Such interactions can result in flow

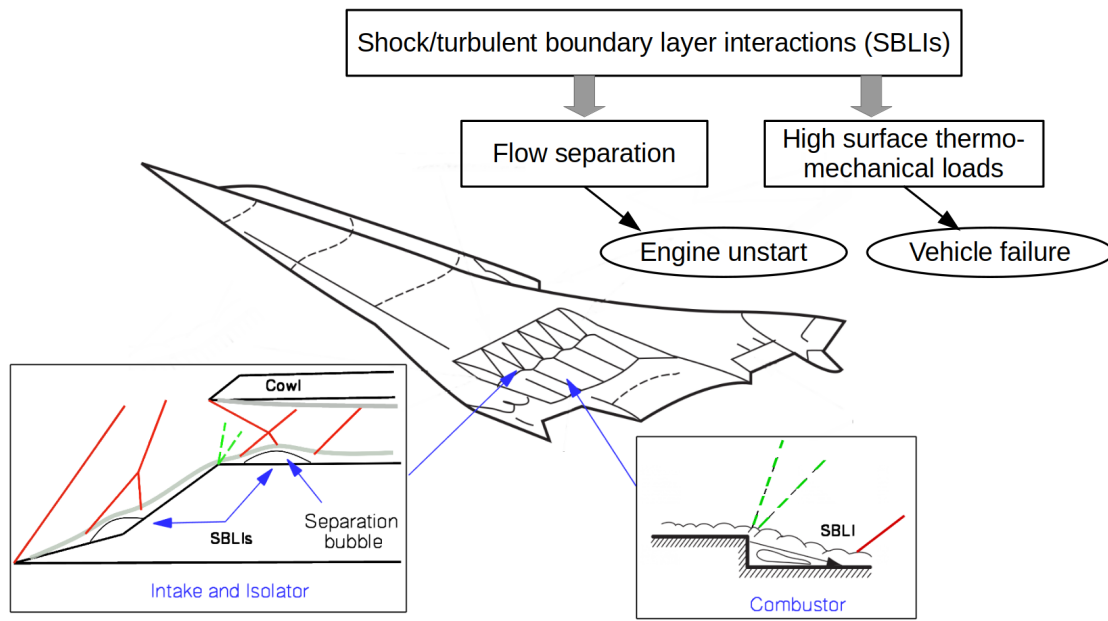


Figure 1.1: Examples of SBLI in a typical scramjet vehicle with consequences.

separation due to the adverse pressure gradient created by shock waves leading to a complex flowfield.

SBLIs can have a detrimental effect on the working of the high-speed vehicle. For example, the resulting separation bubble in the isolator section of the scramjet engine (see Fig. 1.1) can act as a blockage to the incoming air required for the combustion and thus may lead to engine unstart. Also, a shock wave interacting with a turbulent boundary layer enhances the turbulence intensity, which amplifies mixing and dissipation of turbulence kinetic energy (TKE). This causes a decrease in the stagnation pressure, resulting in the decrease in vehicle's efficiency due to the significant increase in drag. SBLIs are generally associated with significantly high thermo-mechanical loading, especially near the flow reattachment location over the component/vehicle surface, and thus can result in overall vehicle failure. Thus, SBLIs limit the vehicle's performance and can also cause structural damages, making their prediction imperative for the design of high-speed vehicles.

Launch vehicle is another important application where interaction between shock waves and turbulence plays a crucial role. Modern day launch vehicles generally use clustered engine configuration for higher thrust and to carry larger payloads, for example Falcon 9 uses nine-clustered first-stage Merlin engines (see Fig. 1.2(a)). During the ascent phase of such launch vehicles, the hot exhaust jets from the adjacent engine nozzles interact with each other owing to large radial expansion due to their underexpanded state at high altitudes. This can be seen for the Falcon 9 in Fig. 1.2(a). This multi-jet interac-

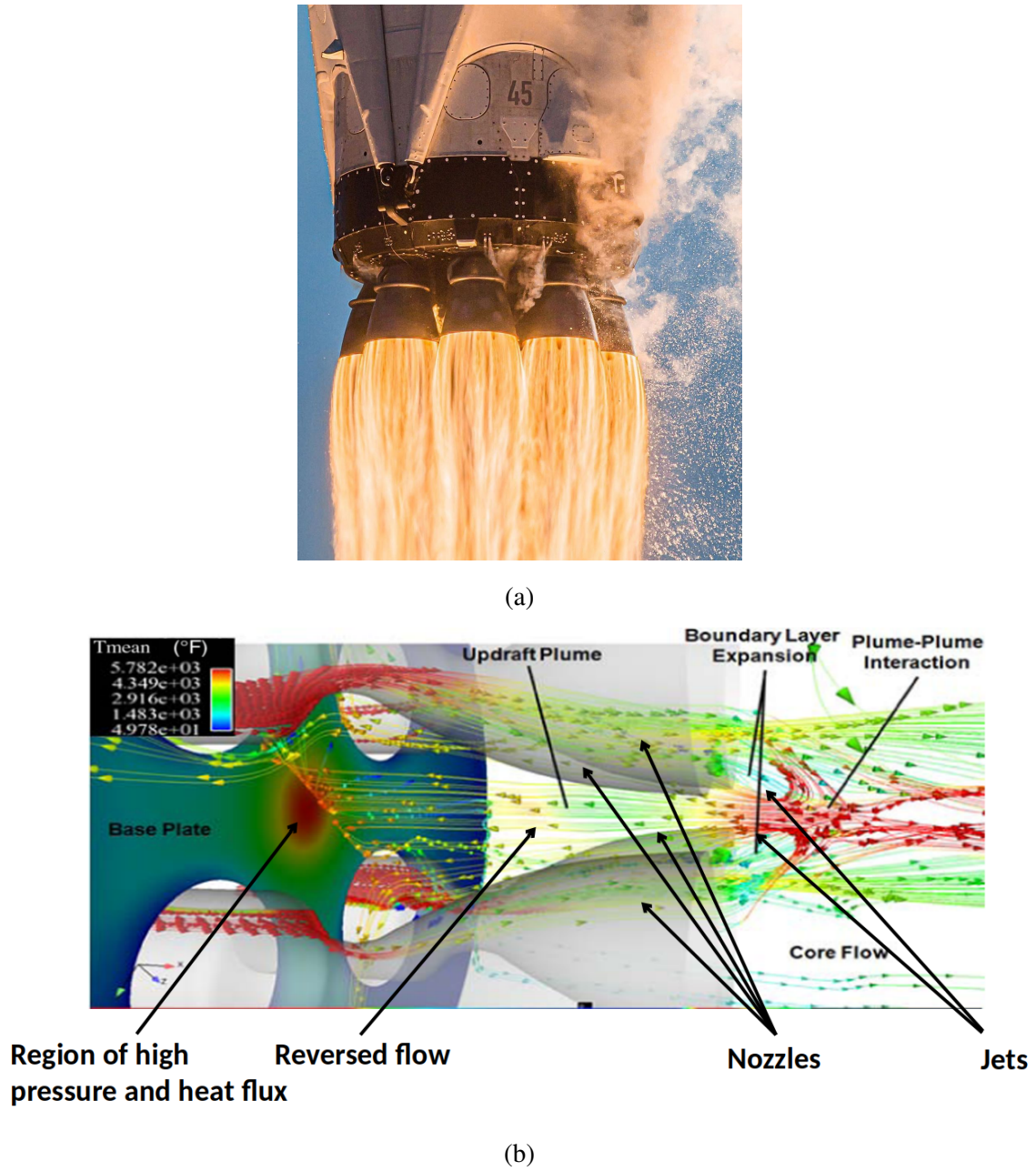


Figure 1.2: (a) Falcon 9 during ascent phase [Source : *Google*] and (b) RANS simulation of a four-nozzle clustered rocket engine configuration showing multi-jet interaction and reverse flow towards base [8].

tion together with the interactions between plume and atmosphere result in significantly complex flowfields.

Figure 1.2(b) shows the CFD results, taken from Mehta et al. [8], for a four-nozzle rocket engine configuration. The multi-jet or plume-plume interaction is characterised by the presence of strong shock-turbulent shear layer interactions (SSLI). This can result in the reverse flow (called as the “updraft plume”) of the hot exhaust gases towards the base.

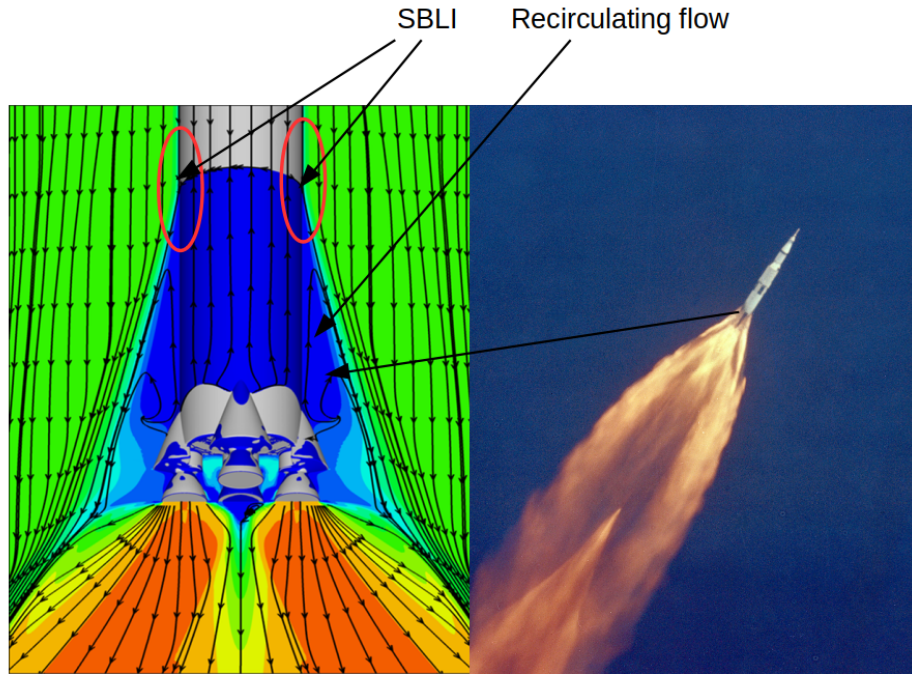


Figure 1.3: RANS simulation of Saturn V performed by Gusman et al. [15] on the left with actual flight picture of Saturn V [Source : *Google*] on the right.

Depending on the strength of interaction, the reversed flow of exhaust gases can even reach supersonic speeds. The impingement of exhaust gases on the rocket engine base creates a stagnation region causing significant thermal and momentum gradients within the base region. As a result, substantial values of heat flux and pressure are present in the base. Such extreme environments in the base region can have deleterious effects on the vehicle performance and can cause change in drag/thrust, severe base heating, loss of stability, etc. As a result, such flows are critical to assess and predict to prevent vehicle failure.

The complexity of the flowfield over a launch vehicle further increases with the increase in altitude. At very high altitudes, the radial expansion of the interacting plumes can become considerably large and the interaction between plumes and atmosphere become significant. This can result in SBLIs over the fuselage inducing a massive flow separation (in the literature this phenomenon is called as Plume Induced Flow Separations (PIFS)). The resulting recirculation bubble can bring up the very hot exhaust gases upwind of the base and threaten the integrity of the structure and equipment. Such PIFS were observed on several launchers such as Saturn V; see Fig. (1.3) which shows the CFD simulations along with an actual flight picture of Saturn V.

1.1.2 Numerical predictions : Issues with popular turbulence models

Shock waves interacting with turbulence thus play a key role in determining the aerodynamic performance of high-speed vehicles. Numerical prediction of this phenomenon is therefore important for the better vehicle design. The limitations of the present day computational resources restrict the Direct Numerical Simulation (DNS) and/or Large eddy simulations (LES) to grid resolutions that cannot fully resolve all of the salient structures present in such complex flows, especially for high Reynolds numbers. Therefore, numerical predictions of such flows in practical high-speed applications are generally carried out using Reynolds-averaged Navier-Stokes (RANS) framework [16–21]. In this approach, the mean flow field is computed and the effect of turbulent fluctuations on the mean flow is introduced using a turbulence model. The choice of turbulence model plays a crucial role in determining the accuracy of results.

In general, turbulence modelling is intuitive in nature and heavily relies on proper calibration and validation and the model performance is extremely sensitive to seemingly minor variations in modelling details and flow conditions. This makes turbulence modelling an especially demanding component of Computational Fluid Dynamics (CFD), however it is crucially important for the correct prediction of complex flows. This is particularly true for the shock-induced separation and recirculation, which is arguably the most challenging flow feature in aerospace-related CFD from a computational point of view. The modelling challenges posed by the type of flows discussed above are rooted in the complexity of the strain field and the high rates of change to which the mean flow as well as the turbulence fields are subjected.

As turbulent flows contain length-scales and time-scales that can dramatically change from one flow configuration to another, two-equation turbulence models represent the simplest level of Reynolds stress closure that can be formulated in a geometry independent fashion. In two-equation turbulence models the transport equations are solved for two turbulent fields that are directly related to the length-scales and time-scales. Linear eddy-viscosity (LEVM) based k - ϵ and k - ω turbulence models are two of the most popular group of two-equation models in the CFD community over the years. However, these models give inaccurate representation of shock/turbulence interaction and show poor performance in case of high-speed separated flows. For example, in case of SBLIs wherein interaction between strong shock waves and turbulent boundary layers leads to large separation, they yield significant errors in the predictions of the size and location of the separation region, surface pressure, and heat-transfer rates. Also, in case of SSLIs, for ex-

ample in case of an underexpanded supersonic jet, standard $k-\epsilon$ gives incorrect mean flow decay caused by the evolution of the mixing layer. Majority of the research work [22–28] has therefore been directed to improve the performance capabilities of these models in such flows. A detailed discussion about the latest trends and challenges can be found in Gaitonde [29], Zheltovodov [5], and Dolling [6].

The standard $k-\epsilon$ model predicts a very high amplification of turbulence kinetic energy (TKE) compared to direct numerical simulation (DNS) data across a shock [30, 31]. The over-amplification of TKE is due to the excessive production of turbulence at the shock wave. Sinha et al. [30] argues that one of the main reasons for the high level of turbulence production is the breakdown of eddy-viscosity assumption in the highly non-equilibrium region of a shock wave. A realizable model [28, 32] suppresses eddy-viscosity and brings down the post-shock turbulence level, but the predictions are still appreciably higher than the DNS data. Setting eddy viscosity to zero in the shock wave matches DNS data for weak shocks, but the predictions are still too high for strong shock waves. The above discussion is also true in case of the $k-\omega$ model. This suggests that these models do not incorporate some of the important physics involved when turbulence interacts with a shock wave.

DNS studies show that a shock wave undergoes unsteady oscillations when it interacts with turbulent fluctuations [33]. Sinha et al. [30] study the physics of the unsteady motion of a shock wave and suggest improvements in the standard $k-\epsilon$ model. The turbulence kinetic energy k equation is modified to include a term due to shock-unsteadiness, which is modeled using linear interaction analysis. This term represents the damping effect of shock oscillations on the amplification of TKE. The dissipation rate ϵ equation is similarly altered based on the linear theory results. Modifications in the k and ϵ equations result in the model parameters to be functions of the upstream Mach number normal to the shock. This dependency of the model parameters on the upstream flow variables arises due to the two basic underlying assumptions used to model the shock-unsteadiness effect:

(a) unsteadiness of shock is caused by turbulent fluctuations in the upstream flow and

(b) amount by which shock experiences unsteady motion depends on shock strength and hence the upstream Mach number M_1 .

These are fair assumptions considering that the physics of shock-turbulence interaction depends significantly on the strength of the shock wave and incoming turbulence.

Shock-unsteadiness (SU) model in the $k-\epsilon$ and $k-\omega$ modelling framework has been successfully applied to SBLI flows in the recent past. The SU $k-\epsilon$ and $k-\omega$ models give improved predictions of the size of separation bubble and the wall pressure distribution

compared to the corresponding standard models [27, 34, 35]. Improved predictions are also obtained for the SSLI in case of the supersonic underexpanded jet [36]. Better representation of the SBLIs and SSLIs results in a better prediction of the flow topology in the shock/turbulence interaction region, in terms of the location and pattern of shock waves.

CFD implementation of the SU $k-\epsilon$ and $k-\omega$ models for realistic applications is, however, difficult and cumbersome. The model parameters in the SU model are functions of the upstream Mach number normal to the shock, making the model implementation non-local in nature. To solve the model equations at points in a shock region therefore requires information about the flow variables at points upstream of the shock wave. The variation of the shock-normal Mach number (M_{1n}) at points upstream of a shock wave is not known a priori. A procedure to evaluate the shock-normal Mach number upstream of a shock wave is presented for simple compression corner configurations in [34]. However, in complex situations where multiple shock waves and shock-shock interactions are present, e.g. hypersonic cone-flare configurations [27], oblique shock impingement cases [35], and underexpanded jets [36], finding the correct variation of the upstream shock-normal Mach number becomes considerably complicated. Overall, the SU model is a promising candidate for the prediction of high-speed flows with SBLIs and SSLIs, but the model suffers from CFD implementation issues and thus can be improved further.

The SU model by Sinha et al.[30], as discussed above, finds its basis on theoretical grounds, i.e. LIA, and specifically targets to include shock/turbulence interaction physics in the standard $k-\epsilon$ and $k-\omega$ models. In the past, there have been a number of different routes followed by researchers to improve the prediction capabilities of the $k-\epsilon$ and $k-\omega$ models in case of separated flows. One of the most significant development in this area is the one by Menter [37]. The SST $k-\omega$ model continues to be arguably the most widely used and popular turbulence model in the CFD community. The model is developed on the basis of empiricism and combines the desired features of standard $k-\omega$ model near the wall in a boundary layer and standard $k-\epsilon$ everywhere else in the flowfield. Thus, SST can be regarded as a zonal model. However, in contrast to the classical zonal approach, it does not require an a priori knowledge of the flowfield in order to define zonal boundaries, where different models are to be used. The switching between different sub-models is achieved by ‘smart’ functions that can distinguish between the different zones.

Menter’s analysis of the standard two-equation turbulence models shows that the classical formulation of the eddy viscosity μ_T violates Bradshaw’s relation that the principal turbulent shear stress is proportional to the turbulence kinetic energy (TKE) in a boundary layer [38]. This shortcoming is attributed to the failure of standard two-equation models in case of adverse pressure gradient boundary layers. Menter modified the defi-

nition of eddy-viscosity so as to enforce Bradshaw's relation. A modification to μ_T that forces the principal turbulent shear stress to be proportional to TKE simulates the transport effect of the principal shear stress. Hence, the resulting model is called shear stress transport (SST) model.

The standard SST k - ω model gives excellent predictions for incompressible and low-speed subsonic and transonic flows with adverse pressure gradient and separation [39]. However, in case of supersonic and hypersonic flows, the model gives inconsistent predictions for the shock-induced separation yielding incorrect separation size and location along with inaccurate surface pressure and skin friction distribution [2, 21, 40, 41]. The model assumes a constant value of principal Reynolds shear stress over turbulence kinetic energy and the ratio is called as the structure parameter (a_1). The assumption is based on incompressible flat plate turbulent boundary layer in equilibrium [42–45]. Interestingly, even in the case of compressible attached flows, the structure parameter is found to be near to the SST model's standard value of $a_1 = 0.31$ for most parts of the boundary layer including the log layer [46, 47]. However, for high-speed flows with shock-induced adverse pressure gradients, experimental evidence suggest otherwise [47–51]. In fact, experiments show that the value of structure parameter increases in the regions of SBLI. Thus, attempts to modify the value of structure parameter in an ad-hoc manner to improve the separation predictions of the SST model can be found in the literature [52–55].

Another major limitation of the SST model is that it employs a linear relation between the Reynolds stress and the mean strain rate tensor, known as Boussinesq hypothesis. Linear eddy viscosity models do not represent correct turbulence physics in the sense that they are not able to predict anisotropy in normal Reynolds stresses and the effects arising because of it [56]. Some applications where Boussinesq hypothesis fails are : flows with sudden changes in mean strain rate, flows over curved surfaces, flows in ducts with secondary motions, and three-dimensional flows [57]. Reynolds stress anisotropy is a key feature of shock-turbulence interaction [58, 86]. A shock wave has strong directionality, i.e. it alters the flow quantities differently between the shock-normal and shock-transverse directions. The anisotropy in the post-shock turbulence is also a function of Mach number, and traditional two-equation models based on linear eddy-viscosity formulation cannot predict it [60].

Strictly speaking, turbulence physics is better represented by Reynolds stress model (RSM), but it requires the solution of additional strongly-coupled equations as against the two less-strongly coupled equations in case of the SST model. Incorporating shock-physics into Reynolds stress models can also lead to additional complexity in terms of the model parameters being function of the local shock orientation [60]. Thus, several efforts

try to develop computationally more feasible approaches to include additional physics in the eddy viscosity models [61–68]. Such attempts mainly focus on using a non-linear relationship between the Reynolds stresses and the mean strain rate and vorticity tensors. However, a limited amount of work is done towards improving supersonic and hypersonic flow predictions, especially those involving SBLIs and SSLIs using these methods [69–71].

A number of non-linear eddy viscosity models, derived along quite different routes, are available in the literature with varying levels of complexities in terms of their formulation and numerical implementation [72]. Such models are developed to improve the predictions of Reynolds stress fields in a given flow and employ either quadratic, cubic, or higher order expansions of the Reynolds stresses in terms of the strain and vorticity tensors. Also, the turbulent eddy viscosity is additionally sensitized to the mean strain and vorticity tensors in these models. However, majority of such models are developed for incompressible flows and a few are explored to predict transonic separated flows [64, 65, 68, 73, 74]. Moreover, a limited amount of work has been directed towards predicting supersonic and hypersonic SBLIs using such models [69, 71].

1.2 Objective

The phenomenon of shock/turbulence interaction (STI) in the form of SBLIs and SSLIs is crucially important in case of shock-dominated flows occurring in the flight of high-speed aerospace vehicles. While the fundamental shock/turbulence interaction mechanisms in such flows is significantly complex, for the engineering predictions, we are usually only interested in a few global manifestations of the interaction : surface pressure, surface heat transfer and skin friction distributions, location and extent of separation, base pressure and heat transfer, and the momentum defect in the wake. Accurate simulation of complex aerospace flows using CFD depends, to a large extent, on the reliable turbulence modelling techniques. Extensive experience in the incompressible environment suggests that atleast two-equation framework turbulence closure models are necessary for accurately predicting separated flows. So, we choose two-equation modelling framework for our model development.

The main objective of this work is to develop an advanced pragmatic engineering turbulence model which

- represents the correct physics of shock/turbulence interaction; specifically the amplification in turbulence kinetic energy across a shock wave, anisotropy in normal

Reynolds stresses in the flowfield, and behaviour of the structure parameter in the shock/turbulence interaction regions.

- is accurate in predicting the engineering quantities of interest; specifically the surface pressure and skin friction distribution, and separation size and location in case of SBLIs, and the mean flow quantities and base pressure in case of SSLIs.
- is valid over a wide range of high-speed practically relevant flows involving SBLIs and SSLIs; this covers flows in supersonic to hypersonic regime and different geometric configurations like compression corner, incident oblique shock over a flat plate boundary layer, backward facing step, flow over a cylindrical blunt body, and multi-nozzle clustered rocket engine.
- is computationally efficient and economical to run; the model development strategy followed here aims to improve the baseline model without paying a large penalty on speed and robustness.
- can be easily implemented in an existing CFD code; minimum possible modelling complexities are used while introducing additional modifications and/or physics to the baseline model.

Specifically, we take two different routes for the model development and the objectives therein are discussed below.

1. In the first part, we consider the SU model of Sinha et al. [30] as the baseline model. Since the CFD implementation of the model is non-local, the objective is to remove the non-local nature making it numerically robust while retaining similar consistency in correctly predicting shock/turbulence interaction as the original model. The model developed in this part is called as the “local SU model”, and it is extensively validated for a range of canonical STI and SBLI flows.
2. In the second part, we take a step further in terms of the modelling complexity as compared to the first part. We consider the standard SST $k-\omega$ as the starting point for our model development. The objective here is to predict anisotropy in normal Reynolds stresses correctly, a key feature in shock/turbulence interaction, which is not possible in case of the LEVM framework. Secondly, the constant value of structure parameter used in the SST model is inconsistent with the experimental observations in the shock/turbulence interaction regions. So, the other objective is to correctly predict the behaviour of structure parameter in such flows.

3. The next objective is to extensively validate the advanced model developed in part 2 against experimental data available for an extremely wide range of high-speed flow configurations. We also attempt, possibly for the first time in existing literature, application of an EARSM-based advanced turbulence model to a realistic four-nozzle clustered rocket engine configuration with multi-jet interaction and reverse flow.

1.3 Scope

The dissertation presents the development and extensive CFD validation of two advanced turbulence models, specifically for shock-dominated flows observed in the high-speed flights of aerospace vehicles. The new models are not developed from scratch, but are improvements over the already existing popular models. For the first model development, the canonical STI problem is used to suggest modelling improvements over the SU model of Sinha et al. [30] while remaining in the LEVM framework. On the other hand, for the second model development, a pragmatic modelling philosophy is followed and an anisotropy-resolving closure, i.e., closure at the level higher than Boussinesq hypothesis used in a LEVM is chosen. Here, the SU model of Sinha et al. [30] is also used to include the physical effect of unsteady oscillations of a shock wave on the turbulence passing through it. The developed model is thus an advanced form of the standard SST $k-\omega$ model with anisotropy resolving features of an EARSM and include additional physics of shock/turbulence interaction due to the SU model. The validation of the new models is carried out using two- and three-dimensional mean flow test cases which cover a wide range of flow conditions, from supersonic to hypersonic regime and attached to fully separated flows, and different geometric configurations, including compression corner, incident oblique shock on a flat plate boundary layer, backward facing step, cylindrical blunt body base flow, single underexpanded jet, and four-nozzle clustered rocket engine configuration with multi-jet interaction.

The physics-based shock-unsteadiness turbulence model of Sinha et al. [30] serves as the starting point for the first advanced turbulence model developed in this work. Using the canonical problem of STI, a physically consistent and numerically robust $k-\epsilon$ model is developed. This is done by eliminating the dependence of model parameters on the non-local upstream flow parameters, as is the case for the shock-unsteadiness model [30]. The turbulence source terms in the proposed model are functions of the local flow variables at a given grid point. The new model equations are also cast in a conservative form to avoid large numerical error at flow discontinuities. As shown in [75], non-conservative source terms can result in unrealistic CFD results at shock waves, and the predictions

are qualitatively and quantitatively different from the analytical solution of the governing equations. Further, the modelling strategy followed allows the retention of the physical characteristics of the problem and maintain the accuracy of the shock-unsteadiness model with respect to DNS data of shock-turbulence interactions.

The local form of the SU model is shown to be consistent with the physics of shock-turbulence interaction. The model accurately predicts turbulence amplification in shock / homogeneous turbulence interaction [61] for a range of Mach numbers and is thus able to match the performance of shock-unsteadiness (SU) model from [30]. The model is then systematically extended to realistic flows in the k - ϵ and k - ω frameworks; similar to the work of Sinha et al. [34]. A detailed comparison of the original and local form of the SU model is presented in terms of their numerical implementation, accuracy of prediction, convergence histories, and computational cost. Different SBLI cases are considered to test the robustness of the new model and to check the generality of the model parameter over a range of shock strengths and flow configuration. The use of different numerical methods and the numerical error at the shock are also addressed. The results for the SBLI cases considered for the model validation show that the local SU model matches the performance of the original SU model in predicting the separation size and location along with surface pressure and skin friction distribution. On the other hand, the newly developed local SU model involves significantly lesser complexities in its implementation resulting in a substantial saving in computational time compared to the original SU model.

Next, an attempt is made to extend the applicability and improve the range of validity of the linear eddy-viscosity based SST model since it displays an inconsistent behaviour in predicting separation and recirculation for different flow conditions and geometric configurations. For the model development, the EARSIM by Rung et al. [73] is considered which has a quadratic relationship between the Reynolds stress and strain-rate and vorticity tensors, thus transcending the Boussinesq hypothesis used in the standard SST model. In doing so, the predictive capabilities of the EARSIM are also explored in case of supersonic and hypersonic SBLIs.

Generally, a quadratic expansion of the Reynolds stresses is found to be sufficient to predict anisotropy in normal Reynolds stresses [72]. Again, a number of choices are available, however, for the purpose of this work, the EARSIM by Rung et al. [73] is considered for the model development because of the model's simplicity, especially for three-dimensional mean flows. This EARSIM is an improved version of the EARSIM by Gatski and Speziale [76] and is developed for compressible flows. It enjoys the desirable properties inherent to the elaborate second-moment closure of Speziale, Sarkar, and Gatski [77]. Thus, it promises to be a less elaborate and application oriented alternative and of-

fers a viable approach towards an all-speed engineering turbulence modelling frame. The model has been implemented in the DLR-TAU and DLR-FLowEr codes, however the model application has been restricted to predicting transonic separated flows in the past [73, 74].

In order to correctly represent the physics of the behaviour of structure parameter in SBLI flows, a new relation for the structure parameter is proposed making it a function of the local mean velocity gradients. Additional shock-physics is introduced to the model by modifying the definition of eddy viscosity at a shock wave using the physics-based shock-unsteadiness (SU) model of Sinha et al. [30]. The SU model brings in the physical effect of unsteady shock oscillations on turbulence passing through a shock wave and gives an upper limit to the values of structure parameter in the shock regions.

The reliability of the new model, called as the SUQ-SST model, in predicting a wide range of high-speed flow configurations involving SBLIs and SSLIs is demonstrated with comparisons to the available experimental data. The results show that the SUQ-SST model is able to correctly predict the Reynolds stress fields in the flows considered for the model validation for which experimental data is available. The model is shown to consistently predict the engineering quantities of interest for the wide range of test cases, including the realistic four-nozzle clustered rocket engine configuration flowfield of multi-jet interaction.

Interestingly, the modelling strategy followed here in the development of SUQ-SST model is not limited to the EARSM by Rung et al. [73] only. It can be readily used with other non-linear models or EARSMs available in the literature. For example, another EARSM developed for compressible flows is by Wallin and Johansson (WJ EARSM) [70]. The model is based on a recalibrated Launder, Reece and Rodi Reynolds stress transport model [78], and consists of a quartic expansion of the Reynolds stresses for a general three-dimensional mean flow. It has been previously applied to hypersonic impinging shock/turbulent boundary layer interaction and results in an underprediction of separation bubble size for moderately to fully separated flows [71]. The implication of the use of other non-linear model/EARSM on the model development process is discussed in Chapter 5.

The scope of application of the the advanced turbulence models developed in this work can go beyond the aerospace-related technological applications considered in this work. One particular example is of Inertial Confinement Fusion (ICF) implosions which involve reshocked Richtmyer-Meshkov induced turbulent mixing process [9–11]. Some of the important mechanisms taking place in ICF implosions are the amplification of turbulence in the mixing layer by a shock-wave and its decay after the shock has passed

through it. In these situations, a turbulence model is expected to correctly predict the evolution of the mixing zone thickness, in particular the effect of the reshocks on the thickening rate. Generally, turbulence models in one-dimensional mean flow set-up are used to simulate ICF implosions [9–11]. As the advanced turbulence models developed here are shown to correctly predict shock/turbulence interaction across a wide range of flow conditions, they can be promising candidates to simulate ICF implosions.

1.4 Organization

The dissertation is organized as follows :

- Chapter 2 provides the governing equations, i.e. RANS, for the compressible turbulent flows used for the CFD simulations in this work. The two-equation k - ϵ , k - ω , SST k - ω models along with two leading EARSMS for compressible flows are discussed. The aerospace-related high-speed flow applications considered in this work for the model validation are discussed in detail. This includes the flowfield details and performance of the k - ϵ and k - ω family of models in such flows.
- Chapter 3 presents the simulation methodology in terms of the numerical schemes and boundary condition details used for the two- and three-dimensional CFD simulations in this work. Computational set-up including the flow conditions, grid details, and boundary conditions employed for the CFD simulations of the test cases are also discussed. The local SU model is validated against six SBLI test cases for two different geometric configurations. On the other hand, the SUQ-SST model is validated using eight SBLI test cases with two different geometric configurations, one case each of supersonic flow over backward facing step and cylindrical blunt body, one case of single underexpanded jet, and two cases of multi-jet interaction.
- Chapter 4 presents the case of canonical shock/turbulence interaction which is used by Sinha et al. [30] to propose the shock-unsteadiness modification to the standard k - ϵ and k - ω models. New forms of the k and ϵ equations are developed that are physically and dimensionally consistent, and numerically more robust than the earlier shock-unsteadiness model. The development of the new model in non-conservative and conservative forms is also discussed. The new model is used to compute a series of canonical shock-turbulence interaction cases and the predictions are compared with the existing DNS data. Extension of the new model to realistic flows is also shown along with validation studies for SBLIs.

- Chapter 5 presents the development of a comparatively more advanced turbulence model, called as SUQ-SST model. The model is an improved version of the LEVM based SST model and uses a quadratic EARSM for the stress-strain constitutive relationship. Modifications to the structure parameter and eddy viscosity formulation in a turbulent boundary layer and in the shock regions in the quadratic eddy viscosity modelling framework are discussed.
- Application of the SUQ-SST model to a wide range of SBLI flows is presented in Chapter 6. Comparisons with available experimental data for Reynolds stresses, structure parameter, surface pressure, and skin friction distribution are also shown.
- Two supersonic base flows, consisting of flow over a backfacing step and cylindrical blunt body, are computed using the standard SST $k-\omega$ and SUQ-SST models and the results in comparison with the experimental data are presented in Chapter 7.
- Chapter 8 presents the application of SUQ-SST model to underexpanded jets. The first test case consists of a supersonic underexpanded jet exhausted into still air. Comparisons between standard SST and SUQ-SST model predictions and experimental measurements for mean flow quantities are shown. The second test case consists of the more realistic flowfield of four-nozzle clustered rocket engine configuration with multi-jet interaction. Two altitude conditions are considered and the SUQ-SST model results are validated for base pressure distributions using the available experimental data.
- The dissertation is concluded and the future work is discussed in Chapter 9.

Chapter 2

Background and Literature Review

2.1 Background

One of the most enthralling things about Nature is that there exists such patterns in its actions, that these patterns can be formulated into some laws using the mathematical language called as the mathematical models. We call these mathematical models the governing equations of that particular physical phenomenon under study. We can write governing equations to predict the change in position or configuration of any solid or fluid in motion or at rest. Navier-Stokes equations are the governing equations in case of fluid flows, used for predicting the fate of any fluid in motion. Solving them, for a particular set of boundary conditions (such as inlets, outlets, and walls), predicts the fluid velocity and its thermodynamic properties like pressure and temperature in a given geometry.

2.1.1 Navier-Stokes equations

The laminar or turbulent motion of a viscous, heat conducting fluid is governed by the Navier-Stokes equations. These are vector equations obtained by applying Newton's 2nd law of motion to a fluid element and they describe the conservation of linear momentum. They are supplemented by the mass conservation equation, also called continuity equation and the energy equation. Usually in the CFD literature, the term Navier-Stokes (NS) equations is used to refer to all of these three equations, i.e., conservation of mass, momentum, and energy. For incompressible flows, the energy equation is not coupled with the mass and momentum equations. Hence, it can be solved independently after getting the velocity and pressure fields from the mass and momentum conservation equations. The NS equations are only valid as long as the fluid is considered to be a continuum i.e. the representative physical length scale of the system is much larger than the mean free path of the molecules that make up the fluid.

2.1.2 Reynolds Averaging

As turbulence consists of highly random fluctuations of the various flow properties, a deterministic approach is not possible and one has to resort to statistical tools. Also, as turbulence consists of large number of scales, resolving each and every scale is not required for engineering applications. So, Reynolds came out with the averaging concepts, i.e. the **time average**, the **spatial average**, and the **ensemble average** in 1895. Generally, most turbulent flows of engineering interest are stationary, i.e. a turbulent flow that on the average does not vary with time. Therefore, time averaging has been considered in this work for arriving at the Reynolds Averaged Navier-Stokes equations (RANS) as it is most appropriate for stationary turbulence. Time average $F_T(x, y, z)$ of an instantaneous flow variable $f(x, y, z, t)$ is given by,

$$F_T(x, y, z) = \lim_{T \rightarrow \infty} \frac{1}{T} \int_t^{t+T} f(x, y, z, t) dt \quad (2.1)$$

Here, x, y, z are the co-ordinates, t is the time instant, and T is the time over which the averaging is done. In reality, infinite T can never truly be realized, however, in practice, a time T that is very long relative to the maximum period of the velocity fluctuations can be chosen to form the time average. Using these averaging concepts, Reynolds introduced a procedure called as the Reynolds decomposition in which all the flow quantities are expressed as the sum of mean and fluctuating parts. For example,

$$u_i(x, y, z, t) = \overline{u_i(x, y, z, t)} + u'_i(x, y, z, t) \quad (2.2)$$

$$= \overline{u_i}(x, y, z, t) + u'_i(x, y, z, t) \quad (2.3)$$

where $\overline{u_i(x, y, z, t)} = \overline{u_i}(x, y, z, t)$ is the time averaged mean velocity and $u'_i(x, y, z, t)$ is the fluctuating component of the velocity. So, using the definition of time averaging, we can write

$$\overline{u_i}(x, y, z) = \lim_{T \rightarrow \infty} \frac{1}{T} \int_t^{t+T} u_i(x, y, z, t) dt \quad (2.4)$$

Some of the important properties of Reynolds time averaging are as follows :

$$\begin{aligned} \overline{\overline{u_i}(x, y, z)} &= \overline{u_i}(x, y, z) \\ \overline{u'_i} &= 0 \\ \overline{\frac{\partial u_i}{\partial x_i}} &= \frac{\partial \overline{u_i}}{\partial x_i} \\ \overline{u_i u_j} &= \overline{u_i} \overline{u_j} + \overline{u'_i u'_j} \end{aligned} \quad (2.5)$$

2.1.3 Consequences of Reynolds averaging procedure

To demonstrate the effect of Reynolds averaging procedure on the Navier-Stokes equations, for simplicity attention would be first confined to incompressible, constant-

property flows. Effects of compressibility would be addressed subsequently. The conservation of mass and momentum equations are given by,

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (2.6)$$

$$\rho \frac{\partial u_i}{\partial t} + \rho \frac{\partial(u_j u_i)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\sigma_{ij}) \quad (2.7)$$

Here, u_i and x_i are velocity and position vectors, t is time, p is pressure, and ρ is density. The viscous stress tensor σ_{ij} is defined as $\sigma_{ij} = 2\mu S_{ij}$, where μ is the molecular viscosity and $S_{ij} = (1/2)(\partial u_i/\partial x_j + \partial u_j/\partial x_i)$. Notice that momentum equations are written in their conservative form so as to simplify the time-averaging process. Reynolds-Averaged Navier-Stokes i.e. RANS equations are derived by taking the time average of the Navier-Stokes equations. Taking time average of mass conservation equation (2.6), we get

$$\overline{\frac{\partial u_i}{\partial x_i}} = 0 \quad (2.8)$$

$$\therefore \frac{\partial \overline{u_i}}{\partial x_i} = 0 \quad (2.9)$$

$$\therefore \frac{\partial (\overline{u_i} + \overline{u'_i})}{\partial x_i} = 0 \quad (2.10)$$

$$\therefore \frac{\partial \overline{u_i}}{\partial x_i} = 0 \quad (2.11)$$

Similarly, by taking the time average of momentum equation (2.7), then decomposing the instantaneous flow variables i.e. velocity and pressure into their time averaged mean and fluctuating parts and using the properties of the Reynolds time averaging given by equation [2.5], we arrive at the momentum equation for the mean velocity given by,

$$\rho \frac{\partial \overline{u_i}}{\partial t} + \rho \overline{u_j} \frac{\partial \overline{u_i}}{\partial x_j} = -\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}(\overline{\sigma_{ij}} - \rho \overline{u'_j u'_i}) \quad (2.12)$$

Equations (2.11) and (2.12) are referred to as the **Reynolds-Averaged Navier-Stokes equations (RANS)**. It is to note that, these equations do not consider fluctuations in density and temperature and are valid for incompressible flows. The quantity $-\rho \overline{u'_j u'_i}$ is known as the **Reynolds-stress tensor** and it is denoted by $\rho \tau_{ij}$. The mean flow and turbulence interact via the Reynolds stresses. Reynolds stress arises as a result of the turbulence and acts on the mean flow, shaping its evolution. It is also responsible for the maintenance of turbulent fluctuations as it channels energy out of the mean flow and into the turbulence. The Reynolds stress is not really a true stress in the conventional sense of word. It represents the flux of momentum caused by the turbulent fluctuations.

From its definition, Reynolds stress is symmetric, i.e. $\tau_{ij} = \tau_{ji}$. Hence, we have produced 6 unknown quantities as a result of Reynolds averaging without gaining any additional equations. So, for general three-dimensional flows, we have 10 unknowns i.e. 4 unknown mean flow properties, viz., pressure and three velocity components along with six Reynolds-stress components. Our equations are mass conservation given by Eqn. (2.11) and the three components of Eqn. (2.12) i.e. total of 4 equations. This means we cannot solve for all the unknowns, i.e. our system is not **closed**. To close the system, we must find enough equations to solve for the unknowns.

2.1.4 The closure problem

In a quest to get additional equations, we can take moments of the NS equations, i.e. we multiply the NS equation by a fluctuating property and time average the product. In doing so, we can derive a differential equation for the Reynolds-stress tensor. Let $N(i)$ denote the Navier-Stokes operator, viz.,

$$N(u_i) = \rho \frac{\partial u_i}{\partial t} + \rho u_k \frac{\partial u_i}{\partial x_k} + \frac{\partial p}{\partial x_i} - \mu \frac{\partial^2 u_i}{\partial x_k \partial x_k} \quad (2.13)$$

The NS equations can be written symbolically as,

$$N(u_i) = 0 \quad (2.14)$$

In order to derive an equation for Reynolds stress tensor, we form the following time average.

$$\overline{u'_i N(u_j) + u'_j N(u_i)} = 0 \quad (2.15)$$

Now, again following the same procedure as that followed for deriving the RANS equations, i.e. splitting the instantaneous quantities into their mean and fluctuating parts and using the properties of the Reynolds time averaging given by Eqn. (2.5), we arrive at the Reynolds stress transport equation given by,

$$\begin{aligned} \frac{\partial \tau_{ij}}{\partial t} + \overline{u_k} \frac{\partial \tau_{ij}}{\partial x_k} = & -\tau_{ik} \frac{\partial \overline{u_j}}{\partial x_k} - \tau_{jk} \frac{\partial \overline{u_i}}{\partial x_k} + 2\nu \frac{\partial u'_i}{\partial x_k} \frac{\partial u'_j}{\partial x_k} + \frac{\overline{u'_i \partial p'}}{\rho \partial x_j} + \frac{\overline{u'_j \partial p'}}{\rho \partial x_i} \\ & + \frac{\partial}{\partial x_k} \left[\nu \frac{\partial \tau_{ij}}{\partial x_k} + \overline{u'_i u'_j u'_k} + \overline{p' u'_i} \right] \end{aligned} \quad (2.16)$$

where $\nu = \mu/\rho$ is the kinematic viscosity. Thus, we have achieved our goal of getting 6 extra equations. However, in doing so we have also produced additional unknowns, which after accounting for all symmetries are 22 in number and are given below.

$$\overline{u'_i u'_j u'_k} \rightarrow 10 \text{ unknowns}$$

$$2\nu \overline{\frac{\partial u'_i}{\partial x_k} \frac{\partial u'_j}{\partial x_k}} \rightarrow 6 \text{ unknowns}$$

$$\overline{\frac{u'_i}{\rho} \frac{\partial p'}{\partial x_j}} + \overline{\frac{u'_j}{\rho} \frac{\partial p'}{\partial x_i}} \rightarrow 6 \text{ unknowns}$$

This illustrates the closure problem of turbulence, i.e. the issue of establishing a sufficient number of equations for all the unknowns. In a nutshell, when NS equations were time-averaged, RANS was obtained. RANS looks similar to NS equations with an additional Reynolds Stress term. So, 6 unknowns were introduced while doing the time averaging without the addition of new equations. As a result, we went for a quest to find extra equations to solve for those 6 extra unknowns. For that we took moment of the NS equations of the form $\overline{[u'_i N(u_j) + u'_j N(u_i)]} = 0$. This led to the Reynolds stress transport equation. Thus, we got 6 new equations to solve for the Reynolds stresses but in doing so, we also introduced 22 new unknowns!

The closure problem occurs because of the non-linearity of the NS equations. As higher and higher moments are taken, additional unknowns are generated at each level. At no point will this procedure balance our unknowns/equations ledger. This is where turbulence modeling comes to our rescue. The function of turbulence modeling is to devise approximations for the unknown correlations in terms of flow properties that are known so that a sufficient number of equations exists. In making such approximations, we close the system. There exists a whole different classes of turbulence models. However, only eddy-viscosity based turbulence models are considered in this work because of their simplicity and effectiveness in solving the engineering problems of interest.

2.1.5 Favre-averaging

In case of flows where compressibility effects are important, we must account for density and temperature fluctuations in the flow field in addition to velocity and pressure fluctuations. Also, we need to introduce conservation of energy and an equation of state along with conservation of mass and momentum. For getting density and temperature fluctuation effects, if we follow conventional Reynolds time averaging, then we end up with equations having such correlations, like triple correlations between ρ' , u'_i , and u'_j , for which suitable closure approximations are very hard to establish. So, in order to reduce the complexity of the process of establishing appropriate form of the time-averaged equations, Favre in 1965 introduced density-weighted averaging also called as **Favre averaging**. The mass-averaged velocity, $\tilde{u}_i$, is defined as

$$\tilde{u}_i = \frac{1}{\bar{\rho}} \lim_{T \rightarrow \infty} \frac{1}{T} \int_t^{t+T} \rho(x, y, z, \tau) u_i(x, y, z, \tau) d\tau \quad (2.17)$$

where $\bar{\rho}$ is the conventional Reynolds-averaged density. For Favre-averaging, various flow properties are decomposed as follows.

$$\begin{aligned}
 u_i &= \tilde{u}_i + u_i'' \\
 \rho &= \bar{\rho} + \rho' \\
 p &= \bar{p} + p' \\
 h &= \tilde{h} + h'' \\
 e &= \tilde{e} + e'' \\
 T &= \tilde{T} + T'' \\
 q_j &= q_{Lj} + q_j'
 \end{aligned} \tag{2.18}$$

where u = velocity, ρ = density, p = pressure, h = specific enthalpy, e = specific internal energy, T = temperature, and q_j = heat flux. Below are some important results for Favre averaging.

$$\begin{aligned}
 \bar{\rho}\tilde{u}_i &= \overline{\rho u_i} = \overline{\rho u_i} + \overline{\rho' u_i'} \\
 \overline{\rho u_i''} &= 0 \\
 \overline{u_i''} &\neq 0
 \end{aligned} \tag{2.19}$$

2.1.6 RANS equations for compressible flows

RANS equations as applicable to compressible flows are derived below.

Conservation of Mass

The conservation of mass equation for compressible flows is given by,

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \tag{2.20}$$

Decomposing the flow variables and performing the Reynolds averaging procedure, we get

$$\frac{\partial(\bar{\rho} + \rho')}{\partial t} + \frac{\partial(\bar{\rho} + \rho')(\tilde{u}_i + u_i'')}{\partial x_i} = 0 \tag{2.21}$$

Making use of the properties of Reynolds averaging and Favre averaging given by Eqns. (2.5) and (2.19), we get

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_i}(\bar{\rho}\tilde{u}_i) = 0 \tag{2.22}$$

Equation (2.22) is the Reynolds-averaged Mass conservation equation.

Conservation of Momentum

The conservation of momentum equation is given by,

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ji}}{\partial x_j} \quad (2.23)$$

Again following the same procedure as that followed for arriving at Reynolds-averaged Mass-conservation equations, i.e. decomposing the flow variables as given by Eqn. (2.18) and then taking the Reynolds average, we get the Reynolds-averaged Momentum equation given by,

$$\frac{\partial}{\partial t}(\bar{\rho} \tilde{u}_i) + \frac{\partial}{\partial x_j}(\bar{\rho} \tilde{u}_j \tilde{u}_i) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} [\bar{\sigma}_{ji} - \overline{\rho u_j'' u_i''}] \quad (2.24)$$

Energy Conservation Equation

The energy conservation equation is given by,

$$\frac{\partial}{\partial t} \left[\rho \left(e + \frac{1}{2} u_i u_i \right) \right] + \frac{\partial}{\partial x_j} \left[\rho u_j \left(h + \frac{1}{2} u_i u_i \right) \right] = \frac{\partial}{\partial x_j} (u_i \sigma_{ij}) - \frac{\partial q_j}{\partial x_j} \quad (2.25)$$

Again following the same procedure as that followed for arriving at Reynolds-averaged Mass and Momentum conservation equations, we get the Reynolds-averaged Energy Equation given by,

$$\begin{aligned} & \frac{\partial}{\partial t} \left[\bar{\rho} \left(\tilde{e} + \frac{1}{2} \tilde{u}_i \tilde{u}_i \right) + \frac{\overline{\rho u_i'' u_i''}}{2} \right] + \frac{\partial}{\partial x_j} \left[\bar{\rho} \tilde{u}_j \left(\tilde{h} + \frac{1}{2} \tilde{u}_i \tilde{u}_i \right) + \tilde{u}_j \frac{\overline{\rho u_i'' u_i''}}{2} \right] \\ &= \frac{\partial}{\partial x_j} \left[-q_{Lj} - \overline{\rho u_j'' h''} + \overline{t_{ji} u_i''} - \overline{\rho u_j'' \frac{1}{2} u_i'' u_i''} \right] + \frac{\partial}{\partial x_j} [\tilde{u}_i (\bar{\sigma}_{ij} - \overline{\rho u_i'' u_j''})] \end{aligned} \quad (2.26)$$

Also, the equation of state given by $p = \rho RT$ on Reynolds-averaging becomes

$$\bar{p} = \bar{\rho} R \tilde{T} \quad (2.27)$$

All terms appearing in these equations are closed except for those resulting from turbulent fluctuations. In particular, the unclosed terms are as follows:

1. the Reynolds stress tensor $(-\overline{\rho u_j'' u_i''})$
2. Turbulent Heat flux $(\overline{\rho u_j'' h''})$
3. Molecular diffusion of turbulence kinetic energy $(\overline{t_{ji} u_i''})$
4. Turbulent transport of turbulence kinetic energy $(\overline{\rho u_j'' \frac{1}{2} u_i'' u_i''})$, and
5. Turbulence kinetic energy $(\frac{1}{2} \overline{\rho u_i'' u_i''})$

Common Closure Approximations

The above unclosed terms in the RANS equations are closed using the following closure approximations.

Reynolds-stress tensor : The Reynolds stress tensor appears in both the Reynolds-averaged momentum (2.24) and energy (2.26) equations. It is closed using Boussinesq hypothesis in case of linear eddy viscosity models (LEVM) or using a non-linear constitutive stress-strain relationship in case of non-linear eddy viscosity models (NLEVMs) or explicit algebraic Reynolds stress models (EARSMS).

Turbulence Heat-flux : The most commonly used closure approximation for the turbulent heat flux vector, q_{Tj} , follows from the analogy between momentum and heat transfer, given by

$$q_{Tj} = \overline{\rho u_j'' h''} = -k_T \frac{\partial \tilde{T}}{\partial x_j} \quad (2.28)$$

where k_T is the eddy thermal conductivity. Now

$$k_T = \frac{\mu_T c_p}{Pr_T} \quad (2.29)$$

where Pr_T is the turbulent Prandtl number, and μ_T is the eddy viscosity. In terms of Reynolds-averaged enthalpy, the closure approximation is

$$q_{Tj} = -\frac{\mu_T}{Pr_T} \frac{\partial \tilde{h}}{\partial x_j} \quad (2.30)$$

Turbulence KE : Generally, for one equation models like Spalart-Allmaras model, TKE is neglected i.e $k = 0$ as we don't have any means to compute it. For the two-equation $k-\epsilon$ and $k-\omega$ family of turbulence models, a separate transport equation for k is solved coupled with turbulent dissipation rate equation.

Molecular Diffusion and Turbulent transport : These terms appear both in Reynolds-Averaged Energy equation and the Reynolds-Averaged TKE (See [57]). The most commonly used approximation used is :

$$\overline{t_{ji} u_i''} - \overline{\rho u_j'' \frac{1}{2} u_i'' u_i''} = \left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \quad (2.31)$$

Using the above discussed closure approximation, the form of the RANS equations that is used in this work is given by,

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_i} (\bar{\rho} \tilde{u}_i) = 0 \quad (2.32)$$

$$\frac{\partial}{\partial t} (\bar{\rho} \tilde{u}_i) + \frac{\partial}{\partial x_j} (\bar{\rho} \tilde{u}_j \tilde{u}_i) + \frac{\partial (-\tau_{ij})}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} \quad (2.33)$$

$$\begin{aligned}
& \frac{\partial \bar{\rho} \tilde{E}}{\partial t} + \frac{\partial}{\partial x_j} (\tilde{\rho} \tilde{H} \tilde{u}_i) + \frac{\partial (-\tau_{ij}) \tilde{u}_i}{\partial x_j} \\
& = \frac{\partial}{\partial x_j} \left[\frac{\bar{\mu}}{Pr} \frac{\partial \tilde{h}}{\partial x_j} + \frac{\mu_T}{Pr_T} \frac{\partial \tilde{h}}{\partial x_j} + \left(\bar{\mu} + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + \frac{\partial (\sigma_{ij} \tilde{u}_i)}{\partial x_j}
\end{aligned} \tag{2.34}$$

The underlined terms in the set of RANS equations describe the average effect of turbulence on the mean flow. Notably, the gradients of Reynolds stresses in the momentum equation (2.33) play a significant role in determining the separation characteristics like separation location and length in case of flows involving shock-induced separations.

The averaged viscous stresses of a Newtonian fluid are described by,

$$\sigma_{ij} = 2\bar{\mu} S_{ij} = \bar{\mu} \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \tilde{u}_k}{\partial x_k} \delta_{ij} \right) \tag{2.35}$$

The averaged molecular viscosity is provided by the Sutherland law,

$$\bar{\mu} = 1.458 \times 10^{-6} \frac{\tilde{T}^{3/2}}{\tilde{T} + 110.3} \tag{2.36}$$

In Eqn. (2.34), we made use of Fourier type law for modeling both laminar and turbulent heat fluxes as per Wilcox [57] with $Pr = 0.72$, $Pr_T = 0.89$ and μ_T is the eddy viscosity. An ideal gas is assumed $\bar{p} = \bar{\rho} R \tilde{T}$ where the specific gas constant for air is $R = 287.1$ J/kg K. For an ideal gas, the averaged specific total energy and total enthalpy are given by,

$$\begin{aligned}
\tilde{E} &= \tilde{e} + \frac{1}{2} \tilde{u}_i \tilde{u}_i + k \\
\tilde{H} &= \tilde{E} + \frac{\bar{p}}{\bar{\rho}} = \tilde{h} + \frac{1}{2} \tilde{u}_i \tilde{u}_i + k
\end{aligned} \tag{2.37}$$

Assuming a perfect gas, the specific heats are constant, so that the averaged specific internal energy and enthalpy are given by,

$$\tilde{e} = c_v \tilde{T}, \quad \tilde{h} = c_p \tilde{T} \tag{2.38}$$

where $\gamma = c_p/c_v = 1.4$ for air.

2.2 Linear eddy viscosity turbulence models

For engineering predictions of aerospace-related high-speed flows, RANS continue to constitute the principal approach for representing the effects of turbulence on the mean flow. However, in order to compute all mean flow properties, we need a prescription for computing the unknown correlation, i.e. the Reynolds stress $(-\overline{\rho u''_j u''_i})$ and turbulence models provide that description. For a turbulence model to be good and useful, it

- must have wide applicability

- must be accurate
- must be simple and
- must be economical to run

Following are the classes of turbulence models in the RANS framework :

- Eddy viscosity models
 - Linear eddy viscosity models
 - Non-linear eddy viscosity models and explicit algebraic Reynolds stress models
- Reynolds stress transport models

Eddy viscosity models relate the Reynolds stresses to the mean strain-rate via the quantity called as eddy viscosity μ_T . Thus, the problem of finding Reynolds stresses gets converted to finding μ_T . Depending on the number of equations to be solved to determine μ_T , eddy viscosity models can be grouped as zero-, one-, or two-equation models. A turbulence modelling class beyond the eddy viscosity framework is that of Reynolds stress transport models and consists of transport equations for all the Reynolds stresses without recourse to the eddy viscosity concept. In case of three-dimensional mean flows, they require the solution of six highly coupled, non-linear partial differential equations along with an addition of at least one equation for the rate of turbulence dissipation. Thus, Reynolds stress transport models are mathematically more complex, pose greater numerical challenges, and require higher computational costs compared to the eddy viscosity models.

In this work, the focus is on the two-equation eddy viscosity models primarily because of the simplicity and economy of these models compared to the Reynolds stress transport models. Also, unlike the zero- and one-equation models, two-equation models do not require the specification of empirical length-scales or time-scales that must be adjusted in an ad-hoc fashion from one flow to the other. Two-equation eddy viscosity models, as the name suggests, employ the concept of eddy viscosity which was first introduced by Boussinesq in 1877 in an attempt to develop a mathematical description of turbulent stresses. Boussinesq linked the Reynolds stresses with the eddy viscosity, called as the Boussinesq eddy viscosity hypothesis, by mimicking the molecular gradient-diffusion process. Eddy viscosity, unlike the molecular viscosity, is not the property of the fluid but it depends upon the flow.

2.2.1 Boussinesq eddy viscosity hypothesis

It has been a major goal of fluid dynamics research for well over a century to develop such a turbulence closure model that is reliably accurate over the range of conditions encountered in essentially all practical problems. The most widely used turbulence models are formulated in a framework that introduces the required closure directly at the level of the second-order single-point moments ($-\overline{\rho u_j'' u_i''}$). These are based on the classical equilibrium Boussinesq hypothesis that assumes the Reynolds stresses to be directly proportional to the mean strain rate. Boussinesq eddy viscosity hypothesis lies at the heart of the eddy viscosity turbulence models. Boussinesq postulated that momentum transfer caused by turbulent eddies can be modeled with an eddy viscosity μ_T . This is in analogy with how the momentum transfer caused by the molecular motion in a gas can be described by a molecular viscosity. The Boussinesq assumption states that the Reynolds stress tensor τ_{ij} is proportional to the trace-less mean strain rate tensor S_{ij} and can be written in the following way:

$$\tau_{ij} = -\overline{\rho u_j'' u_i''} = 2\mu_T S_{ij} - \frac{2}{3}k\delta_{ij} \quad (2.39)$$

where $S_{ij} = (1/2)(\partial \tilde{u}_i / \partial x_j + \partial \tilde{u}_j / \partial x_i) - (1/3)(\partial \tilde{u}_k / \partial x_k)\delta_{ij}$ and k is the turbulence kinetic energy (TKE). The second term is included so as to balance the normal Reynolds stresses. Eddy viscosity models based on the Boussinesq hypothesis are called as Linear eddy viscosity models (LEVM) as Reynolds stresses are linearly related to the strain-rate tensor in Eqn. (2.39).

On dimensional grounds, eddy viscosity is related to the length-scale and time-scale of turbulence and the k - ϵ and k - ω family of LEVM provide an estimate of these scales using turbulence kinetic energy k , and turbulence dissipation ϵ or specific turbulence dissipation rate ω . It is important to note at this point some of the important shortcomings of the LEVM which are especially pertinent to and influential in the flows considered in this work :

- they do not resolve anisotropy in normal Reynolds stresses, a key feature in shock/turbulence interaction;
- they significantly over-estimate Reynolds stresses at high strain rates, e.g. in the vicinity of shock waves;
- they gravely mispredict the heat-transfer rates when used in conjunction with eddy-diffusivity /gradient-diffusion approximations with a constant turbulent Prandtl number.

So, a number of efforts are directed to improve the performance of these models in complex flows, especially involving separation and recirculation. LEVMs still remain arguably the most popular group of models in aerospace-related CFD. This is because the extremely complex geometries and the associated meshing problems involved in such applications favor the simplicity and economy of these models for engineering predictions.

2.2.2 Turbulence kinetic energy equation

Turbulence kinetic energy (TKE) is the mean kinetic energy per unit mass associated with eddies in a turbulent flow. The value of TKE directly represents the strength of the turbulence in the flow. TKE can be produced by fluid shear, friction or buoyancy. It is then transferred down the turbulence energy cascade, and is dissipated by viscous forces. So, TKE is a very important quantity in turbulence. The two-equation k - ω and k - ϵ family of turbulence models are based upon TKE.

We determine TKE, k , by taking the trace of specific Reynolds-stress tensor as,

$$-\overline{u_i'' u_i''} = -2k \quad (2.40)$$

Thus, the trace of Reynolds-stress tensor is proportional to the TKE. So, TKE equation is derived by taking the trace of the Reynolds stress transport equation; see Wilcox [57] for details. The TKE transport equation for compressible flows is given by,

$$\frac{\partial \bar{\rho} k}{\partial t} + \bar{\rho} \tilde{u}_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} - \bar{\rho} \epsilon + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad (2.41)$$

where σ_k is the closure coefficient. Note that, we can relate ϵ and ω in the following manner,

$$\epsilon = C_\mu k \omega \quad \text{with} \quad C_\mu = 0.09 \quad (2.42)$$

Meaning of various terms of TKE equation

$$LHS = \frac{\partial \bar{\rho} k}{\partial t} + \bar{\rho} \tilde{u}_j \frac{\partial k}{\partial x_j} \quad (2.43)$$

The first term represents the local change in k and the second term represents the convection of k . Together they represent the rate of change of k if we follow a turbulent eddy.

$$\tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} \Rightarrow \text{Production term}$$

The production term represents the rate at which TKE is transferred from the mean flow to the turbulence.

$$\epsilon = \nu \overline{\frac{\partial u'_i}{\partial x_k} \frac{\partial u'_i}{\partial x_k}} \Rightarrow \text{Dissipation}$$

Dissipation represents the rate at which TKE is converted into thermal internal energy.

$$\left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \Rightarrow \text{Molecular + Turbulent Diffusion}$$

The above term represents the diffusion of TKE due to molecular and turbulent transport processes.

2.2.3 Standard k - ϵ model

The standard k - ϵ model of Launder and Sharma [79] is given by,
 k -equation

$$\frac{\partial \bar{\rho} k}{\partial t} + \bar{\rho} \tilde{u}_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] - \bar{\rho} \epsilon - 2\mu \left(\frac{\partial \sqrt{k}}{\partial x_j} \right)^2 \quad (2.44)$$

ϵ -equation

$$\begin{aligned} \frac{\partial \bar{\rho} \epsilon}{\partial t} + \bar{\rho} \tilde{u}_j \frac{\partial \epsilon}{\partial x_j} = & c_{\epsilon 1} \frac{\epsilon}{k} \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] - c_{\epsilon 2} f_2 \bar{\rho} \frac{\epsilon^2}{k} \\ & + 2 \frac{\mu \mu_T}{\rho} \left(\frac{\partial^2 \tilde{u}_i}{\partial x_j \partial x_l} \right)^2 \end{aligned} \quad (2.45)$$

Stress-strain relation :

$$\tau_{ij} = 2\mu_T S_{ij} - \frac{2}{3} \rho k \delta_{ij} \quad (2.46)$$

Eddy viscosity :

$$\mu_T = c_\mu f_\mu \bar{\rho} \frac{k^2}{\epsilon} \quad (2.47)$$

Model constants and parameters :

$$c_{\epsilon 1} = 1.44 \quad c_{\epsilon 2} = 1.92 \quad c_\mu = 0.09 \quad (2.48)$$

$$f_\mu = \exp \left[\frac{-3.4}{(1 + 0.02 R_t)^2} \right] \quad f_2 = 1 - 0.3 e^{-R_t^2} \quad R_t = \frac{\bar{\rho} k^2}{\mu \epsilon} \quad \sigma_k = 1 \quad \sigma_\epsilon = 1.3 \quad (2.49)$$

The different mechanisms which can be identified in the k and ϵ equations are,

Rate of change of k or ϵ + Transport of k or ϵ by convection
 = Rate of production of k or ϵ + Transport of k or ϵ by diffusion - Rate of destruction
 of k or ϵ + low Reynolds number correction for k or ϵ

2.2.4 Standard k - ω model

The standard k - ω model of Wilcox [80] is given by,

k -equation

$$\bar{\rho} \frac{\partial k}{\partial t} + \bar{\rho} \tilde{u}_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} - \beta^* \bar{\rho} k \omega + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad (2.50)$$

ω -equation

$$\bar{\rho} \frac{\partial \omega}{\partial t} + \bar{\rho} \tilde{u}_j \frac{\partial \omega}{\partial x_j} = \alpha \frac{\omega}{k} \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} - \beta \bar{\rho} \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma} \right) \frac{\partial \omega}{\partial x_j} \right] \quad (2.51)$$

Stress-strain relation :

$$\tau_{ij} = 2\mu_T S_{ij} - \frac{2}{3}\rho k \delta_{ij} \quad (2.52)$$

Eddy viscosity :

$$\mu_T = \frac{\bar{\rho} k}{\omega} \quad (2.53)$$

Closure Coefficients

$$\alpha = \frac{5}{9}, \quad \beta = \frac{3}{40}, \quad \beta^* = \frac{9}{100}, \quad \sigma = 2, \quad \sigma_k = 1 \quad (2.54)$$

Similar to the standard k - ϵ model, the different mechanisms that can be identified in the k and ω transport equations are,

Rate of change of k or ω + Transport of k or ω by convection
 = Rate of production of k or ω - Rate of destruction of k or ω + Transport of k or ω
 by diffusion

2.2.5 Shock-unsteadiness k - ϵ and k - ω models

Sinha et al. [34] propose modifications to the standard k - ϵ and k - ω models for high-speed shock-dominated flows in order to remedy the problem of excessive production obtained using the standard models at shock waves. The TKE production term in the k -equation is recast into a form such that it switches between the shock-unsteadiness (SU) modified form, active in the shock regions, and the standard form, active everywhere else. The switching is done using an algebraic shock-detecting function f_s with $f_s = 1$ in the shock regions and everywhere else it is zero. Interestingly, the form of f_s needs to be changed from one flow configuration to the other; see Sinha et al. [34], Pasha and Sinha [35]. The modified form of the production term used in the k -equation of the standard k - ϵ and k - ω models is given by,

$$P'_k = c'_\mu \mu_T \left(2S'_{ij} S'_{ji} - \frac{2}{3} S_{ii}^2 \right) - \frac{2}{3} \rho k S_{ii} \quad (2.55)$$

where $S'_{ij} = (1/2)(\partial \tilde{u}_i / \partial x_j + \partial \tilde{u}_j / \partial x_i)$ and S_{ii} is the dilatation. Here, P'_k is the modified production term to be used in place of the standard k -equation's production term $P_k = \tau_{ij} \partial \tilde{u}_i / \partial x_j$ and the form in Eqn. (2.55) is applicable to both the k - ϵ and k - ω models. For the k - ϵ model, the SU model parameters in Eqn. (2.55) are given by,

$$c'_\mu = 1 - f_s \left(1 + (b'_1/3) / \zeta C_\mu f_\mu \right) \quad \zeta = \frac{S k}{\tilde{\epsilon}} \quad (2.56)$$

where, $\tilde{\epsilon} = \epsilon + 2\nu(\partial \sqrt{k} / \partial x_j)^2$ is the total dissipation rate obtained by adding the low-Reynolds-number term in the k -equation to the turbulent dissipation rate and f_μ is as per Eqn. (2.49). On the other hand, for the model in the k - ω framework, the model parameters in Eqn. (2.55) are given by,

$$c'_\mu = 1 - f_s (1 + (b'_1/3) / \zeta) \quad \zeta = \frac{S}{\omega} \quad (2.57)$$

In Eqns. (2.56) and (2.57), $S = \left(2S'_{ij}S'_{ji} - \frac{2}{3}S_{ii}^2 \right)^{1/2}$ and $b'_1 = \max[0, 0.4(1 - e^{1-M_{1n}})]$ which represents the damping effect caused by the coupling between the shock unsteadiness and the upstream velocity fluctuations. Here, M_{1n} is the upstream shock-normal Mach number and it is measured in a frame of reference that is stationary with respect to the shock wave.

2.2.6 Menter's analysis : SST correction to eddy viscosity

It is well known that standard k - ϵ and k - ω turbulence models fail to accurately predict flows with strong pressure gradients and separation. Menter [37] attributed the failure with regard to such adverse gradient flows to the definition of eddy viscosity μ_T . Menter's analysis of the general two-equation turbulence models showed that the classical formulation of eddy viscosity in these models violates Bradshaw's assumption, i.e., principal Reynolds shear stress $\tau_{xy} = \bar{\rho} a_1 k$ with $a_1 = 0.31$, in an adverse pressure gradient boundary layer. The analysis is based on a parallel shear flow and notes that the two-equation turbulence models like the standard k - ω model compute Reynolds shear stress as $\tau_{xy} = \mu_T (\partial \tilde{u} / \partial y)$. The TKE production and dissipation in case of the k - ω model can be written as,

$$\begin{aligned} \text{Production}_k &= \tau_{xy} \frac{\partial u}{\partial y} = \mu_T \Omega^2 \\ \text{Dissipation}_k &= \beta^* \bar{\rho} k \omega \end{aligned} \quad (2.58)$$

where the mean velocity gradient is generalized using the vorticity magnitude $\Omega = \sqrt{2\Omega_{ij}\Omega_{ij}}$ and $\Omega_{ij} = (1/2)(\partial \tilde{u}_i / \partial x_j - \partial \tilde{u}_j / \partial x_i)$. The ratio between TKE production and

dissipation is then given by,

$$\begin{aligned}
 \frac{Production_k}{Dissipation_k} &= \frac{\mu_T \Omega^2}{\beta^* \bar{\rho} k \omega} = \frac{\bar{\rho} k}{\omega} \frac{\Omega^2}{\beta^* \bar{\rho} k \omega} = \frac{\Omega^2}{\beta^* \omega^2} = \frac{\Omega^2}{a_1^2 \omega^2}, \text{ where } \beta^* = a_1^2 \\
 \therefore \sqrt{\frac{Production_k}{Dissipation_k}} &= \frac{\Omega}{a_1 \omega} \\
 \therefore \Omega &= \sqrt{\frac{Production_k}{Dissipation_k}} a_1 \omega \\
 \therefore \frac{\tau_{xy}}{\mu_T} &= \sqrt{\frac{Production_k}{Dissipation_k}} a_1 \omega \\
 \therefore \tau_{xy} &= \mu_T \sqrt{\frac{Production_k}{Dissipation_k}} a_1 \omega = \frac{\rho k}{\omega} \sqrt{\frac{Production_k}{Dissipation_k}} a_1 \omega \\
 \therefore \tau_{xy} &= \sqrt{\frac{Production_k}{Dissipation_k}} \rho a_1 k
 \end{aligned} \tag{2.59}$$

Now, for an equilibrium turbulent boundary layer, the ratio of production to dissipation of TKE is close to one, and therefore $\tau_{xy} = \bar{\rho} a_1 k$ and the two-equation models satisfy the Bradshaw's assumption. However, in adverse pressure gradient flows, the ratio of production to dissipation can be significantly larger than one. Hence, above equation leads to an overprediction of τ_{xy} as compared to that given by the Bradshaw's assumption.

In order to satisfy Bradshaw's assumption, the definition of eddy viscosity can be changed as,

$$\mu_T = \frac{\bar{\rho} a_1 k}{\Omega} \tag{2.60}$$

which translates to

$$\tau_{xy} = \mu_T \sqrt{\frac{Production_k}{Dissipation_k}} a_1 \omega = \frac{\bar{\rho} a_1 k}{\Omega} \sqrt{\frac{Production_k}{Dissipation_k}} a_1 \omega \tag{2.61}$$

$$\therefore \tau_{xy} = \frac{\bar{\rho} a_1 k}{\Omega} \frac{\Omega}{a_1 \omega} a_1 \omega = \bar{\rho} a_1 k \tag{2.62}$$

Now, $\mu_T = \bar{\rho} a_1 k / \Omega$ is not desirable for the complete flowfield, since it leads to infinitely high μ_T at points where Ω goes to zero. In order to avoid the singular behaviour for the $\Omega = 0$ case, eddy viscosity can be formulated as,

$$\mu_T = \frac{\bar{\rho} a_1 k}{\max(a_1 \omega, \Omega F_2)} \tag{2.63}$$

This ensures that $\tau_{xy} = \bar{\rho} a_1 k$ is satisfied for the adverse pressure gradient flows. The principal Reynolds shear stress τ_{xy} therefore satisfies the same transport equation as the

TKE. Hence, the model is termed as “shear stress transport” (SST) model since the definition of μ_T accounts for the transport of principal turbulent shear stress. Equation (2.63) guarantees therefore the selection of $\mu_T = \bar{\rho}a_1k/\Omega$ for most of the adverse pressure gradient regions including the wake region of boundary layer. The blending function F_2 is introduced here so as make sure that the SST modified μ_T remains active only inside the boundary layers; $F_2 = 1$ in a boundary layer and outside it is zero. The SST modified μ_T can be cast in several forms as,

$$\mu_T = \frac{\bar{\rho}a_1k}{a_1\omega \left[\max\left(1, \frac{\Omega F_2}{a_1\omega}\right) \right]} = \frac{\bar{\rho}a_1k}{\max(a_1\omega, \Omega F_2)} = \min\left[\frac{\rho k}{\omega}, \frac{\rho a_1 k}{F_2 \Omega}\right] \quad (2.64)$$

2.2.7 Standard SST k - ω model

The standard SST k - ω model is a hybrid model based on the k - ω framework, which switches between high-Re k - ϵ formulation in free-shear layers and standard k - ω model in the boundary layer regions near a wall. The switching is done using a function F_1 and the model uses a modified definition of eddy viscosity as discussed in the earlier section. The model is given by [37],

k -equation

$$\bar{\rho} \frac{\partial k}{\partial t} + \bar{\rho} \tilde{u}_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} - \beta^* \bar{\rho} k \omega + \frac{\partial}{\partial x_j} \left[(\mu + \mu_T \sigma_k) \frac{\partial k}{\partial x_j} \right] \quad (2.65)$$

ω -equation

$$\begin{aligned} \bar{\rho} \frac{\partial \omega}{\partial t} + \bar{\rho} \tilde{u}_j \frac{\partial \omega}{\partial x_j} &= \frac{\gamma}{\nu_T} \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} - \beta \bar{\rho} \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \mu_T \sigma_\omega) \frac{\partial \omega}{\partial x_j} \right] \\ &+ 2(1 - F_1) \frac{\bar{\rho} \sigma_{\omega_2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \end{aligned} \quad (2.66)$$

Stress-strain relation :

$$\tau_{ij} = 2\mu_T S_{ij} - \frac{2}{3}\rho k \delta_{ij} \quad (2.67)$$

Eddy viscosity :

$$\mu_T = \frac{\bar{\rho}a_1k}{\max(a_1\omega, \Omega F_2)} \quad (2.68)$$

The function F_2 switches eddy viscosity smoothly between the SST modified form active inside boundary layers and the standard form everywhere else. Here, Ω is the vorticity magnitude, while some variants of the SST (given in Appendix D) use the strain-rate magnitude S instead. Here, $\Omega = \sqrt{-2II_\Omega}$ and $S = \sqrt{2II_S}$, with the second invariants of vorticity and strain-rate tensors given by $II_\Omega = \Omega_{ij}\Omega_{ji}$ and $II_S = S_{ij}S_{ji}$, respectively. The vorticity and strain-rate tensors are $\Omega_{ij} = (1/2)(\partial \tilde{u}_i/\partial x_j - \partial \tilde{u}_j/\partial x_i)$ and $S_{ij} = (1/2)(\partial \tilde{u}_i/\partial x_j + \partial \tilde{u}_j/\partial x_i) - (1/3)(\partial \tilde{u}_k/\partial x_k)\delta_{ij}$, respectively.

Model constants and parameters :

$$\begin{aligned}
F_1 &= \tanh(\arg_1^4) \\
\arg_1 &= \min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega d}, \frac{500\nu}{d^2 \omega} \right); \frac{4\rho\sigma_{\omega_2} k}{C_{D_{k\omega}} d^2} \right] \\
C_{D_{k\omega}} &= \max \left(2\rho\sigma_{\omega_2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-20} \right) \\
F_2 &= \tanh(\arg_2^2) \\
\arg_2 &= \max \left(2 \frac{\sqrt{k}}{\beta^* \omega d}, \frac{500\nu}{d^2 \omega} \right)
\end{aligned} \tag{2.69}$$

and d is the shortest distance from the wall. Each of the constants in Eqs. (2.65) and (2.66) is a blend of an inner (1) and outer (2) constants, blended via:

$$\phi = F_1 \phi_1 + (1 - F_1) \phi_2 \tag{2.70}$$

where ϕ_1 represents constant 1 and ϕ_2 represents constant 2. The constants are :

$$\begin{aligned}
\gamma_1 &= \frac{\beta_1}{\beta^*} - \frac{\sigma_{\omega_1} \kappa^2}{\sqrt{\beta^*}}, \gamma_2 = \frac{\beta_2}{\beta^*} - \frac{\sigma_{\omega_2} \kappa^2}{\sqrt{\beta^*}} \\
\sigma_{k_1} &= 0.85, \sigma_{\omega_1} = 0.5, \beta_1 = 0.075 \\
\sigma_{k_2} &= 1.0, \sigma_{\omega_2} = 0.856, \beta_2 = 0.0828 \\
\beta^* &= 0.09, \kappa = 0.41, a_1 = 0.31
\end{aligned} \tag{2.71}$$

2.3 Explicit algebraic Reynolds stress models

The inherent limitations of the LEVM, especially their inability to predict anisotropy in Reynolds stresses, along with the modelling complexities, CFD implementation difficulties, and higher computational costs associated with Reynolds stress transport models have motivated efforts to construct a different class of turbulence models. Such class of turbulence models still use the two-equation differential forms of k - ϵ or k - ω transport equations discussed earlier. However, they differ from the LEVM in that the representation of the Reynolds stress tensor contain higher-order terms and the expansion coefficients are dependent on turbulent-to-mean flow time scale ratios. Nonlinear eddy-viscosity models (NLEVM) and explicit algebraic Reynolds stress models (EARSMS) fall under the category of turbulence models mentioned above.

The modelling strategy adopted in constructing NLEVMs and EARSMS is discussed below. First, Reynolds stress is written in terms of its isotropic form and the deviations from isotropy as,

$$\tau_{ij} = -\overline{\rho u_i'' u_j''} = - \left(-(\overline{\rho u_i'' u_j''})_{dev} + \frac{2}{3} \bar{\rho} k \delta_{ij} \right) \tag{2.72}$$

where the subscript “*dev*” represents the deviatoric part. The anisotropy part $(\overline{\rho u_i'' u_j''})_{dev}$ can be equivalently expressed in terms of the Reynolds stress anisotropy tensor a_{ij} as,

$$a_{ij} = -\frac{(\overline{\rho u_i'' u_j''})_{dev}}{\bar{\rho}k} \quad (2.73)$$

and hence the Reynolds stress tensor can be written as,

$$\tau_{ij} = -\bar{\rho}k \left(a_{ij} + \frac{2}{3} \delta_{ij} \right) \quad (2.74)$$

Thus, the closure for Reynolds stress tensor amounts to constructing a representation for the anisotropy tensor a_{ij} . Using theories from Linear algebra, the most general form for a_{ij} in terms of S_{ij} and Ω_{ij} can be written in terms of ten tensorially independent groups to which all higher-order tensor combinations can be reduced with the aid of the Cayley-Hamilton theorem. It is given by [70],

$$\begin{aligned} a_{ij} = & \beta_1 S_{ij} + \beta_2 (S_{ik} S_{kj} - II_S \delta_{ij}/3) + \beta_3 (\Omega_{ik} \Omega_{kj} - II_\Omega \delta_{ij}/3) + \beta_4 (S_{ik} \Omega_{kj} - \Omega_{ik} S_{kj}) \\ & + \beta_5 (S_{ik} S_{kl} \Omega_{lj} - \Omega_{ik} S_{kl} S_{lj}) + \beta_6 (S_{ik} \Omega_{kl} \Omega_{lj} + \Omega_{ik} \Omega_{kl} S_{lj} - IV \delta_{ij}/3) \\ & + \beta_7 (S_{ik} S_{kl} \Omega_{lp} \Omega_{pj} + \Omega_{ik} \Omega_{kl} S_{lp} S_{pj} - 2V \delta_{ij}/3) \\ & + \beta_8 (S_{ik} \Omega_{kl} S_{lp} S_{pj} - S_{ik} S_{kl} \Omega_{lp} S_{pj}) + \beta_9 (\Omega_{ik} S_{kl} \Omega_{lp} \Omega_{pj} - \Omega_{ik} \Omega_{kl} S_{lp} \Omega_{pj}) \\ & + \beta_{10} (\Omega_{ik} S_{kl} S_{lp} \Omega_{pq} \Omega_{qj} - \Omega_{ik} \Omega_{kl} S_{lp} S_{pq} \Omega_{qj}) \end{aligned} \quad (2.75)$$

The number of independent scalar invariants is limited to five in the case of three-dimensional mean flow and are given by,

$$II_S = S_{kl} S_{lk}, \quad (2.76)$$

$$II_\Omega = \Omega_{kl} \Omega_{lk}, \quad (2.77)$$

$$III_S = S_{kl} S_{lm} S_{mk}, \quad (2.78)$$

$$IV = S_{kl} \Omega_{lm} \Omega_{mk}, \quad (2.79)$$

$$V = S_{kl} S_{lm} \Omega_{mn} \Omega_{nk}. \quad (2.80)$$

EARSMs and NLEVMs both use an explicit tensor representation for Reynolds stress tensor which consist of polynomial expansion of the anisotropy tensor that is a subset of Eqn. (2.75). However, despite the formal functional equivalence, fundamental differences exist in their origin, precisely in the way the expansion coefficients, i.e. β_i 's (with $i = 2, 3, \dots, 10$) are obtained. In case of NLEVMs, the expansion coefficients are determined based on calibrations with experimental or numerical data, and on some physical consistency constraints. On the other hand, in case of EARSMs, the expansion coefficients

are derived from the full differential Reynolds stress equations under weak equilibrium assumption.

EARSMs, particularly, can be considered as the approximations of Reynolds stress transport models. They are numerically simpler and more economical than Reynolds stress transport closures and provide higher accuracy compared to LEVM and are therefore gaining popularity. The modelling strategy followed in deriving an EARSM involves a mathematically sound transfer of desirable model properties inherent to the elaborate Reynolds stress transport models to a less elaborate modelling framework. Thus, EARSMs can be considered as an intermediate-level between Reynolds stress transport models and LEVMs. Specifically, these models neglect the advection and diffusion terms in the exact transport equation for a_{ij} which yields an algebraic expression that can be written as,

$$-\frac{\tau_{ij}}{\bar{\rho}k}(P - \epsilon) = P_{ij} - \epsilon_{ij} + \Pi_{ij} \quad (2.81)$$

Here, P and ϵ represents the TKE production and dissipation, respectively. On the other hand, P_{ij} represents the production of Reynolds stresses, Π_{ij} and ϵ_{ij} , respectively, the pressure-strain correlation representing the mechanism of turbulence redistribution and relaxation, and the dissipation rate tensor. Depending on the choice of the models for ϵ_{ij} and Π_{ij} , the expansion coefficients in Eqn. (2.75) can be obtained.

In case of EARSMs, the general relationship between Reynolds stress and mean strain-rate and vorticity tensors can be written in terms of an effective turbulent viscosity and an extra anisotropy term as [70],

$$\tau_{ij} = 2\mu_T S_{ij} - \frac{2}{3}\bar{\rho}k\delta_{ij} - \bar{\rho}ka_{ij}^{(ex)} \quad (2.82)$$

where superscript “*ex*” refers to the extra part added to the definition of τ_{ij} in comparison to that given by the Boussinesq hypothesis. The form of the effective eddy viscosity μ_T and the extra anisotropy tensor $a_{ij}^{(ex)}$ differs from one EARSM to other and depends on the parent Reynolds stress model from which it is derived. Thus, the performance across different EARSMs can vary considerably for identical flow conditions. First attempt for the derivation of EARSM was done by Pope for two-dimensional mean flows and it was later extended and refined by Gatski and Speziale [66] and by Jongen and Gatski [81] for three-dimensional flows. Two of the leading EARSMs derived for compressible flows are given below.

2.3.1 EARSM by Wallin and Johansson [70]

Wallin and Johansson’s [70] explicit algebraic Reynolds stress model (WJ-EARSM) is based on a recalibrated Launder, Reece and Rodi Reynolds stress transport model

(RSTM), from which a fully explicit and self-consistent algebraic relation is derived. The approach replaces the eddy-viscosity assumption by a more general constitutive relation for the Reynolds stress anisotropy in terms of the strain- and rotation rate tensors, as given by Eqn. (2.82). The effective turbulent viscosity and the extra anisotropy for a general three-dimensional mean flow are given by,

$$\mu_T = -\frac{1}{2}(\beta_1 + II_\Omega \beta_6) \bar{\rho} k \tau \quad (2.83)$$

$$\begin{aligned} a_{ij}^{(ex)} = & \beta_3 \tau^2 (\Omega_{ik} \Omega_{kj} - II_\Omega \delta_{ij}/3) + \beta_4 \tau^2 (S_{ik} \Omega_{kj} - \Omega_{ik} S_{kj}) \\ & + \beta_6 \tau^3 (S_{ik} \Omega_{kl} \Omega_{lj} + \Omega_{ik} \Omega_{kl} S_{lj} - II_\Omega S_{ij} - (2/3)IV \delta_{ij}) \\ & + \beta_9 \tau^4 (\Omega_{ik} S_{kl} \Omega_{lp} \Omega_{pj} - \Omega_{ik} \Omega_{kl} S_{lp} \Omega_{pj}) \end{aligned} \quad (2.84)$$

Here, $\tau = \max(k/\epsilon, 6\sqrt{\mu/\bar{\rho}\epsilon})$ is the turbulent time scale incorporating the Durbin time-scale realizability limitation [70].

$$\begin{aligned} \beta_1 = -\frac{N(N^2 - 7II_\Omega)}{Q}, \quad \beta_3 = -\frac{12N^{-1}IV}{Q}, \quad \beta_4 = -\frac{2(N^2 - 2II_\Omega)}{Q} \\ \beta_6 = -\frac{6N}{Q}, \quad \beta_9 = \frac{6}{Q}, \quad Q = \frac{5}{6}(N^2 - 2II_\Omega)(2N^2 - II_\Omega) \end{aligned} \quad (2.85)$$

and N is obtained from the solution of a cubic polynomial equation (see [70] for details),

$$N = \begin{cases} \frac{c'_1}{3} + (P_1 + \sqrt{P_2})^{1/3} + \text{sgn}(P_1 - \sqrt{P_2})|P_1 - \sqrt{P_2}|^{1/3}, & \text{for } P_2 \geq 0 \\ \frac{c'_1}{3} + 2(P_1^2 - P_2)^{1/6} \cos\left(\frac{1}{3} \arccos\left(\frac{P_1}{\sqrt{P_1^2 - P_2}}\right)\right), & \text{for } P_2 < 0 \end{cases} \quad (2.86)$$

with

$$P_1 = c'_1 \left(\frac{c_1^2}{27} + \frac{9}{20}II_S - \frac{2}{3}II_\Omega \right), \quad P_2 = P_1^2 - \left(\frac{c_1^2}{9} + \frac{9}{10}II_S - \frac{2}{3}II_\Omega \right)^3. \quad (2.87)$$

and

$$c'_1 = \frac{9}{4} \left(c_1 - 1 - \frac{2}{3} \tau \frac{\partial \tilde{u}_k}{\partial x_k} \right), \quad c_1 = 1.8 \quad (2.88)$$

Here, $II_S = S_{ij}S_{ji}$ and $II_\Omega = \Omega_{ij}\Omega_{ji}$, $IV = S_{kl}\Omega_{lm}\Omega_{mk}$ are the tensor invariants of the strain and vorticity tensors.

2.3.2 EARSM by Rung et al. [73]

EARSM of Rung et al. [73] is a realizable quadratic eddy viscosity model and is based on the original EARSM by Gatski and Speziale [76]. It also makes use of the general constitutive relation for Reynolds stress tensor as given by Eqn. (2.82). The extra anisotropy tensor for this model is given by a quadratic form [73],

$$a_{ij}^{(ex)} = -2C_\mu^* \tau^2 \left[\beta'_2 (S_{ik} \Omega_{kj} - \Omega_{ik} S_{kj}) - \beta'_3 \left(S_{ik} S_{kj} - \frac{1}{3} S_{kl} S_{kl} \delta_{ij} \right) \right] \quad (2.89)$$

where $\tau = 1/(C_\mu\omega)$ is the turbulence time scale with $C_\mu = 0.09$ and unlike the EARSM by Wallin and Johansson, the Durbin limitation is not employed to τ . Eddy viscosity for this EARSM is given by,

$$\mu_T = \frac{C_\mu^* \rho k}{C_\mu \omega} \quad (2.90)$$

where C_μ^* is the anisotropy parameter defined as,

$$C_\mu^* = \frac{\beta'_1}{1 - \frac{2}{3}\eta^2 + 2\xi^2} \quad (2.91)$$

with the β' coefficients and the auxiliary functions given by,

$$\begin{aligned} \beta'_1 &= \frac{\frac{4}{3} - C_2}{2g}, \quad \beta'_2 = \frac{2 - C_4}{2g}, \quad \beta'_3 = \frac{2 - C_3}{g} \\ \eta^2 &= \frac{\beta'_3 \widehat{S}^2}{8}, \quad \xi^2 = \frac{\beta'_2 \widehat{\Omega}^2}{2}, \quad g = g_{ic} + g_c \\ g_{ic} &= f_g(C_1 - 1) + \psi, \quad f_g = 1 + 0.95 \left(1 - \tanh \left(\frac{\widehat{S}^2}{4.6225} \right) \right) \\ \psi &= \frac{\widehat{S}^2}{4 + 1.83 \sqrt{0.8\widehat{\Omega}^2 + 0.2\widehat{S}^2}} \\ g_c &= -0.15M_t \left[\frac{P}{(\bar{\rho}\epsilon)} - \frac{4}{3}M_t \right], \quad \frac{P}{\bar{\rho}\epsilon} = \frac{0.8 + \psi}{1.9} \end{aligned} \quad (2.92)$$

The non-dimensional strain-rate and vorticity magnitudes are $\widehat{S} = \tau S$ and $\widehat{\Omega} = \tau \Omega$, respectively, and the model constants are : $C_1 = 2.5$, $C_2 = 0.39$, $C_3 = 1.25$, $C_4 = 0.45$. Compressibility correction (g_c) in the form of Sarkar's pressure-dilatation model is adopted as per Rung et al. [73]. It models the dilatational part of the pressure-strain correlation, which represents the mechanism of turbulence redistribution and relaxation. Here, $M_t = \sqrt{2k}/\tilde{a}$ is the turbulent Mach number and $\tilde{a}$ is the mean speed of sound.

2.4 Literature Review

Physical insights to the flowfield features help in devising reliable turbulence models. Shock/ turbulence interaction is of particular interest because of the influential role it plays in the aerospace-related applications considered in this work. High-speed shock-dominated flows require refined modelling techniques to provide an acceptably realistic predictive description. The flows considered here usually occur in off-design operational conditions of the high-speed vehicles. In many cases, shock-induced separation is the key feature that requires resolution e.g. SBLIs, however there are also cases where separation occurs because of the geometric conditions but shock/turbulence interaction is still important e.g. backward facing step, blunt bodies, etc. There are also cases, such as the

underexpanded jets and their interaction, where shock/turbulence interaction influences the evolution of the mean flow downstream and the flow recirculation towards the vehicle body. Starting with the simplest mean flow with canonical STI, relatively complex flows with SBLIs and SSLIs in canonical geometries along with a realistic geometry are studied wherein shock/turbulence interaction plays a crucial role in shaping the flowfield and determining the quantities of engineering interest. Performance of several conventional turbulence models in these flows is surveyed from the available literature.

2.4.1 Canonical shock/turbulence interaction

Shock/turbulence interaction has been vastly studied using analytical tools like linear interaction analysis (LIA) and rapid distortion theory (RDT), numerical tools like DNS and LES along with few experiments. A common methodology followed in the literature to study this interaction process is by considering homogeneous and isotropic turbulence interacting with a nominally normal shock in the absence of any walls. This model problem is referred to as canonical shock/turbulence interaction (STI). Physical insights to the shock/turbulence interaction process can be obtained using this canonical problem and the performance of turbulence models can also be evaluated as well. Such insights can then be used along with LIA or DNS to develop turbulence models which are able to predict the shock/turbulence interaction correctly. Additionally, numerical errors at a shock wave associated with non-conservative source terms in the turbulence model equations can also be studied using this problem.

The modal description of compressible turbulence proposed by Kovaszny [82] is generally used to study the canonical STI problem. In this work, the incoming turbulence

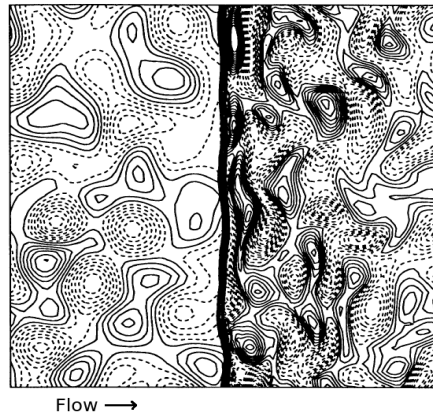


Figure 2.1: A snapshot of transverse vorticity contours from DNS of a Mach 2 canonical shock/turbulence interaction showing turbulence amplification and shock wave undulations (Reproduced from [33]).

is considered to consist of purely the vortical mode. Fig. 2.1, taken from [33], shows an instantaneous snapshot of the turbulent vortical structures interacting with a Mach 2 normal shock wave. The mean flow is from left to right; it is one-dimensional, uniform and steady. The jump in the mean flow quantities across the shock is governed by the Rankine-Hugoniot relations. Turbulence gets amplified across the shock wave and the amplification in the turbulence quantities, i.e., TKE and its dissipation rate, is proportional to the gradients in the mean flow variables. The amplification of the turbulent vorticity fluctuations across the shock is visible in Fig. 2.1. The shock wave shows unsteady oscillations due to the passage of turbulent fluctuations through it.

The model problem simplifies the mean conservation equations as well as the turbulence model equations considerably. Complexities due to the boundary-layer velocity gradients, flow separation, streamline curvature, which are encountered in SBLI flows, are not involved. However, the model problem exhibits a range of physical effects, like unsteady shock oscillations, amplification of TKE across the shock, high heat transfer rates in the shock-turbulence interaction region, etc. The main aim of investigating such a flow is to isolate the net effect of the strong gradients of mean quantities imposed by the shock wave on the turbulence. Such studies provide important insights into the underlying physics of shock-turbulence interaction. It can serve as a building block flow, from which information about the physical interactions could be applied to the modelling of more complex inhomogeneous flows.

The canonical shock/turbulence interaction (STI) has been the subject of study for almost half a century and extensive DNS studies have been carried out, e.g. Hannappel et al. [83], Lee et al. [33, 84], Mahesh et al. [85], Jamme et al. [86], Larsson et al. [58, 87]. These studies mainly focus on understanding the physics of the shock-turbulence interactions. They provide qualitative and quantitative information about the Reynolds stresses, TKE and vorticity amplifications, and variations in turbulence length scales, temperature and density variances across the shock. Figure 2.2 presents a sample DNS data of TKE variation in a Mach 3 shock-turbulence interaction. The turbulent Mach number and the Reynolds number based on the Taylor microscale are 0.11 and 19.7, respectively. The shock is located at $x = 3$, and the TKE is normalized by its values immediately upstream of the shock. The TKE upstream of the shock decays from its inlet value, due to the absence of mean velocity gradients. There is an amplification in its value across the shock, followed by a faster decay due to the increase in the turbulent dissipation rate at the shock. A rapid non-monotonic variation is observed immediately behind the shock and it corresponds to the acoustic adjustment region, as described by Mahesh et al. [85]. Several

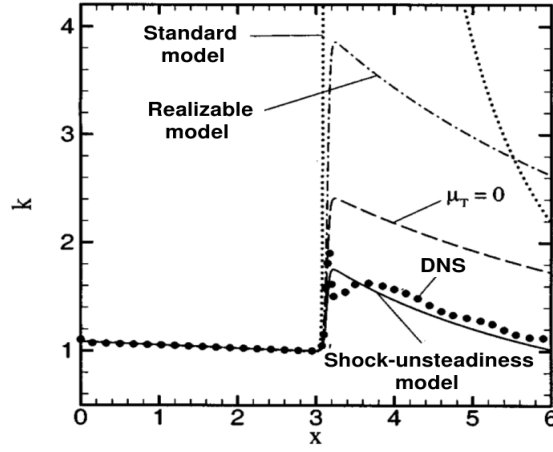


Figure 2.2: Streamwise evolution of TKE in a Mach 3 shock/turbulence interaction obtained using DNS and different turbulence models (Reproduced from [30]).

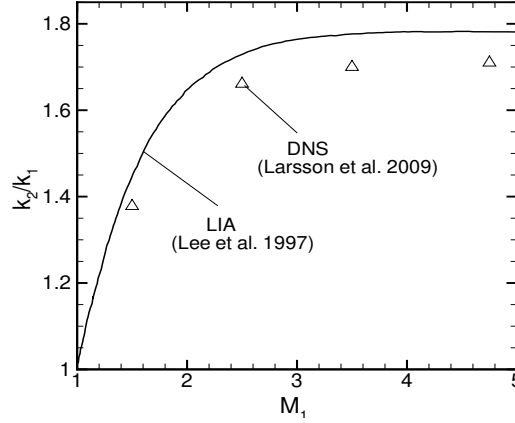


Figure 2.3: Variation of TKE amplification across a normal shock with upstream Mach number obtained from LIA [33] and DNS [58]; subscript ‘1’ denotes the upstream value and ‘2’ denotes the value downstream of the shock.

turbulence model predictions are compared with the DNS data in Fig. 2.2 and they are discussed subsequently.

Theoretical tools like linear interaction analysis (LIA) and rapid distortion theory are also well developed to study the shock-turbulence interaction problem. The rapid distortion theory only accounts for a homogeneous mean flow compression, whereas LIA includes vorticity and TKE generation due to shock-front distortion and unsteady oscillations, along with the mean flow compression. Thus, LIA is a closer representative to the flow physics [88] and is based on the inviscid assumption of the flow [89]. The upstream turbulence is decomposed into Kovasznay modes. Linearised Euler equations are then solved downstream of the shock-wave, with linearised Rankine-Hugoniot relations applied at the shock. The solution gives the amplifications of TKE, fluctuating vorticity

and thermodynamic fluctuations at the shock wave. Figure 2.3 shows amplification in the TKE across the shock as a function of the upstream Mach number, obtained using LIA [33] and DNS [58]. Both the DNS and LIA results show a similar trend; TKE amplification increases till $M_1 = 3$, after which it saturates for higher Mach numbers. DNS results are slightly lower compared to the LIA theory. This is because, DNS solves for the full Navier-Stokes equations, where viscous effects are considered, that brings down the TKE levels slightly as compared to the inviscid linear theory results.

The understanding of physics together with the availability of extensive DNS data and LIA theory results for the canonical shock/turbulence interactions helps in evaluating existing models and developing new closure models for turbulence correlations of interest. In Fig. 2.2, the results obtained using the standard and realizable $k-\epsilon$ models are evaluated against the DNS data of Lee et al. [33]. The results show that the TKE amplification across the shock is overly predicted by these models in comparison with the DNS data. The reason behind this discrepancy can be attributed to the fact that the equilibrium concept of the eddy viscosity breaks down in a highly non-equilibrium flow, such as a shock/turbulence interaction [30]. Suppressing μ_T entirely in the shock region by setting it to zero brings down the TKE amplification level, but still a significant overprediction can be seen. A good match with the DNS data is obtained for the shock-unsteadiness $k-\epsilon$ model [30] (see Fig. 2.2), which is developed using the LIA theory results. Linear theory results have also been used in other modeling works [90, 91] and the ideas have been extended to $k-\omega$ and Spalart-Allmaras turbulence models. Application of these physics-based models to complex two-dimensional SBLI flows [27, 34, 35] has also been carried out and successfully validated against available experimental data.

Another important advantage of studying canonical shock/turbulence interaction is in terms of numerical error analysis. Shock-waves are numerical discontinuities and are prone to large numerical error. The simplified one-dimensional mean flow and turbulence model equations for canonical shock/turbulence interactions can be solved analytically, and the closed form exact solutions serve as a benchmark for assessing the accuracy of numerical methods. Sinha and Balasridhar [75] use this approach to quantify the large numerical error in k and ϵ values, when turbulence models are employed in shock-capturing numerical simulations. They propose a new form of the model equations based on a transformation of the turbulence variables to eliminate these errors, and the new approach is able to match analytical results and DNS data closely. A similar analysis of the $k-\omega$ turbulence model is presented in [92].

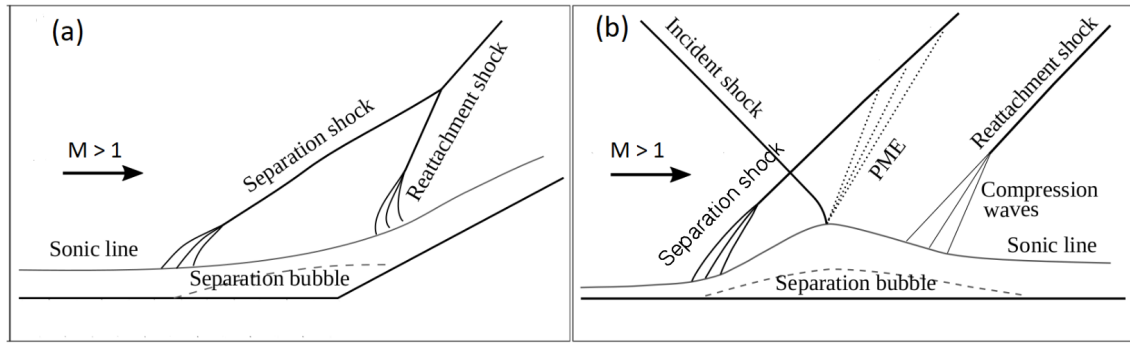


Figure 2.4: Schematic of (a) compression ramp SBLI and (b) impinging oblique shock/boundary layer interaction configuration .

2.4.2 Shock/turbulent boundary layer interaction

Given the importance of shock-turbulent boundary layer interactions (SBLIs) in many practical engineering applications involving high-speed flows, they have been a field of study for past few decades. A few simple canonical geometries have been used widely in both laboratory experiments and simulations to study various aspects of SBLIs. These include the compression ramp, the compression-expansion ramp, the oblique incident/reflected shock wave, cylinder and conical flare configurations, and single or double fins on plates. In most of the supersonic and hypersonic flows of interest, the shock waves produced by the different sources could be strong enough such that the adverse pressure gradient created by them may result in the boundary layer separation. Concerning the boundary layer behaviour, there are no basic differences among the separations induced by an incident shock or ramp or any other source. In all instances, the boundary layer responds in the same way to a given pressure rise regardless of the cause. SBLIs for cases involving compression corner flows and an impinging oblique-shock reflection on a flat plate are the focus of study in this work.

A high-speed inflow ($M > 1$) encountering an angled ramp is illustrated in Fig. 2.4(a). Such geometric configuration can be found in ramp-type intakes of high speed air-breathing engines, deflected control surfaces, etc. The incoming flow “feels” the presence of the ramp upstream of its actual location due to the upstream propagation of pressure signal through the subsonic portion of the turbulent boundary layer. Compression waves emanate from a point upstream of the ramp foot and coalesce into a shock wave further from the wall. The system of compression waves and the shock wave deflect the flow over the ramp. The adverse pressure gradient imposed by the shock wave causes the boundary layer to separate in the vicinity of the compression corner. The presence and size of a separated region is dependent upon the Reynolds number, the incoming

boundary layer thickness, the freestream Mach number, and the ramp angle. The boundary layer reattachment on the ramp gives rise to reattachment compression waves which coalesce into a reattachment shock. The region between the separation and reattachment points is called “separation bubble” wherein the flow recirculates and the velocity profile reverses its direction, as shown in Fig. 2.4(a). The flow remains subsonic inside the separation bubble. The boundary layer goes over the separation bubble to form a free shear layer. Delery and Marvin (1986) summarize the following trends for two-dimensional compression ramp SBLIs at a fixed Reynolds number:

1. the separation length relative to the boundary layer thickness increases with increasing ramp angle at constant Mach number
2. the separation length relative to the boundary layer thickness decreases with increasing Mach number at constant ramp angle

The interaction resulting from the reflection of an oblique shock wave from a turbulent boundary layer is illustrated in Fig. 2.4(b). The oblique shock impingement case finds application in analysis of internal flows such as isolator section of scramjets. A high-speed flow ($M > 1$) over the shock generator produces an incident oblique shock. This incident shock impinges on flat plate and is reflected. The adverse pressure gradient across the incident oblique shock causes the boundary layer to separate on flat plate and forms a separation bubble. The separation bubble acts as a ramp causing compression waves, which coalesce to form an oblique separation shock. The interaction of incident shock and separation shock generates additional shocks, i.e. induced shock and transmitted shock. Transmitted shock penetrates the separated viscous flow, where it is reflected as Prandtl-Meyer expansion (PME) wave because there is a near-constant pressure level in the bubble. This causes a deflection of the shear layer towards the wall, where it eventually reattaches. Compression waves emanate from the reattachment location which coalesce to form a reattachment shock.

Predicting the distribution of mean flow properties through a region of SBLI for supersonic and hypersonic flows remains a major challenge for numerical simulation, which, thus far, has had little success. The major problem encountered is the difficulty in finding adequate ways to model the turbulence through the separated-interaction regions in which reverse flow, turbulent nonequilibrium, compressibility, and shock-turbulence interaction effects must be accurately reproduced. The important parameters of engineering interest from the analysis of SBLIs are the surface pressure, heat transfer, skin friction coefficient, force and moment values, and separation bubble size. The correct prediction of the location of flow separation and the separation bubble size determines the correct

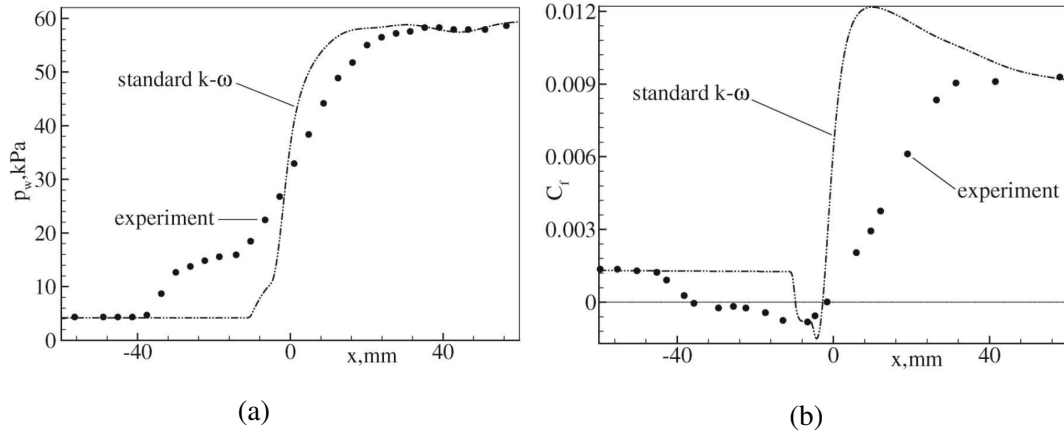


Figure 2.5: Comparison of (a) wall pressure and (b) skin friction distribution obtained using standard $k-\omega$ model and experimental data of Schulein (2006) for shock deflector angle of 14° and freestream Mach number of 5 (Reproduced from [35]).

flow topology in terms of shock angles, strength of shock waves and expansion fan formation. The location and size of the separation bubble has a strong influence on the flow pattern as well as surface pressure, skin friction and heat transfer distribution. Pressure increases at separation point and increases further at reattachment. Skin friction and heat transfer attain high values at reattachment point. Accurate prediction of these quantities is very much essential for producing good designs for aerodynamic devices while avoiding the unwanted consequences of these interactions.

Figure 2.5, taken from Pasha and Sinha [35], shows the variation in wall pressure and skin friction coefficient along the flat plate for the case of an oblique shock impingement. The shock deflector angle is 14° and the freestream Mach number is 5. In case of experimental measurements, surface pressure rises from its free-stream value across the separation shock and still rises at flow reattachment. Adverse pressure gradient of the separation shock decreases the skin friction coefficient C_f from its undisturbed boundary layer value. C_f is negative in the recirculation bubble and attains high values downstream of the interaction region. For the computations using the standard $k-\omega$ model, the predicted separation point is far downstream of its experimentally measured location. The pressure rise at the separation shock is smaller, followed by a higher pressure gradient in the reattachment region. The zero crossing of the computed C_f determines the separation bubble size as 7.6 mm. The corresponding experimental value is 34 mm. The steep rise in skin friction at reattachment grossly overpredicts the measurements. C_f decreases further downstream to values closer to the experimental data. Thus, there is a significant mismatch in the computational and experimentally measured values of surface pressure and skin friction coefficient in the interaction region.

Figure 2.6 shows the heat transfer rate along the flat plate for the identical flow situation. The experimental data shows an increase in the heat transfer rate across the separation shock. At the flow reattachment, the heat transfer rate further increases. The trend of the heat transfer rate is similar to that of surface pressure. However, the heat transfer rate decreases away from the interaction region. A similar trend is obtained using the standard $k-\omega$ computations. However, apart from the mismatch in the separation bubble size and its location, there is a significant mismatch in the heat transfer rate at the flow reattachment. Infact, the $k-\omega$ computation overpredicts the heat transfer rate at the flow reattachment by nearly two times when compared with the experimental data.

The standard $k-\epsilon$ and $k-\omega$ models give similar prediction trends for SBLIs in other geometric configurations like compression corner, cone-flare, etc. [94]. A general tendency of the standard models is to predict significantly high values of TKE in the interaction region thereby inhibiting or delaying the separation and giving considerably smaller recirculation regions, wall-pressure variation that is seriously misrepresented, and excessive skin friction and dramatically higher heat transfer rates at the reattachment location. In the past few decades, many modifications, e.g. realizability constraint, compressibility correction, length-scale modification, and rapid compression correction, to the standard $k-\epsilon$ and $k-\omega$ turbulence models have been proposed to improve the model's prediction capabilities. However, for cases involving shock-induced separation, such modifications do not significantly improve the performance of the standard models.

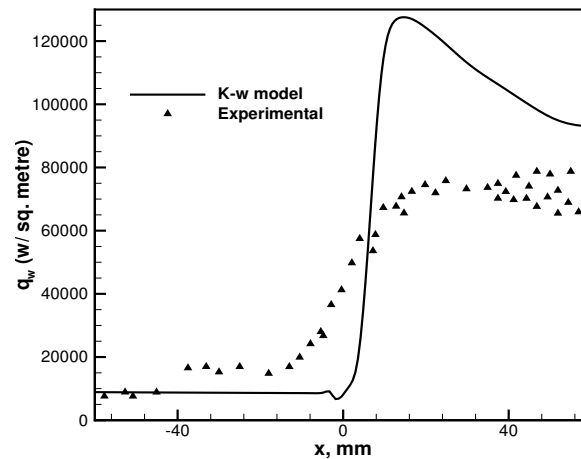


Figure 2.6: The shock-impingement case with shock deflector angle of 14° and freestream Mach number of 5 shows that the standard $k-\omega$ model overpredicts the heat transfer-rate by about two times at the reattachment point compared with the experimental data of Schulein [93].

The shock-unsteadiness (SU) model proposed by Sinha et al. [30] is one of the few models available in the literature which specifically targets to improve the performance of standard models for such flows by considering the shock/turbulence interaction physics. The SU model has been tested in supersonic and hypersonic compression corner, oblique shock impingement/reflection, cone-flare, etc. geometries [34, 94]. The results show significant improvement over standard models in predicting the separation-shock location, flow separation and reattachment, and surface pressure distribution. However, discrepancies still remain in terms of the slight underprediction of separation. Also, the dependence of the model parameter on the upstream shock-normal Mach number (M_{1n}) makes the CFD implementation of the model difficult and cumbersome because M_{1n} is not readily available and is a non-local quantity. A procedure to evaluate the shock-normal Mach number at points upstream of a shock wave is presented in [34] for the compression corner geometry. The upstream direction is identified by the local streamline, and velocity and temperature at an upstream point outside the shock is used to compute the Mach number. The direction of the shock wave is then obtained in terms of the local mean pressure gradient, and is used to compute the component of the upstream Mach number normal to the shock wave. However, for more complex flowfields with multiple shock systems, this can be a complicated exercise thereby adding limitation to this model. This aspect is discussed in detail in Chapter 4.

The standard SST $k-\omega$ model is another advanced turbulence model which was specifically developed to correctly predict separated flows, albeit in the incompressible regime. The model gives excellent predictions for incompressible as well as low-speed subsonic and transonic flows with adverse pressure gradient and separation [2, 37–39]. For the transonic axisymmetric bump geometry, surface pressure distribution along with shock location predicted by the model matches well with the experiment for a range of free-stream Mach numbers of 0.85 – 0.925 [2]. However, the model significantly overpredicts the separation in case of Mach 2.3 supersonic impinging shock case with separated flow [2]. The model is also shown to incorrectly predict the Reynolds stresses, especially the anisotropy in normal Reynolds stresses, owing to the use of isotropic Boussinesq hypothesis [95].

Application of the SST model to Mach 2.9 supersonic impinging shock cases with varying strengths of SBLIs in terms of the flow deflection angle results in significantly bigger separation than the experiments for the separated flow cases [2]. Similarly, the model predicts almost twice the size of separation than the experiments for Mach 2.8 supersonic compression corner cases for moderately to fully separated flows [2]. Also, for a sonic underexpanded jet injected into a Mach 3.75 supersonic cross flow, the resulting

shock-induced separation ahead of the jet injection slot over the flat plate is significantly overpredicted by the model [41]. The discrepancy in the results is found to increase with the increase in jet exit to ambient pressure ratio. In general, SST model is found to overpredict the separation extent in case of supersonic SBLI flows and thus results in an incorrect surface pressure and skin friction distribution.

On the other hand, the SST model significantly underpredicts the separation for Mach 8, 33° compression corner SBLI [17]. The model results also show a significant underprediction of separation for 38° compression ramp case with a free stream of Mach 9.22 [21]. Similar underpredictions are also obtained for cone-flare and compression ramp geometries with fully separated flows for a free stream of Mach 11 [96, 97]. However, for the HiFire I configuration with Mach 7.9 freestream and a 33° flare, SST model significantly overpredicts the size of the separated region at the cylinder/flare junction [98]. Thus, standard SST model gives inconsistent predictions for the shock-induced separation for the hypersonic SBLI flows.

A number of variants of the standard SST model are available in the literature (see Appendix D). Brown [17] computes a series of hypersonic SBLI cases with a modified version of the standard SST model. The computations show that the model slightly overpredicts the separation for Mach 5 impinging SBLI with 10° shock generator angle while it underpredicts the separation for 14° case. Model underpredictions for the separation size are also observed for Mach 8.2 impinging shock with 10° shock generator angle. Overall, the standard SST model along with its variants (as discussed in Appendix D) are found to give inaccurate predictions for separation size and location along with surface pressure and skin friction distribution for supersonic and hypersonic SBLI cases.

On the other hand, application of NLEVMs and EARSMs is generally limited to predict transonic SBLIs, where they show improved predictions compared to the standard Boussinesq hypothesis-based models [64, 65, 68, 73, 74]. It is to note that, NLEVMs and EARSMs provide only a constitutive stress-strain relation and still require two-equation baseline models. Many times the choice of the baseline two equation models is arbitrary, i.e. either a $k-\epsilon$ or $k-\omega$ model can be used without changing the model coefficients. Recently, the EARSM by Wallin and Johansson [70] was applied to Mach 5 impinging oblique shock cases of Schulein [93] in the $k-\omega$ model framework. The model is shown to give smaller separation than experiments for two separated flow cases, however significant improvements over the standard $k-\omega$ model are obtained.

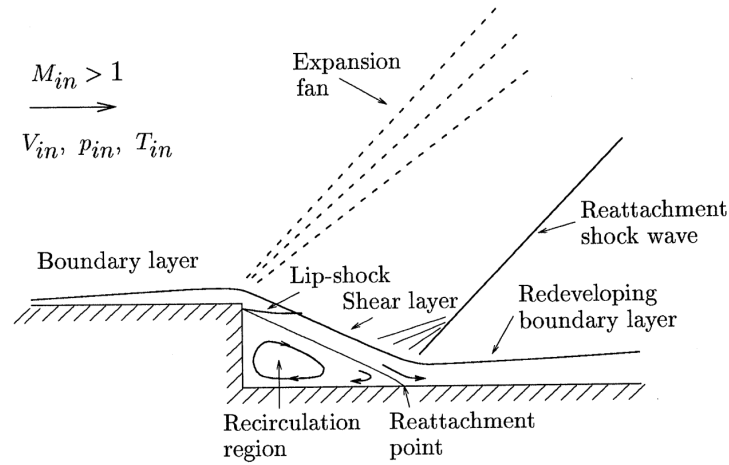


Figure 2.7: Schematic of a supersonic flow over a backward facing step [102]

2.4.3 Supersonic flow over a backward-facing step

The geometric configuration of backward-facing step (BFS) is found in many of the aerospace-related applications including scramjet, missile, rocket, and spacecraft. Backward-facing step can generate recirculation zones with low flow velocity and prolong the residence time of the flow formed in the step corner. Hence, in the recent years, BFS geometry has drawn considerable attention as the flame holder in combustor design of a scramjet engine [99–101]. Also, space launchers such as Ariane 5, have a geometric discontinuity at the end of their main stage ahead of the cryogenic engine, similar to a BFS configuration. Supersonic flow over a BFS poses complex physics involving boundary-layer separation, flow recirculation, shock-boundary layer interaction, and flow reattachment. Thus, the simple geometry of BFS can be an ideal candidate for the assessment of the RANS based models in supersonic flow.

Supersonic flow over a backward facing step is characterized by a complex combination of several interesting fluid dynamics phenomena and includes flow separation, recirculation, reattachment, and viscous-inviscid interactions, as shown in Fig. 2.7. The turbulent boundary layer developed over the flat surface undergoes expansion around the step corner with the formation of a centred Prandtl-Meyer expansion (PME) fan. Experimental observations suggest that the flow separation occurs slightly below the corner, and a lip shock is formed [99–101]. The boundary layer which is separated behind the step develops into a free shear layer in a region of essentially constant pressure. The shear layer approaching the wall downstream of the step is compressed, and the low-velocity part of the shear layer reverses into the slowly circulating region within the base cavity. In the base region the pressure is low and nearly constant. Downstream of the reattachment, the high-velocity part of the shear layer overcomes the pressure rise at reattachment and

forms a new uniform stream with the re-developed boundary layer. The external flow which turns beyond the step corner through the expansion fan, turns back to a direction approximately parallel to the inflow by the oblique reattachment shock wave. The interaction between the reattachment shock and attached boundary layer, i.e. SBLI over such configurations share similar physical mechanisms as that seen for the impinging shock and compression ramp cases. However, in contrast to the SBLI cases discussed in Section 2.4.2, supersonic flow over a BFS shows a distinctly different flow topology, as shown in Fig. 2.7.

The practical significance of the BFS geometry in many of the engineering applications has led to a number of experimental studies that are performed to understand the physical behavior of BFS flows [99–101]. Analytical methods, such as the classical Chapman and Korst theory, allow for a rather accurate prediction of the BFS base pressure, but they cannot predict the entire flow field. The prediction of the supersonic flow over a BFS in its complexity is possible only by means of the Navier-Stokes solution. However, the complexity of the flow field in terms of the large separation, recirculation, and flow reattachment along with SBLI cause significant numerical challenges to accurately simulate this problem.

BFS flow-field is often used as a benchmark problem to test the numerical schemes and validate the performance of turbulence models [103]. Recent interest in the RANS based studies is mainly driven by application of BFS geometry in scramjet engines as flame-holders [104–108]. Such studies usually employ $k-\epsilon$ and $k-\omega$ based eddy viscosity models for the flow field predictions. The $k-\epsilon$ model is found to be inappropriate for this flow field and it grossly underpredicts the reattachment length of flow and gives inaccurate predictions of mean quantities in the near wall region behind the step [106]. On the other hand, $k-\omega$ and SST models are found to be good for this case and give a better representation of the mean flowfield than the $k-\epsilon$ model [106]. However, these models slightly overpredict the separation region. Interestingly, EARSM-based advanced turbulence models have never been applied to predict the supersonic flow over a BFS geometry in the available literature.

2.4.4 Supersonic axisymmetric base flow

Various aerospace-related vehicles, such as high-speed projectiles, powered missiles, reusable launch vehicles etc., comprise of an axisymmetric body with a blunt base. Supersonic flows found behind such bodies are referred to as supersonic axisymmetric base flows in this work. Such vehicles at high-speed experience massive separation at the base resulting in the formation of a recirculation region behind the base that is respon-

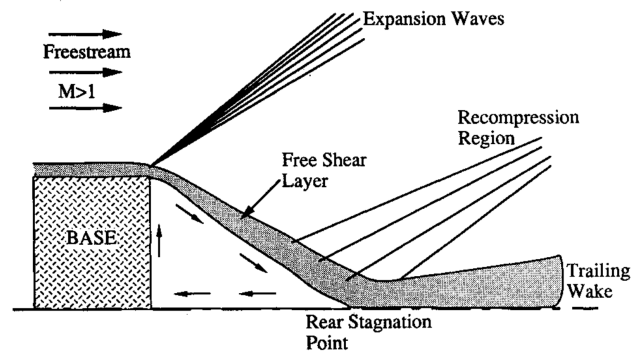


Figure 2.8: Schematic of a supersonic axisymmetric base flow [109].

sible for low base pressure, thus causing aerodynamic drag or base drag. The base drag forms a large part of the total aerodynamic drag (can be as high as 50% or more) and is thus crucial in determining the range, terminal velocity, or the payload carrying capacity of such vehicles. On the other hand, flow behind a launch vehicle at very high altitudes can result in higher pressure loads in the base regions because of the reverse flow of exhaust gases caused by the multi-jet interaction phenomenon. This may inadvertently lead to an increase in the thrust of the vehicle, however, at the same time heat loads are also significantly increased as well. The extensive flow separation can also reduce control effectiveness of the vehicle if the control surfaces are covered by the separated region. Hence, the aerodynamic characteristics of such high-speed vehicles are strongly influenced by the base flowfield in their flight path. An accurate estimation of the base flowfield and the pressure acting on the base are, therefore, important issues for the design of such vehicles. The canonical and simple geometry of a cylindrical body with a blunt base is studied here as it provides for a rather challenging test case for turbulence model validation.

The flowfield over a cylindrical blunt body is similar to that of the back-facing step geometry. A schematic diagram of the mean flowfield structure in the near wake of a cylindrical afterbody aligned with a supersonic flow is shown in Fig. 2.8. The supersonic afterbody freestream flow undergoes a strong expansion centered at the base corner as the turbulent boundary layer separates geometrically from the body. A free shear layer is formed which separates the outer inviscid flow from a relatively large recirculation region immediately downstream of the base. The intense turbulent mixing and energy exchange that characterize the free shear layer are important in determining the flowfield properties throughout the near wake including the recirculation region. As the free shear layer approaches the axis of symmetry, a recompression process occurs which eventually realigns the flowfield with the axis. A rear stagnation point, where the mean velocity vanishes, is located on the centerline and separates the recirculation region from the wake

which develops downstream. The interaction of the recompression waves with the free shear layer, i.e. SSLI, plays a crucial role in determining the recirculation region and hence the base properties.

The base flow field discussed above has been subjected to numerous RANS-based numerical investigations using $k-\epsilon$, $k-\omega$, and SST $k-\omega$ models [110–118]. The $k-\epsilon$ model is found to underpredict the size of the recirculating region, similar to the BFS flow configuration and gives inaccurate mean and turbulent quantities in the flowfield. A number of variants of the standard $k-\epsilon$ model are also assessed, but they generally show significant overprediction of the separation bubble and the base pressure coefficient. On the other hand, unlike the BFS case, $k-\omega$ model is found to give poor predictions than the $k-\epsilon$ model for these cases and it significantly mismatches the base pressure distribution. Menter SST $k-\omega$ model is shown to underpredict base pressure and reattachment length, however gives overall better results than the $k-\epsilon$ and $k-\omega$ models.

Compressibility corrections to $k-\epsilon$ and SST models are found to adversely impact the model predictions and degrade the results. Such modifications often result in significantly larger reattachment lengths and give erroneous base pressure distributions [119]. Advanced turbulence models in the form of algebraic stress models (ASMs) and explicit algebraic Reynolds stress models (EARSMS) have also been applied to such configurations [110, 120, 121]. These models are found to improve the predictions than the standard $k-\epsilon$, $k-\omega$, and SST $k-\omega$ models and give a better representation of the mean and turbulent quantities. However, discrepancies still remain in terms of the separation bubble size and base pressure distribution predictions.

2.4.5 Underexpanded supersonic jet

Many practical engineering applications involve underexpanded supersonic jets, such as the exhaust and plumes of rockets. An underexpanded jet occurs whenever the nozzle exit jet pressure is more than the ambient pressure, which is the case with rockets at high altitudes. The structure of these jets has been associated with many problems such as reverse flow towards the base in rocket nozzle boosters [122] and jet noise [123].

Figure 2.9(a), adapted from Pack [124], shows the structure of an underexpanded axisymmetric supersonic jet exhausted into the still air, and the interplay between the Mach waves and compression waves is observed. At the lips of the nozzle exit, Prandtl-Meyer expansion fans are formed, which are shown in terms of the dashed lines representing the Mach waves. These Prandtl-Meyer expansion fans expand the fluid downstream of the nozzle exit to ambient conditions. There is a discontinuity in velocity across the jet boundary, however, pressure remains constant. The jet boundary bents towards the jet

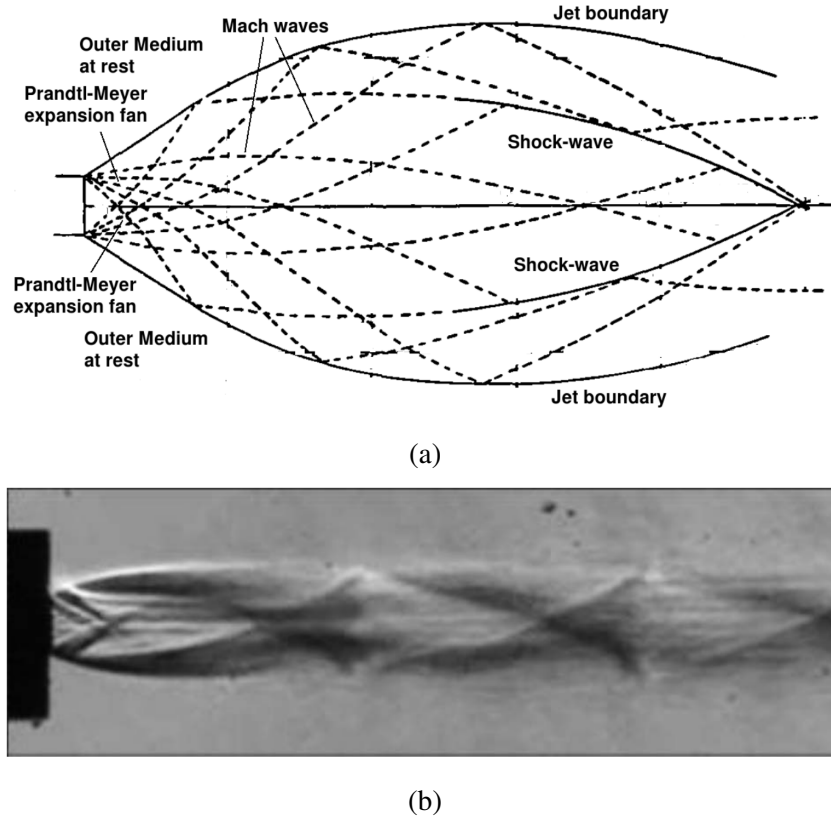


Figure 2.9: (a) Structure of a supersonic underexpanded jet exhausted into still air (Adapted from Pack 1948) (b) Schlieren image of a Mach 2.53 jet with NPR (P_j/P_∞) = 3.12, where P_j and P_∞ are the nozzle exit and freestream static pressure values, respectively (Taken from [36]).

centerline to maintain a constant pressure along it. In order to turn the flow parallel to this boundary, the Mach waves intersect the jet boundary and reflect back to the jet centerline as compression waves. These compression waves coalesce to form a shock wave, known as the incident or intercepting shock. The incident shock can be an oblique shock or a barrel-shaped shock, depending on the degree of under expansion of the jet. Also, depending on the nozzle pressure ratio (NPR) values, the incident shock may undergo a regular reflection at the jet centerline or can result in a Mach reflection. A regular reflection leads to the formation of a reflected shock, whereas, the Mach reflection leads to the apparition of a normal shock, called the Mach disk.

The flowfield discussed in Fig. 2.9(a) corresponds to a single shock-cell structure and in reality the underexpanded jet consists of a number of such shock-cells as shown in Fig. 2.9(b). Several RANS-based studies have been carried out in the past to predict the structure of such underexpanded jets [36, 125–129]. Mean flow decay is the principal feature of this flow which characterises the rate of mixing of any turbulent jet. Shock-

cell decay results from shock-mixing layer interactions. Standard RANS models fail to provide an adequate representation of the shock/turbulent free shear layer interaction, i.e. SSLI. These models are found to significantly overpredict turbulence levels leading to an overprediction of mixing rates. Hence, they lead to an incorrect prediction of shock wave attenuation.

The standard $k-\epsilon$ model severely overpredicts the rate at which the shocks dissipate owing to an overprediction of turbulence levels within the shear layer [36, 127]. More turbulent shear layers results in weaker reflection of shock waves, i.e., shock-cells decay faster. To accurately predict the supersonic underexpanded jet, modifications to the turbulence models from modifying the modelling constants to that modeling of compressibility effects on the turbulence energy dissipation rate are examined in the literature [36, 127]. However, such modifications still show inconsistencies compared to experiment. Application of EARSM-based advanced models result in better prediction of mean flow properties compared to the standard models [120, 130], however discrepancies still remain.

2.4.6 Multi-jet interaction and the reverse flow

The clustering of multiple engines and hence exhaust nozzles for thrust generation is a common practice followed in rocket propulsion. During the ascent phase of a rocket, the freestream pressure decreases with altitude. At high altitudes, the nozzle exit pressure is more than the ambient pressure resulting in an underexpanded condition. The underexpanded jets coming out from the neighbouring nozzles expand not only longitudinally but also radially. The radial expansion of these jets leads to their mutual interaction which can result in exhaust reverse flow towards the rocket engine base.

Musial and Ward [131] and Mehta et al. [8] put forward a flow model for the base flowfield induced due to multi-jet interaction with the help of wind tunnel experiments and CFD simulations. The flow model is proposed using a four-engine configuration and it describes the various base flow regimes due to multi-jet interaction for the ascent of rocket from lift-off to in-space environments. These regimes are observed for all flight NASA launch vehicles [8]. Figure 2.10 shows one such flow regime for the medium altitude case with moderately underexpanded plumes. The base flowfield observed for the test cases considered in this work falls in this regime and plume induced flow separation is not observed. A number of basic flow structure interactions, identified in Fig. 2.10, are observed in a multi-jet interaction flowfield. Table 2.1 describes the major features of these different interactions.

The boundary layer developed over the vehicle body undergoes an expansion at the base shoulder and separates as a free-shear layer. This free-shear layer encloses the

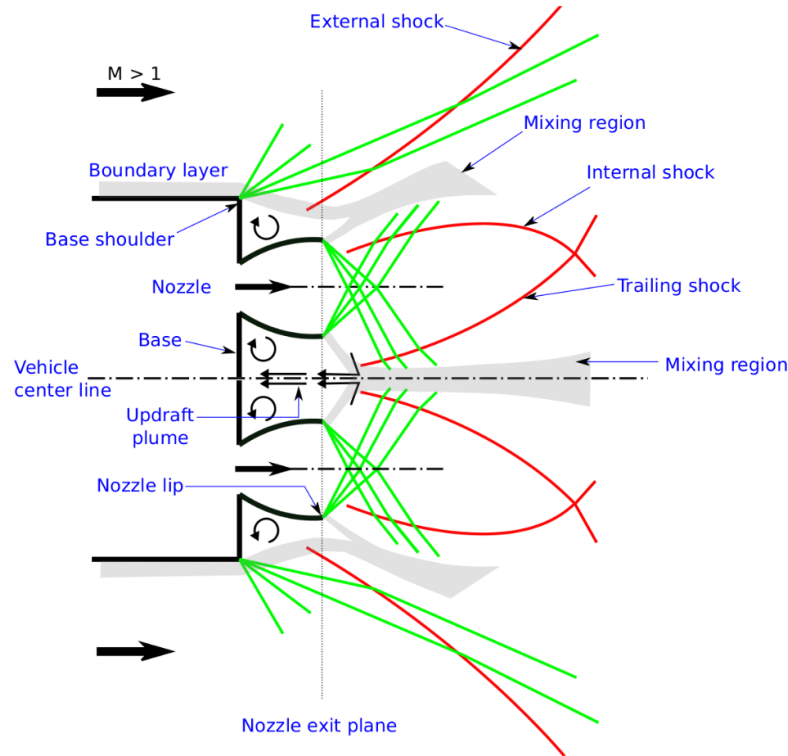
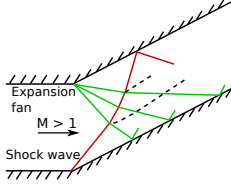
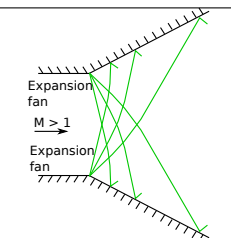
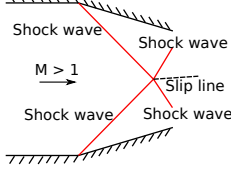
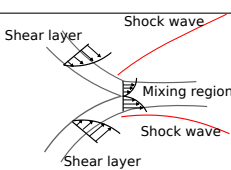


Figure 2.10: A schematic showing the multi-jet interaction flowfield and the associated base flow for moderately underexpanded plumes.

low pressure base flow region with embedded subsonic vortices. At the nozzle exit, the exhausted high pressure gas expands rapidly by generating an expansion fan at the nozzle lip. The free-shear layer emanating from the base shoulder and the plume boundary, which separates the exhaust gas from the ambient flow, interact with each other near the nozzle-exit plane. This interaction results in the formation of an “external shock” and it spreads circularly around the vehicle’s rear. Also, an “internal shock” is formed which is embedded inside the plume.

The radially expanding plumes from the neighbouring nozzles interact with each other at the model centerline downstream of the nozzle-exit plane. As a result of this mutual impingement, the supersonic exhaust gases are deflected, producing trailing shock waves inside the interacting plumes. The low-energy gas within the mixing region of the interacting jets, which is unable to negotiate the adverse pressure due to the trailing shock wave, flows back into the base. A stagnation point is formed at the intersection of the plumes and the resulting reverse flow is called as the “updraft plume”. The recirculated mass flows into the center area in a flow passage formed by the jet plume boundaries and the nozzle external walls and flows out of the base through the vent area between the nozzles.

Table 2.1: Basic flow structure interactions with their salient features observed in a multi-jet interaction flowfield

Type of interaction	Interaction	Representation	Interaction features
Inviscid-Inviscid	Oblique Shock-Expansion fan		• Transmission through each other • Reduced strength of the transmitted waves
	Expansion fan-Expansion fan		• Transmission through each other
	Oblique Shock-Oblique Shock		• Edney type I interaction • Transmission of the shocks through each other
Viscous-Viscous	Shear layer-Shear layer		• Formation of shocks on either sides of the two shear layers • Compression of free-stream shear layer across the external shock • Mixing of the shear layers after the external shock
Viscous-Symmetry line	Shear layer-Symmetry line		• Formation of trailing shock at the intersection point • Compression of shear layer across the trailing shock

The works of Goethert [132], Page and Dixon [133], and Lamb et al. [134], Marion et al. [135], and Taylor and Tou [136] are the major analytical contributions in the area of predicting base pressure and convective heating rates for multi-nozzle configurations. These works divide the complex multi-plume induced reverse flowfield into different components which are then analyzed individually. The final solution for the base pressure and base heat flux makes use of results obtained from these different component analyses. However, such methods provide only qualitative estimates as the solution inherently contain all errors and assumptions present in each component analysis. The complexity of

the multi-nozzle configurations and the multi-plume induced reverse flow phenomena has restricted attempts to develop a single analytical model for the entire flowfield. So, experiments and CFD simulations are the only viable means to study this problem in its entirety.

Majority of the experiments to study the multi-nozzle plume interaction and the associated base flowfield have been performed in the 1960s. Notable experimental works in this area include the experiments performed by Goethert and Barnes [137], Goethert [132], Musial and Ward [131], Charczenko and Hayes [138], Cubbage [139], Wasko and Cover [140], Brewer and Craven [141], and Desikan et al. [122]. The main objectives of these tests were to provide base pressure and heating trends due to various changes in geometry configuration and ambient pressure. However, these experimental investigations have been limited to obtaining base surface measurements; consequently, the flowfield and the shock structure has not been analysed in detail and it is difficult to generalize the findings.

CFD simulations of the multi-jet interactions flows are also limited, as modeling of such flows is a challenging task. Mehta et al. [7, 8], Nallasamy et al. [142], Ebrahimi [143], Wang [144], Gusman et al. [15], Deere et al. [145], and Zhou et al. [146] are the notable CFD related works in this area. Majority of these works aim at providing the best CFD practices for multi-plume simulations by studying the effects of mesh resolution, nozzle exit and other boundary conditions, multi-species gas model and single species gas model assumptions on the base pressure and base heat flux. However, these studies do not attempt to study in detail the multi-jet interaction flowfield and the reverse flow and the effect of turbulence on the prediction of base pressure and base heat flux.

The complexity of the multi-jet interaction flows arises due to various flow structures, flow regimes and large thermal and momentum gradients within the base region. Such flows span from subsonic to supersonic fluid dynamics regimes and consists of regions of strong inviscid/viscous interactions, free-stream shear layers, wall boundary layers (external and internal), shocks, separation and recirculation regions, and plumes/plume interactions. Additionally, the base flowfield induced due to the multi-jet interaction can be highly three-dimensional. Also, the complex multi-nozzle configuration requires an elaborate grid generation and thus CFD simulation of these flows is an exceptionally challenging task. However, three-dimensional Reynolds-averaged Navier-Stokes (RANS) simulations still remain a viable option to predict the multi-jet interaction flowfield and the base pressure and base heat flux.

2.5 Summary

High-speed flows encountered in the aerospace-related applications generally involve complex physical processes and viscous-inviscid interactions, especially the interaction of shock wave and turbulence. Such phenomena can be found in different geometric configurations such as air-intakes, isolator regions, and BFS configurations in the combustor of a scramjet, behind the blunt bases of rockets, missiles, launch vehicles, etc. RANS-based two-equation eddy viscosity turbulence models employing a linear stress-strain constitutive relation, called as Boussinesq hypothesis, are routinely used for the engineering predictions of these flows. However, the standard $k-\epsilon$ and $k-\omega$ models are found to give incorrect shock/turbulence interaction physics. Application of these models to high-speed flows often result in serious underpredictions of separation regions with inaccurate representation of the associated mean flow field and turbulence quantities. This is true for shock-induced separations in case of SBLIs, separation due to geometry change in case of BFS and base flows. Also, the decay of mean flow and the length of underexpanded jet is underpredicted by these models. Menter SST $k-\omega$ model shows a better performance than the standard models in some of the high-speed separated flow cases. But the model predictions are inconsistent across a wide range of flow conditions and sometimes it even results in poorer predictions than the standard models. Introduction of compressibility corrections is a commonly followed approach to deal with the poor performance of these models, however, such modifications often degrade the predictions further. A non-linear constitutive stress-strain relation, as given by EARSMS, is another modelling avenue which is gaining popularity recently. EARSMS-based models generally result in better predictions than the standard linear eddy viscosity based models, but discrepancies still remain.

Chapter 3

Simulation methodology and validation cases

One, two, and three-dimensional RANS-based steady-state simulations are performed in this work using a well-validated in-house CFD code. The in-house CFD code has been successfully used in the past to compute a wide range of high-speed flows [27, 34, 35, 147–149]. Simulation details, including the numerical method employed, test cases considered for turbulence model validation, domain, boundary conditions, and grids used, is provided in this chapter. Structured single block grids are used for the 2D simulations of SBLI test cases and the grids are generated using an in-house code. On the other hand, structured multi-block grids, generated using the commercial software *Gridgen ver15*, are used for the 3D simulations of base flows, underexpanded jet, and multi-jet test cases. The interaction of homogeneous isotropic turbulence with a normal shock wave, referred to as canonical shock/turbulence interaction, is solved in a 1D RANS set-up.

3.1 Canonical shock/turbulence interaction simulations

The one-dimensional Reynolds-averaged Navier-Stokes equations are solved for the mean flow and the standard and the newly proposed forms of k - ϵ model are used for turbulence closure. The diffusive fluxes are not included in the computation because their contribution is expected to be small in the current shock-turbulence interaction. All the equations are solved in their non-dimensional form. The dimensionless and normalized quantities are defined as :

$$\begin{array}{lll} x = x^* \kappa_0 & t = t^* / (1/a_\infty^* \kappa_0) & u = u^* / a_\infty^* \\ \rho = \rho^* / \rho_\infty^* & p = p^* / (\rho_\infty^* a_\infty^{*2}) & T = T^* / T_\infty^* \\ R = R^* / (\gamma R^*) & k = k^* / a_\infty^{*2} & \epsilon = \epsilon^* / (a_\infty^{*3} \kappa_0) \end{array}$$

where $(.)^*$ represent quantities in their dimensional form and $(.)_\infty$ represent the corresponding free-stream values. Here, a is the mean speed of sound, $\gamma = 1.4$ and $R = 287.1$ J/kg.K is the specific gas constant. The most energetic wave number $\kappa_0 = 4 \text{ m}^{-1}$ in the incoming turbulence field is taken as the characteristic length scale.

A finite-volume approach is used to solve the mean conservation equations fully coupled with the k - ϵ model equations. The system of governing equations in the conservation form can be written as,

$$\frac{\partial}{\partial t} \underbrace{\begin{pmatrix} \rho \\ \rho u \\ E \\ \rho f \\ \rho g \end{pmatrix}}_{[U]} + \frac{\partial}{\partial x} \underbrace{\begin{pmatrix} \rho u \\ \rho u^2 + p + (2/3)\rho u^{-2/3} p^{-2c/3} f \\ (E + p)u + (2/3)\rho u^{1/3} p^{-2c/3} f \\ \rho u f \\ \rho u g \end{pmatrix}}_{[F]} = \underbrace{\begin{pmatrix} 0 \\ 0 \\ 0 \\ -\rho g(u^{\frac{5}{3}-c_w} T^\alpha p^{2c/3}) \\ -c_2 \rho g^2(u^{\frac{5}{3}-c_w} T^\alpha p^{2c/3})/f \end{pmatrix}}_{[S]} \quad (3.1)$$

Here, $[U]$ is the vector of conserved variables, $[F]$ is the vector containing inviscid fluxes and $[S]$ is the vector containing the source terms. The total mean energy E is given by,

$$E = \frac{p}{(\gamma - 1)} + \frac{1}{2} \rho u^2 + \rho u^{-2/3} p^{-2c/3} f \quad (3.2)$$

The non-conservative form of k - ϵ equations can also be written in a form similar to Eq. (3.1). The equations are integrated using the conservation form of Lax-Friedrichs scheme. A fixed CFL value of 0.2 is used for all the computations, which renders the scheme to be first order accurate in space and time. The turbulent source terms are evaluated at the cell centers. In case of the simulations of non-conservative form of the turbulence equations, the velocity, pressure and temperature gradients in the production terms are discretized using a second-order accurate central difference scheme. Explicit time integration is achieved using the first order forward Euler method. The code has been successfully used to reproduce the results from [75], which were obtained using Steger-Warming flux-vector splitting method. However, Lax-Friedrichs scheme requires more number of grid points to obtain identical results for a particular Mach number when compared with the code used in [75].

The mean flow Mach number for the test cases range from low supersonic to nearly the hypersonic limit. DNS data for a limited number of cases are presented by Larsson and Lele [58], where the turbulent Mach number M_t , defined as $M_t^2 = 2k^*/a^{*2}$, immediately upstream of the shock wave is 0.22. The Reynolds number based on Taylor microscale Re_λ is 40 at this location. The computational domain ranges from $x = -7.6$ to $x = 29.6$. The normalized value of TKE just upstream of the shock is calculated as

$$k_1 = \frac{M_t^2}{2} \quad (3.3)$$

The Taylor microscale is given by $\lambda^* = \sqrt{10\nu^*k^*/\epsilon^*}$ and the Reynolds number based on Taylor microscale is defined as $Re_\lambda = \lambda^*u'_{rms}/\nu^*$. For homogeneous-isotropic turbulence, we have $u'_{rms} = \sqrt{2k^*/3}$. Using these relations, the normalized value of turbulence dissipation rate immediately upstream of shock can be calculated as

$$\epsilon_1 = \frac{5}{\sqrt{3}} \left(\frac{M_t^3}{\kappa_0\lambda^*} \right) \frac{1}{Re_\lambda} \quad (3.4)$$

Using a non-dimensional value of Taylor microscale $\kappa_0\lambda^* = 0.84$, we get $k_1 = 0.0242$ and $\epsilon_1 = 9.15 \times 10^{-4}$. Upon solving the equations for homogeneous isotropic turbulence governing the evolution of k and ϵ from the inlet station to the shock location (for more details, see Appendix A), we get the relations for the inlet values of the turbulence variables in their normalized form as

$$\frac{k_0}{\epsilon_0} = \frac{k_1}{\epsilon_1} - t_{sh} \quad , \quad \epsilon_0 = \epsilon_1 \left[1 + \frac{t_{sh}}{\left(\frac{k_1}{\epsilon_1} - t_{sh} \right)} \right]^{\frac{c_{\epsilon 2}}{c_{\epsilon 2} - 1}} \quad (3.5)$$

where, $t_{sh} = (c_{\epsilon 2} - 1)x_{sh}/u_1$, $c_{\epsilon 2} = 1.2$ and $x_{sh} = 6.83$ is the shock location from the inlet. The canonical shock-turbulence interaction flows computed in this work are listed in Table 4.2. The values of k_0 and ϵ_0 for all the cases are obtained by using Eq. (3.5) with the conditions just upstream of the shock taken from DNS of Larsson and Lele [58]. Free-stream and inlet conditions are specified at the upstream boundary. Rankine-Hugoniot jump relations are used to set-up the shock at $x = 0$. For the exit boundary, a Neumann condition of the form $\frac{\partial}{\partial x}[U_{exit}] = 0$ is employed, where $[U_{exit}]$ is the vector of conserved variables at the exit.

The original CFD code is based on the non-conservative form of k and ϵ equations (4.8) and (4.15). Like Sinha et al. [75], the conservative form of the model Eqs. (4.24) and (4.25) can be easily incorporated by interpreting the turbulence variables in the code as f and g , instead of k and ϵ . Minimal changes are then required to the CFD code to implement the conservative form of the k - ϵ equations. These are listed next.

1. The initial and boundary conditions for k and ϵ are transformed to the new variables f and g as per Eqs. (4.18) and (4.22).
2. Computation of the turbulent source terms are modified by dropping the production terms and multiplying the dissipation term by a factor $(u^{5/3-c_w} T^\alpha p^{2c/3})$.
3. The pressure is calculated from the total mean energy Eq. (3.2), which requires numerically solving the algebraic non-linear equation,

$$p + (\gamma - 1)fu^{-2/3}p^{-2c/3} = (\gamma - 1)\left(E - \frac{1}{2}\rho u^2\right) \quad (3.6)$$

at the end of each time step. In this work, Secant method is used and it takes three iterations to get the pressure at each grid point for all the upstream Mach number cases considered.

4. The original turbulence variables k and ϵ are then recovered from the converged flowfield solution by invoking the reverse transform as per Eqs. (4.18) and (4.22).

3.2 Two-dimensional and three-dimensional mean flow simulations

A well-validated in-house CFD code capable of parallel computing is used for the computations. The code uses finite volume formulation to discretize the governing mean flow equations fully coupled with the turbulence model equation. The SBLI test cases are solved using two-dimensional Reynolds-averaged Navier-Stokes (RANS) equations whereas the base flows, underexpanded jet, and the multi-jet cases are solved using the three-dimensional RANS equations. The standard forms of the k - ϵ , k - ω , SST k - ω , EARSM of Rung et al. [73] models along with the newly developed local SU k - ϵ , local SU k - ω , and SUQ-SST models are used for the turbulence closure. The terms in the RANS equations along with turbulence models can be viewed in three categories: the inviscid or convective terms, the viscous terms, and the source terms and their numerical treatment is discussed in the subsequent sections.

In the following sections, the finite volume formulation and flux-vector splitting scheme of Steger-Warming and modified Steger-Warming [152] for solving RANS equations using the $k - \omega$ turbulence model for a perfect gas are presented. The discussion for the finite volume formulation is shown for two-dimensional RANS set-up whereas the numerical method of Steger-Warming is shown for 1D mean flows. However, this can be easily extended to the two and three-dimensional mean flows following a similar methodology.

3.2.1 Finite volume formulation

For the two-dimensional simulations, a total of six equations are solved simultaneously to obtain six conservative variables $\bar{\rho}$, $\bar{\rho}\bar{u}$, $\bar{\rho}\bar{v}$, $\bar{\rho}\bar{E}$, $\bar{\rho}k$, $\bar{\rho}\omega$. Here $\bar{E}$ is the total mean flow energy. From these conservative variables, the six primitive mean flow quantities are calculated as, streamwise velocity $\bar{u}$, wall-normal velocity $\bar{v}$, density $\bar{\rho}$, temperature $\bar{T}$, TKE k and the specific dissipation rate of turbulent kinetic energy ω . The mean pressure $\bar{p}$ is obtained from the perfect gas equation. The two-dimensional RANS system of equations can be written in conservation form as,

$$\frac{\partial U}{\partial t} + \nabla \cdot (\vec{F} + \vec{G}) = Q, \quad (3.7)$$

where U is the set of conservative variable, $\vec{F}$ is the flux vector in the streamwise direction, $\vec{G}$ is the flux vector in flow normal direction, and Q is the source term. The total fluxes are split into the inviscid and viscous part. The subscripts I and V represent the inviscid and viscous terms.

$$\frac{\partial U}{\partial t} + \nabla \cdot (F + G) = Q, \quad (3.8)$$

where

$$F = F_I - F_V; \quad G = G_I - G_V; \quad (3.9)$$

and

$$U = \begin{bmatrix} \bar{\rho} & \bar{\rho}\bar{u} & \bar{\rho}\bar{v} & \bar{\rho}\bar{E} & \bar{\rho}k & \bar{\rho}\omega \end{bmatrix}^T, \quad (3.10)$$

here superscript T stands for the transpose of the matrix.

$$F_I = \begin{bmatrix} \bar{\rho}\bar{u} \\ \bar{\rho}\bar{u}^2 + \bar{p} + \frac{2}{3}\bar{\rho}k \\ \bar{\rho}\bar{u}\bar{v} \\ (\bar{\rho}\bar{E} + \bar{p} + \frac{2}{3}\bar{\rho}k)\bar{u} \\ \bar{\rho}\bar{u}k \\ \bar{\rho}\bar{u}\omega \end{bmatrix}; \quad G_I = \begin{bmatrix} \bar{\rho}\bar{v} \\ \bar{\rho}\bar{u}\bar{v} \\ \bar{\rho}\bar{v}^2 + \bar{p} + \frac{2}{3}\bar{\rho}k \\ (\bar{\rho}\bar{E} + \bar{p} + \frac{2}{3}\bar{\rho}k)\bar{v} \\ \bar{\rho}\bar{v}k \\ \bar{\rho}\bar{v}\omega \end{bmatrix} \quad (3.11)$$

$$F_V = \begin{bmatrix} 0 \\ \bar{\tau}_{xx} + \bar{\rho}R_{xx} \\ \bar{\tau}_{xy} + \bar{\rho}R_{xy} \\ (\bar{\tau}_{xx} + \bar{\rho}R_{xx})\bar{u} + (\bar{\tau}_{xy} + \bar{\rho}R_{xy})\bar{v} + (\kappa + \kappa_T) \frac{\partial \bar{T}}{\partial x} + \left(\mu + \frac{\mu_T}{\sigma_k}\right) \frac{\partial k}{\partial x} \\ \left(\mu + \frac{\mu_T}{\sigma_k}\right) \frac{\partial k}{\partial x} \\ \left(\mu + \frac{\mu_T}{\sigma}\right) \frac{\partial \omega}{\partial x} \end{bmatrix} \quad (3.12)$$

$$G_V = \begin{bmatrix} 0 \\ \bar{\tau}_{xy} + \bar{\rho}R_{xy} \\ \bar{\tau}_{yy} + \bar{\rho}R_{yy} \\ (\bar{\tau}_{xy} + \bar{\rho}R_{xy})\tilde{u} + (\bar{\tau}_{yy} + \bar{\rho}R_{yy})\tilde{v} + (\kappa + \kappa_T) \frac{\partial \tilde{T}}{\partial y} + \left(\mu + \frac{\mu_T}{\sigma_k}\right) \frac{\partial k}{\partial y} \\ \left(\mu + \frac{\mu_T}{\sigma_k}\right) \frac{\partial k}{\partial y} \\ \left(\mu + \frac{\mu_T}{\sigma}\right) \frac{\partial \omega}{\partial y} \end{bmatrix} \quad (3.13)$$

$$Q = \left[0 \quad 0 \quad 0 \quad 0 \quad (P_k - \beta^* \bar{\rho} k \omega) \quad \left(\alpha \frac{\omega}{k} P_k - \beta \bar{\rho} \omega^2 \right) \right]^T \quad (3.14)$$

Here $\bar{\rho}$ is the mean density of the fluid, $\bar{\tau}_{xx}$, $\bar{\tau}_{xy}$, $\bar{\tau}_{yy}$ are the mean viscous stresses acting on fluid particles, R_{xx} , R_{xy} , R_{yy} are the Reynolds stresses, P_k represents the production term of the turbulent kinetic energy. The conservation Eq. (3.7) is integrated over the control volume (see Fig. (3.1)) to give:

$$\frac{\partial}{\partial t} \iiint_{CV} U dV + \iint_{CS} [(\vec{F}_I + \vec{G}_I) + (\vec{F}_V + \vec{G}_V)] \cdot (\hat{n} dS) = Q \quad (3.15)$$

$$\frac{\partial U}{\partial t} = -\frac{1}{V_{i,j}} \left[\vec{F}_{i+1/2,j} \cdot \vec{S}_{i+1/2,j} + \vec{F}_{i-1/2,j} \cdot \vec{S}_{i-1/2,j} + \vec{G}_{i,j+1/2} \cdot \vec{S}_{i,j+1/2} + \vec{G}_{i,j-1/2} \cdot \vec{S}_{i,j-1/2} \right] + Q, \quad (3.16)$$

where

$$\begin{aligned} \vec{S}_{i+1/2,j} &= (y_{i+1,j+1} - y_{i+1,j}) \hat{i} - (x_{i+1,j+1} - x_{i+1,j}) \hat{j} \\ \vec{S}_{i-1/2,j} &= -(y_{i,j+1} - y_{i,j}) \hat{i} + (x_{i,j+1} - x_{i,j}) \hat{j}, \end{aligned} \quad (3.17)$$

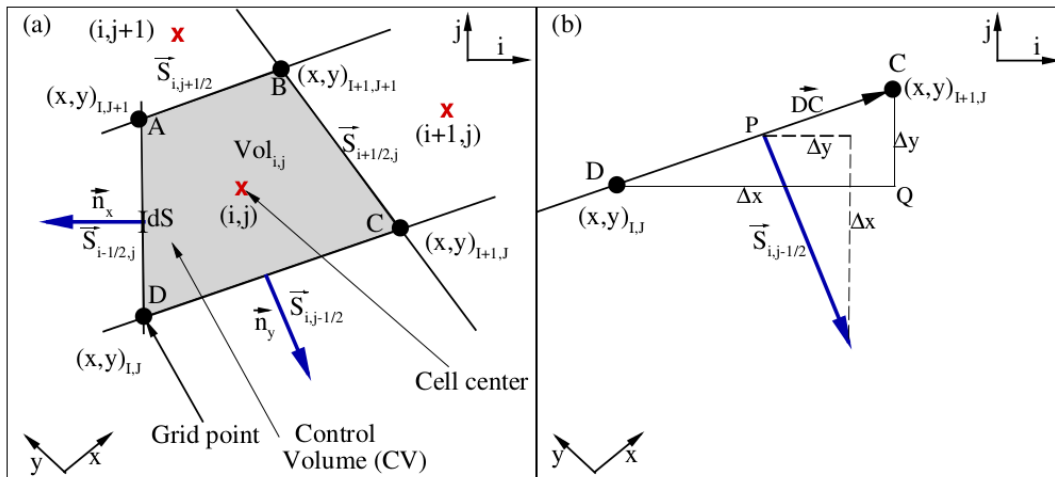


Figure 3.1: (a) Two-dimensional control volume used in finite volume formulation, where the grid points are represented by black solid circles with capital indices of I and J and cell centers are shown by crosses represented by small indices of i and j and (b) Calculation of normal surface vector $\vec{S}_{i,j-1/2}$ [150].

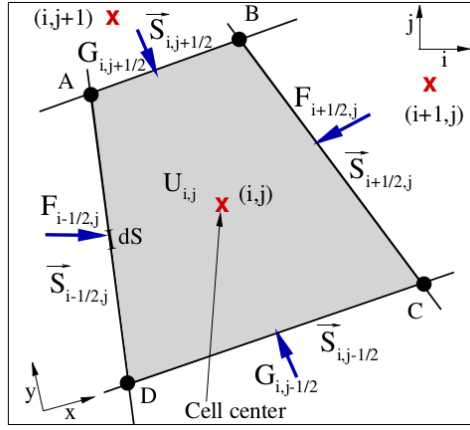


Figure 3.2: The conservative variable $U_{i,j}$ at cell center calculated from the fluxes F and G passing through the cell boundaries in the x - and y -directions [150].

and

$$V_{i,j} = [| (x_{i,j} - x_{i+1,j}) y_{i+1,j+1} + (x_{i+1,j} - x_{i+1,j+1}) y_{i,j} + (x_{i+1,j} - x_{i,j}) y_{i+1,j}| \\ + | (x_{i,j} - x_{i+1,j+1}) y_{i,j+1} + (x_{i+1,j+1} - x_{i,j+1}) y_{i,j} + (x_{i,j+1} - x_{i,j}) y_{i+1,j+1}|] \quad (3.18)$$

Here $\hat{n}$ is unit normal vector to surface area dS as shown in Fig. (3.1) and $\vec{S}_{i+1/2,j}$, $\vec{S}_{i-1/2,j}$ are the surface area vectors and $\hat{i}$, $\hat{j}$ are unit vectors in i and j direction respectively. x and y are the coordinates of the grid points and $V_{i,j}$ is the control volume. The calculation of $\vec{S}_{i,j-1/2}$ is explained in Fig. (3.1b). The vector $\vec{DC} = (x_{i+1,j} - x_{i,j})\hat{i} + (y_{i,j+1} - y_{i,j})\hat{j}$. Hence the normal vector is evaluated as $\vec{S}_{i,j-1/2} = (y_{i,j+1} - y_{i,j})\hat{i} - (x_{i+1,j} - x_{i,j})\hat{j}$. The normal vector $\vec{S}_{i,j+1/2}$ is calculated in a similar way. The numerical methodology is the same approach used in Ref.[150].

3.2.2 Numerical method

The numerical method uses a second order modified (low dissipation) form of Steger-Warming flux vector splitting for evaluating the convective fluxes. This method reduces the numerical dissipation which is very useful for high speed flows with strong shock waves and viscous-inviscid interactions with boundary layers and free shear layers. As a result, very thin and well defined shock waves are captured over minimal grid cells. The viscous fluxes and the turbulent source terms are calculated using second-order accurate central difference method. The code is implemented in parallel using the Message Passing Interface. For the two-dimensional simulations, an implicit Data-Parallel Line Relaxation (DPLR) method is used whereas for the three-dimensional simulations, an implicit full matrix Data-Parallel Lower-Upper Relaxation method [151] is used to integrate the equations in time until steady state is reached.

The high-speed flow configurations considered in this work include different high gradient regions such as wall boundary layers, free shear-layers, separation bubble, shock-waves, expansion waves, boundary-layer flow separation point, and reattachment point. It is important to accurately capture all of these physical features in the flowfield using appropriate CFD techniques. For the steady-state solution of high speed flows, the hyperbolic and elliptic nature of the equations dominate the flow. For high speed steady flows, the convective terms dominate in the inviscid regions and the flow is hyperbolic in nature in space. For boundary-layer flows and separated flows, the diffusive terms dominate. In a separation bubble, the flow is diffusive in nature and hence the information can flow in all the directions and the flow possesses elliptic nature in this confined region. The central difference scheme can be used to discretize the viscous/diffusive terms.

Steger-Warming method

The Steger-Warming method is used for high-speed inviscid flows to capture shock waves by taking advantage of the hyperbolic nature of the Euler equations. The essential requirement for hyperbolic flow is that the eigenvalues should be real, distinct, and positive. The upwind schemes use the local wave propagation information, and the eigenvalues provide the direction of the information in the flowfield. Spatial differencing is done on the mesh points, towards the side of which the information flows. To demonstrate the Steger - Warming method, we take the case of one-dimensional Euler equations for simplicity. The same methodology can be extended to the two- and three-dimensional flows.

$$U = \begin{bmatrix} \bar{\rho} \\ \bar{\rho} \tilde{u} \\ E \end{bmatrix} = \begin{bmatrix} U_1 \\ U_2 \\ U_3 \end{bmatrix}, \quad (3.19)$$

$$F_1 = \begin{bmatrix} \bar{\rho} \tilde{u} \\ (\bar{\rho} \tilde{u}^2 + \bar{p}) \\ (E \tilde{u} + \bar{p} \tilde{u}) \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \\ F_3 \end{bmatrix}, \quad (3.20)$$

where U is the conserved variables vector, $E = \bar{\rho} e + \frac{1}{2} \bar{\rho} \tilde{u}^2$ is the total energy and $e = c_v \bar{T}$ is the internal energy. The flux vector is function of the conservative variable vector and is evaluated as,

$$[F] = [A] [U] \quad (3.21)$$

The Jacobian matrix, A is calculated by taking the partial derivative of the flux-vector with respect to the vector of conserved variables.

$$A = \begin{bmatrix} \frac{\partial F_1}{\partial U_1} & \frac{\partial F_1}{\partial U_2} & \frac{\partial F_1}{\partial U_3} \\ \frac{\partial F_2}{\partial U_1} & \frac{\partial F_2}{\partial U_2} & \frac{\partial F_2}{\partial U_3} \\ \frac{\partial F_3}{\partial U_1} & \frac{\partial F_3}{\partial U_2} & \frac{\partial F_3}{\partial U_3} \end{bmatrix} = \begin{bmatrix} 0 & 1 & 0 \\ (\gamma - 3)\tilde{u}^2/2 & (3 - \gamma)\tilde{u} & (\gamma - 1) \\ (\gamma - 1)\tilde{u}^3 - \gamma e\tilde{u}/\bar{\rho} & \gamma e/\bar{\rho} - 3(\gamma - 1)\tilde{u}^2/2 & \gamma\tilde{u} \end{bmatrix} \quad (3.22)$$

With this Jacobian matrix we write the characteristic equation to find the eigen values as

$$\det[A - \lambda I] = 0, \quad (3.23)$$

where I is the identity matrix. The three eigen values of the above matrix are

$$\Lambda = \begin{bmatrix} \tilde{u} & 0 & 0 \\ 0 & \tilde{u} + a & 0 \\ 0 & 0 & \tilde{u} - a \end{bmatrix} \quad (3.24)$$

Here $\tilde{u}$ is speed of the flow and a corresponds to the speed of the sound. The corresponding left eigen vector matrix and right eigen vector matrix is evaluated as T^+ and T^- . For each eigen value the corresponding eigen vectors are calculated as,

$$[A] \begin{bmatrix} r_1^1 r_1^2 r_1^3 \end{bmatrix} = \lambda_1 \begin{bmatrix} r_1^1 r_1^2 r_1^3 \end{bmatrix}^T \quad (3.25)$$

$$[A] \begin{bmatrix} r_2^1 r_2^2 r_2^3 \end{bmatrix} = \lambda_2 \begin{bmatrix} r_2^1 r_2^2 r_2^3 \end{bmatrix}^T \quad (3.26)$$

$$[A] \begin{bmatrix} r_3^1 r_3^2 r_3^3 \end{bmatrix} = \lambda_3 \begin{bmatrix} r_3^1 r_3^2 r_3^3 \end{bmatrix}^T \quad (3.27)$$

Each of the these single column 1×3 matrix of eigen vectors are arranged in three columns of a 3×3 square matrix to form a right eigen vector matrix T as given by Eqs. 3.28.

$$[T] = \begin{bmatrix} |r_k^p| |r_k^p| |r_k^p| \end{bmatrix} \quad (3.28)$$

Here r is a 1×3 single column eigen vector with $p = (1, 2, 3)$ and $k = (1, 2, 3)$. $|r_k^p|$ corresponds to the right eigen vector single column matrix formed by calculating the three eigen vectors in Eqs. 3.27, for corresponding eigenvalues. The left eigen vector matrix is calculated by taking the inverse matrix of the right vector matrix. Using T^+ and T^- , we calculate two Jacobian matrices containing positive and negative eigenvalues,

such that the diagonal matrix Λ_+ contains only positive eigenvalues and a diagonal matrix Λ_- has only negative eigenvalues.

$$A = A_+ + A_- \quad (3.29)$$

$$\lambda_k = \lambda_{k+} + \lambda_{k-} \quad (3.30)$$

$$\lambda_{k+} = \frac{\lambda_k + |\lambda_k|}{2} \quad (3.31)$$

$$\lambda_{k-} = \frac{\lambda_k - |\lambda_k|}{2} \quad (3.32)$$

$$\Lambda = \Lambda_+ + \Lambda_- \quad (3.33)$$

$$\Lambda_+ = \text{diag}\left(\frac{\lambda_k + |\lambda_k|}{2}, \frac{\lambda_k + |\lambda_k|}{2}, \frac{\lambda_k + |\lambda_k|}{2}\right) \quad (3.34)$$

$$\Lambda_- = \text{diag}\left(\frac{\lambda_k - |\lambda_k|}{2}, \frac{\lambda_k - |\lambda_k|}{2}, \frac{\lambda_k - |\lambda_k|}{2}\right) \quad (3.35)$$

$$A_+ = T^+ \Lambda_+ T^-; \quad A_- = T^+ \Lambda_- T^- \quad (3.36)$$

The information in the flowfield can come from an upstream or downstream direction. Depending on it, the discretization is done to the fluxes. In a control volume, as shown in Fig. (3.3), the fluxes are calculated at the cell faces and the values of conservative variables at the cell center. According to the Jacobians' sign as positive and negative parts, the fluxes are split into

$$F = F_+ + F_- \quad (3.37)$$

$$F_+ = F_+ U \quad F_- = F_- U \quad (3.38)$$

The above fluxes are evaluated for subsonic flows. For supersonic flows, the information moves in a downstream direction, and no information influences the upstream direction flowfield. Hence for supersonic flows $F_+ = F$ and $F_- = 0$.

The first order fluxes are given as

$$F_{+i} = A_{+i} U_i, \quad (3.39)$$

$$F_{-i+1} = A_{-i+1} U_{i+1}. \quad (3.40)$$

For the hyperbolic nature of the Euler equations, the eigenvalues are real and distinct. In the Steger-Warming method, the modification is made to the eigenvalues when they become either zero or one. This can happen when the flow becomes sonic for local Mach = 1 or stagnant for $M = 0$ in some regions. One of the examples in the regions at blunt-nosed of the vehicle. Therefore, the eigenvalues are corrected ($\tilde{\lambda}$) with a small value of a constant, $\epsilon = 0.3$.

$$\tilde{\lambda}_{\pm} = \lambda_{\pm} \pm \sqrt{\lambda_{\pm}^2 + \epsilon^2 a^2} \quad (3.41)$$

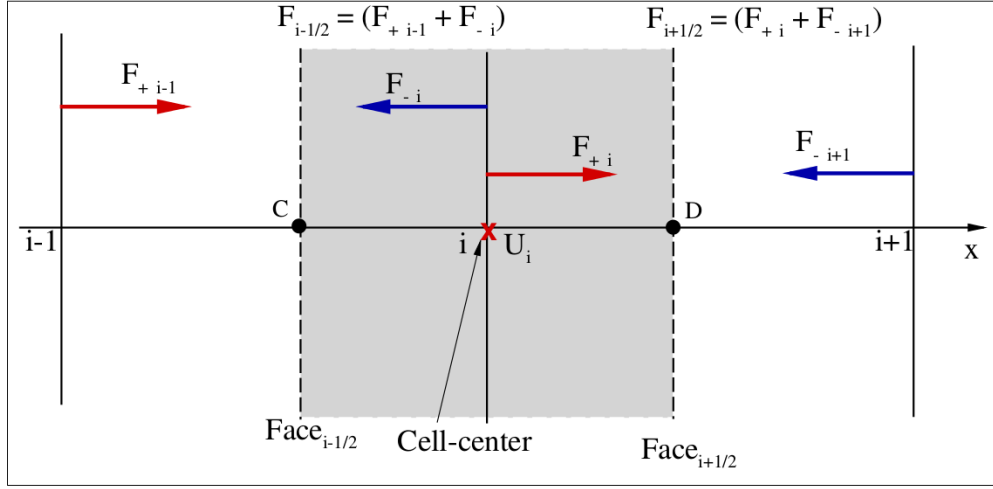


Figure 3.3: Physical interpretation for evaluation of conservative variable at cell-center, using fluxes at cell-interfaces, for the first-order method [150].

The high-quality solutions of undisturbed supersonic boundary-layers are obtained by setting $\epsilon = 0$, and the original eigenvalues are restored from this modified eigenvalue.

Modified Steger-Warming method

The Steger-Warming method, when applied to the viscous flows, results in large numerical dissipation [152] across the boundary-layer. Also, it creates a fictitious pressure gradient, which causes unrealistic convection within the layer. It has similar problems across free shear-layers. MacCormack and Candler [152] modified the Steger-Warming method to reduce the excessive numerical mixing in the boundary layer and free shear-layer flows. In the Steger-Warming method, the evaluation of Jacobians at the cell centers results in excessive numerical dissipation in the boundary-layer. In the modified Steger-Warming method [152] the Jacobians are modified and are calculated at cell-interfaces (see Fig. (3.3)), and from these values, the first order fluxes were calculated as given by Eq. 3.42

$$\begin{aligned}
 F_{+i} &= A_{+i+1/2} U_i \\
 F_{-i+1} &= A_{-i+1/2} U_{i+1} \\
 A_{\pm i+1/2} &= A_{\pm} \left[\frac{1}{2} (U_i + U_{i+1}) \right]
 \end{aligned} \tag{3.42}$$

Second order Steger-Warming method

The first-order upwind schemes are less accurate as they contain less number of points in a stencil. They are more dissipative and do not give any oscillations across the high

gradient regions, such as shock waves. The second-order central differencing schemes are more accurate than first-order upwind schemes and are less dissipative, but they are unstable across the shock waves. The second-order upwind schemes possess less numerical dissipation than the first-order upwind schemes and are more accurate because of more points in the cell face. By increasing the number of points minute details of flow, physics can be accurately captured by the numerical method.

$$\begin{aligned} F_{+i} &= A_{+i} U_{i+1/2}^L \\ F_{-i+1} &= A_{-i+1} U_{i+1/2}^R \end{aligned} \quad (3.43)$$

$$\begin{aligned} U_{i+1/2}^L &= \frac{3}{2} U_i - \frac{1}{2} U_{i-1} \\ U_{i+1/2}^R &= \frac{3}{2} U_{i+1} - \frac{1}{2} U_{i+2} \end{aligned} \quad (3.44)$$

Second order modified Steger-Warming method

$$\begin{aligned} F_{+i} &= A_{+i+1/2} U_{i+1/2}^L \\ F_{-i+1} &= A_{-i+1/2} U_{i+1/2}^R \end{aligned} \quad (3.45)$$

$$\begin{aligned} U_{i+1/2}^L &= \frac{3}{2} U_i - \frac{1}{2} U_{i-1} \\ U_{i+1/2}^R &= \frac{3}{2} U_{i+1} - \frac{1}{2} U_{i+2} \end{aligned} \quad (3.46)$$

Here U_L and U_R correspond to the left and right states. Fig. (3.4) gives the physical interpretation for evaluating the fluxes using a second-order scheme. The fluxes are extrapolated to the interfaces defining the flux equal to the numerical scheme. Here, the flux is evaluated by summing up the first-order and second-order fluxes from the left and right states of the cell interface $i + 1/2$, as shown in Fig. (3.4). The second-order schemes give spurious non-physical oscillations across high gradients regions, such as shockwaves. Flux limiters are used to avoid these spurious oscillations and to limit the spatial derivatives to physically realistic values. Moreover, the modified Steger-Warming method gives less dissipation in the boundary-layers and result in oscillations in the high gradient regions such as shockwaves. In contrast, the standard Steger-Warming gives more numerical dissipation in the boundary-layers and is stable across the shockwaves. To take advantage of both the methods, a hybrid scheme is used to evaluate the fluxes, and the pressure limiter is used to switch from a modified Steger-Warming scheme to the standard Steger-Warming scheme across the shockwaves. When the pressure gradient is more than 50% across the shockwaves, the scheme is reverted to the Steger-Warming method of first-order, and when the pressure gradient is less, the scheme is reverted to modified Steger-Warming method in the boundary-layer regions. The pressure jump across high

gradients is detected with the use of pressure limiter. For smooth gradient regions, these flux limiters are not active.

3.2.3 Boundary condition formulation

Computational domain with different boundary conditions for supersonic flow over flat-plate is shown in Fig. (3.5). Here, $mx \times my$ are the total number of cells, where mx and my are the number of cells in flow-parallel direction and flow-normal direction, respectively. Dashed-lines represent the dummy cells, and circles mark their cell centers. Solid-lines represent the inner cells, and cross marks their cell centers. Dummy cells serve to calculate the fluxes at the cell faces of inner cells along the boundaries, as shown in Fig. (3.5). Dummy cells are used to impose boundary conditions. Dummy cells are assigned appropriate values to obtain the required flow condition at the boundaries. We refer to the dummy cell as the 1st cell and the interior cell adjacent to the boundary as 2nd cell.

The viscous boundary condition (no-slip condition) is achieved by setting an opposite sign condition to the velocity value taken from the 2nd cell, so the wall velocity to be zero. For isothermal boundary condition, the temperature at the 2nd cell is extrapolated to the 1st cell, such that the wall will have a specified and constant temperature. For the inviscid boundary condition, we calculate the wall-normal velocity component at the 2nd cell and put the value at the dummy cell so that this wall-normal component becomes zero. The symmetry boundary condition is similar with an added clause that the gradients of all the variables should be zero. In Fig. (3.5), the extrapolation boundary condition at the

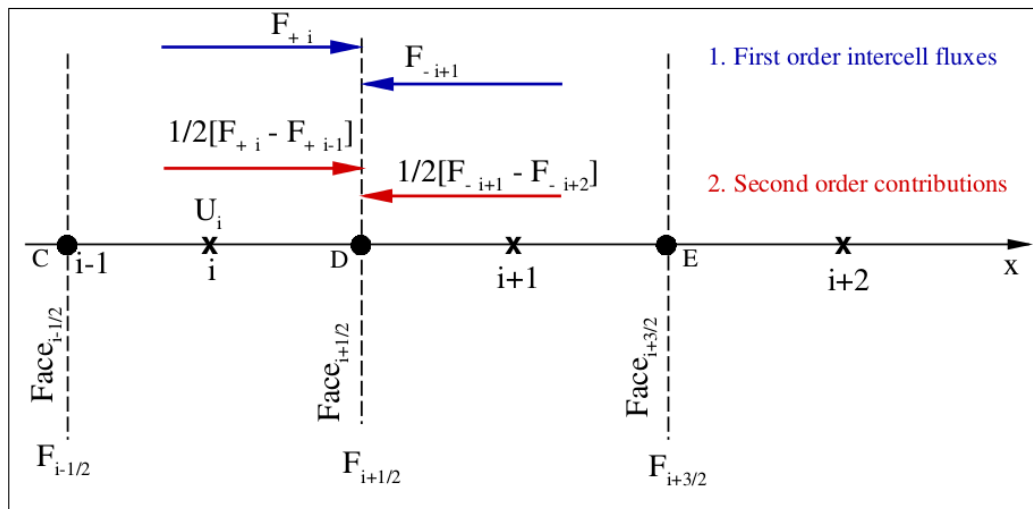


Figure 3.4: Physical interpretation for evaluation of fluxes for the second-order method [150].

exit plane means that the mass flux should be balanced, and the diffusion fluxes for all the variables should be zero. By which the dummy cells do not affect the interior flowfield. For the inlet boundary, the dummy cell values are always set to freestream values.

The free stream and wall boundary conditions for $k - \omega$ turbulence model are stated below with subscript ∞ and w representing the freestream and wall conditions: $\omega_\infty = (10\tilde{u}_\infty)/L$ and $k_\infty = 0.01\nu_\infty\omega_\infty$ and $\omega \rightarrow 6\nu_w/(\beta y^2)$ as $y \rightarrow 0$; $\omega_w = 106\nu_w/(\beta_1\Delta y_1^2)$ as $y \rightarrow 0$; for $\Delta y_1^+ < 3$ (for smooth wall), where Δy_1 is the normal distance to the next point nearest from the wall, $\beta_1 = 3/40$ and L is the length of the domain.

Inlet

The free-stream conditions are assigned at $i = 1$ with j varying from 1 to my .

$$u(1, j) = V_\infty, \quad (3.47)$$

$$\rho(1, j) = \rho_\infty, \quad (3.48)$$

$$v(1, j) = 0, \quad (3.49)$$

$$T(1, j) = T_\infty. \quad (3.50)$$

Top

The free-stream conditions are assigned at top boundary at $j = my$ with i varying from 2 to $(mx-1)$.

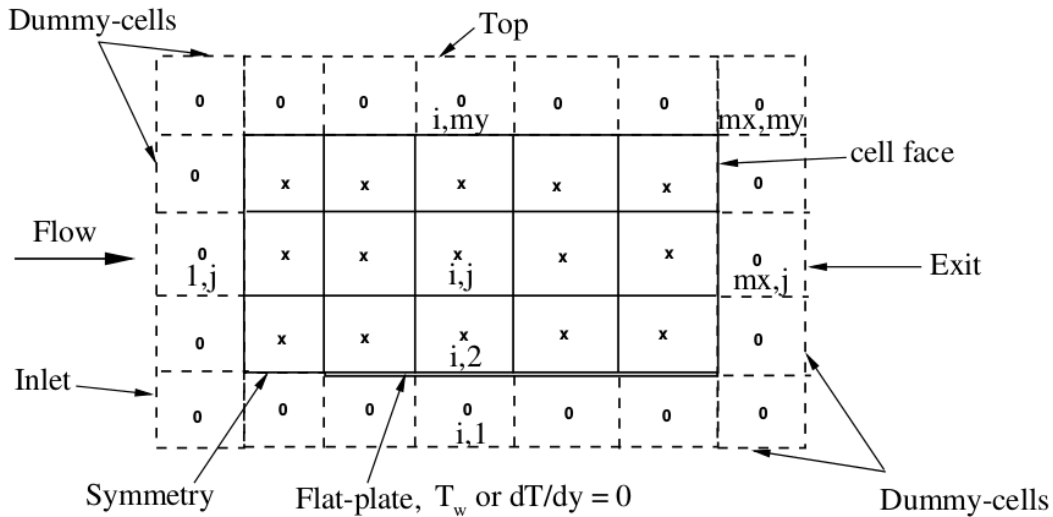


Figure 3.5: Computational domain along with boundary conditions for a flat plate configuration.

Symmetry

The symmetry conditions are assigned at $j = 1$ and $i = 2$. The variables are mirrored over symmetry boundary.

$$u(2, 1) = u(2, 2), \quad (3.51)$$

$$v(2, 1) = -v(2, 2), \quad (3.52)$$

$$\rho(2, 1) = \rho(2, 2), \quad (3.53)$$

$$T(2, 1) = T(2, 2). \quad (3.54)$$

Wall

The physical boundary conditions applied at stationary flat-plate for the viscous flow are,

$$\vec{v} = 0, \quad \text{no slip condition} \quad (3.55)$$

$$\partial P / \partial n, \quad \text{constant pressure boundary layer} \quad (3.56)$$

$$T_w = \text{constant}, \quad \text{isothermal wall} \quad (3.57)$$

$$\partial T / \partial n, \quad \text{adiabatic wall} \quad (3.58)$$

Here, $\vec{v}$ is the velocity vector with components u and v and n is the normal direction to wall. The above conditions are applied numerically at $j = 1$ with i varying from 3 to mx . The density is calculated using the equation of state.

$$u(i, 1) = -u(i, 2), \quad (3.59)$$

$$v(i, 1) = -v(i, 2), \quad (3.60)$$

$$\rho(i, 1) = \rho(i, 2)T(i, 2)/T(i, 1), \quad (3.61)$$

$$T(i, 1) = 2T_w - T(i, 2), \quad (3.62)$$

$$T(i, 1) = T(i, 2) \quad (3.63)$$

Exit

At the exit, the supersonic extrapolation condition is applied where the computed flow variable values are extrapolated from inner cells with $i = mx$ and j varying from 2 to my .

$$u(mx, j) = u(mx - 1, j), \quad (3.64)$$

$$v(mx, j) = v(mx - 1, j), \quad (3.65)$$

$$\rho(mx, j) = \rho(mx - 1, j), \quad (3.66)$$

$$T(mx, j) = T(mx - 1, j) \quad (3.67)$$

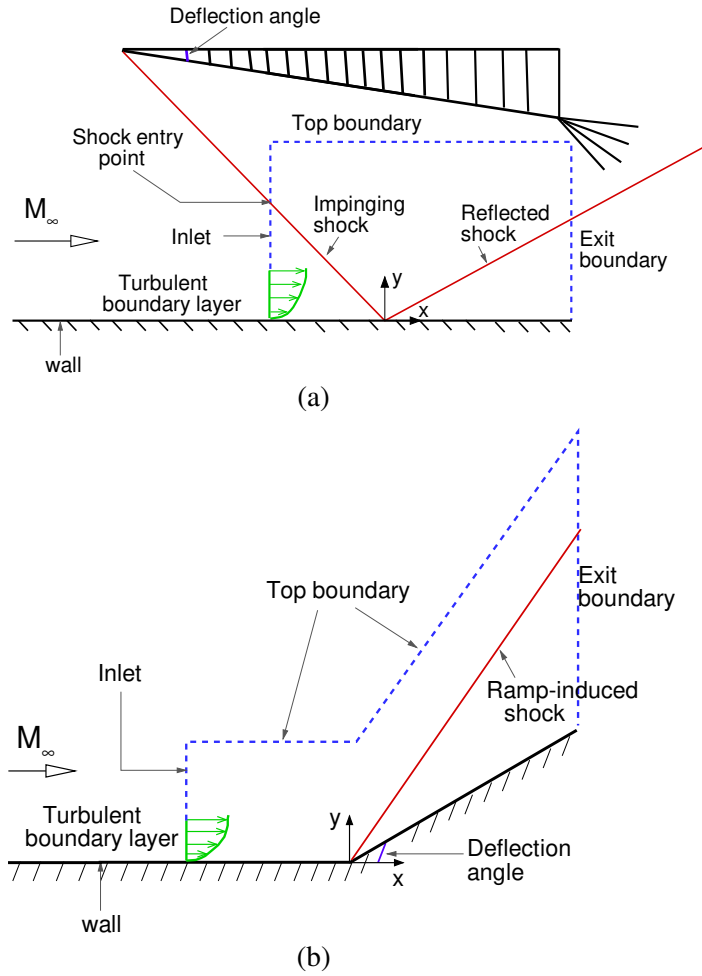


Figure 3.6: Schematic of (a) incident oblique shock wave/turbulent boundary layer interaction on a flat plate and (b) two-dimensional compression corner, with the computational domain marked by dashed line for the two configurations.

3.3 UFAST impinging shock simulations

The experiment for this case was carried out in the Mach 2.3 supersonic wind tunnel at the Institut Universitaire des Systemes Thermiques Industriels (IUSTI) in Marseille, France [153]. The experimental data was part of the European Union SBLI research project referred to as UFAST [51]. The experiment utilizes an 8° shock generator and the incident shock would impact at $x = 336$ mm from the flat plate leading edge in the absence of a boundary layer. Mean flow profiles are provided at $x = 260$ mm from the plate leading edge, where the boundary layer thickness and displacement thickness are found to be 11 mm and 3.535 mm, respectively.

Figure 3.6(a) shows the schematic of experimental set-up and the computational domain is shown using dashed lines. The freestream conditions used for the simulations are given in Table 3.1. The SBLI simulations start at $x = 260$ mm from the plate leading

Table 3.1: Freestream conditions for the SBLI test cases as reported in the experiments.

	Deflection angle	M_∞	ρ_∞ (kg/m ³)	T_∞ (K)	T_w (K)	Re_∞/m $\times 10^6$	Reference
UFAST	8°	2.3	0.0966	144.6	–	5.4	[153]
Schulein	6°	5	0.2043	68.3	300	37	[93]
SBLI	10°	5	0.2043	68.3	300	37	[93]
	14°	5	0.2043	68.3	300	37	[93]
	27°	8.2	0.5090	71.1	296	147	[154]
Holden	30°	8.3	0.5060	68.9	296	149	[154]
SBLI	33°	8.1	0.4910	71.7	297	139	[154]
	36°	8.2	0.4990	70	298	145	[154]

edge. Inlet conditions for the mean and turbulence variables are obtained from separate flat plate simulations and they correspond to the displacement thickness of 3.535 mm. The inlet profile for the mean flow variables is modified at the shock entry point to post-shock conditions calculated analytically (see Fig. 3.6(a)). For the turbulence model equations, k and ω values at the boundary layer edge in the inlet profile are prescribed as the post-shock conditions. The solid wall is considered to be adiabatic and a no-slip wall boundary condition is used. An extrapolation condition is used at the top and exit boundaries.

A structured Cartesian grid with 200×270 finite volume cells is used for the SBLI simulations. The grid size is based on a rigorous grid convergence study described in Appendix F. Grid points are clustered near the wall with a minimum spacing of 1×10^{-6} m, which is equivalent to 0.1 wall units or less along the flat plate. Clustering is also done in the interaction region and the smallest cell size used is 3×10^{-4} m. An exponential stretching is used in the wall-normal direction as well as in the upstream and downstream directions of the interaction location.

3.4 Schulein's impinging shock simulations

The test case consists of a Mach 5 flow with oblique shock impinging on a turbulent boundary layer developed over a flat plate. The experiments [93] were done over a 500

mm long flat plate, and the inviscid shock was adjusted to impinge at a fixed location of 350 mm from the leading edge of the flat plate (Fig. 3.6(a)). The experiments provide mean flow profiles at $x = 266$ mm from the plate leading edge, where the boundary layer thickness, displacement thickness, and momentum thickness are measured as 3.81 mm, 1.576 mm, and 0.157 mm, respectively. The experimental uncertainties are quoted as approximately 2% for pressure and 4% to 10% for skin friction. The experimental data for comparison is taken from [96].

The freestream conditions for these cases are given in Table 3.1 and the computational domain is shown with dashed lines in Fig. 3.6(a). The SBLI computations begin at $x = 266$ mm from the plate leading edge. Inlet conditions are obtained from separate flat plate computations by matching the momentum thickness with the experiment. At the shock entry location (see Fig. 3.6(a)), inlet profile for the mean flow variables is modified and analytical relations are used to prescribe post-shock conditions corresponding to the three different flow deflection angles of 6° , 10° , and 14° . The values of k and ω at the boundary layer edge in the inlet profile are directly used as the post-shock conditions for the turbulence variables. At the solid wall, a no-slip isothermal-wall boundary condition is applied and the wall temperature is taken to be 300 K. Mean and turbulence variables are directly extrapolated from the interior cells at the top and exit boundaries.

A structured Cartesian grid consisting of 300×400 cells is used for the SBLI simulations. The grid is exponentially stretched away from the wall and uses a near-wall spacing of 1×10^{-6} m that is equivalent to 0.55 wall units or lower along the flat plate. Exponential stretching is also used in the upstream and downstream directions of the interaction location, where the minimum cell size is 1×10^{-4} m. The grid is based on extensive grid convergence studies similar to that shown for the UFAST SBLI case in Appendix F.

3.5 Holden's compression corner simulations

Experiments were performed in the CUBRC shock tunnel facility for a range of ramp angles varying between 27° and 36° with a freestream Mach number of about 8 [154]. The length of the plate before the corner is 995.68 mm and the ramp length is 304.8 mm (see Fig. 3.6(b)). Working fluid is dry air and Table 3.1 gives the freestream conditions for these cases. The flow is characterised by a fully developed turbulent boundary layer ahead of the interaction region. Accuracy in surface pressure measurements was estimated as $\pm 3\%$ and the experimental data for comparison is taken from [96].

The computational domain is shown with dashed lines in Fig. 3.6(b). The flat plate length before the corner is kept to be 0.2 m for the SBLI simulations. Separate flat plate

simulations are performed, and mean and turbulence profiles are extracted at 0.8 m from the plate leading edge. These profiles then serve as inlet conditions for the ramp computations. The highest temperature encountered in these simulations is around 750 K; calorically perfect gas assumption is used in the governing equations. A no-slip isothermal-wall boundary condition is employed at the solid walls. An extrapolation condition for the mean and turbulence variables is used at the top and exit boundaries.

The SBLI simulations for these cases are performed on a body-fitted structured grid with 400×250 cells. The grid uses exponential stretching in the wall-normal direction and the height of first cell next to the wall is 1×10^{-6} m which is equivalent to about 0.5 wall units or lower. The grid is finer near the corner with a minimum cell spacing of 1×10^{-4} m and an exponential stretching is used in the upstream and downstream directions along the wall. A previous study [94] based on different $k-\omega$ models uses identical computational grid in these SBLI cases. The SST modification to eddy viscosity is not expected to alter the grid requirement significantly.

3.6 Backward-facing step simulations

A supersonic Mach 2 flow over the backward facing step (BFS) is considered next for which the experiments were performed by Hartfield et al.[100] and McDaniel et al.[99]. The experimental studies examined the BFS geometry as a part of a scramjet engine. Following are the freestream conditions recorded in the experiments :

$M_\infty = 2$, $T_o = 301\text{K}$, $P_o = 34.8 \text{ kPa}$, $U_\infty = 520 \text{ m/s}$, step height $h = 3.18 \text{ mm}$, $Re_h = 1.033 \times 10^5$, $P_\infty = 0.35 \text{ atm} = 38.6 \text{ kPa}$, $T_\infty = 167 \text{ K}$, $\rho_\infty = 0.726 \text{ kg/m}^3$,

where T_o and P_o are the total temperature and pressure, respectively. Boundary layer velocity profile at a distance of $x/h = -1$ is provided in the experiment and the boundary layer thickness measured at this location is 1.6 mm. Measurements for mean flow properties are well documented in the experiments.

A structured multi-block grid is used for the simulations (see Fig. 3.7) and the grid generation is carried out using *Gridgen ver15* commercial software. Free stream conditions are specified in the inlet whereas no-slip boundary conditions are employed at the bottom, top, and step walls. The variables in the interior domain are directly extrapolated at the exit boundary. A periodic boundary condition is used in the span-wise direction as the reported mean flowfield shows two-dimensional characteristics. The incoming block length before the step corner is $23h$ and the length after the corner is $12h$, where ' h ' is the step height. The top boundary is located at a height of about $6.7h$ from the bottom wall after the step corner. The grid consists of 3 blocks and the total number of cells

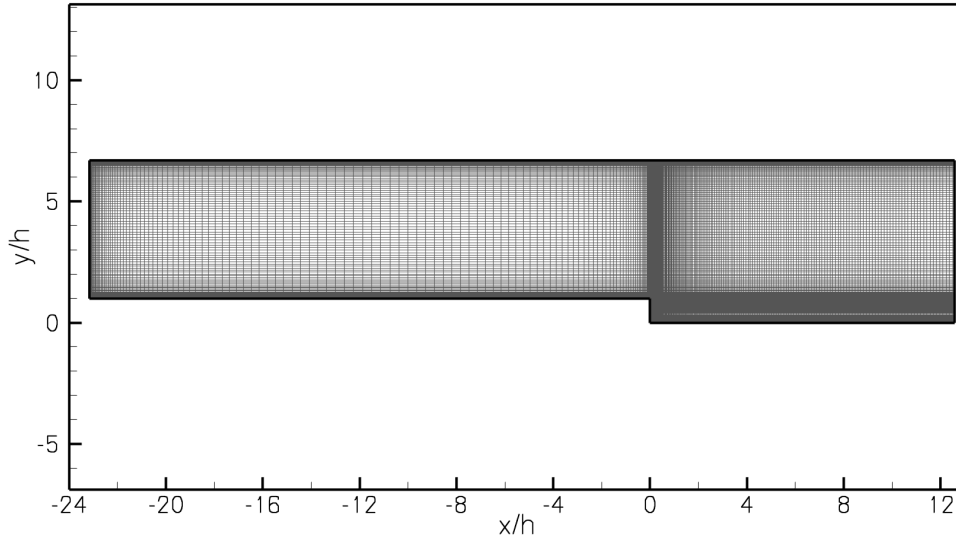


Figure 3.7: Structured 3-block computational Grid for the BFS simulations.

is 220.5×10^3 . The grid is obtained using a detailed grid convergence study using the standard SST model. A similar grid size is expected to work for the SUQ-SST model as well.

The grid is clustered near the bottom, top, and step walls and the incoming bottom wall boundary layer grid is directly extended without adapting it to the free-shear layer development. The minimum cell size near the bottom and top walls in the wall normal direction is 1×10^{-6} m whereas the smallest cell size in the axial direction is near the step corner and it is 1×10^{-5} m. Studies with different grids wherein the grid lines are aligned with the free-shear layer and the reattachment shock wave showed almost no difference with the present grid solutions. The length of incoming boundary layer block is chosen by matching the boundary layer profiles before the corner with the experiment.

3.7 Axisymmetric base flow simulations

A supersonic flow over a cylindrical blunt body is considered and the experiments were performed by Herrin and Dutton [109]. Following are the flow conditions :

$$M_\infty = 2.46, T_o = 294\text{K}, P_o = 515\text{ kPa}, U_\infty = 593.8\text{ m/s}, R = 31.75\text{ mm}, Re/m = 45 \times 10^6, \\ P_\infty = 31.4\text{ kPa}, T_\infty = 145\text{ K}, \rho_\infty = 0.7549\text{ kg/m}^3,$$

where T_o and P_o are the total temperature and pressure, respectively and R is the radius of the solid cylinder. Boundary layer characteristics at a distance of 1 mm before the base are measured in the experiments and the boundary layer thickness, displacement thickness, and momentum thickness are 3.2 mm, 0.96 mm, and 0.26 mm, respectively. The experimental measurements for mean and turbulence quantities are well documented

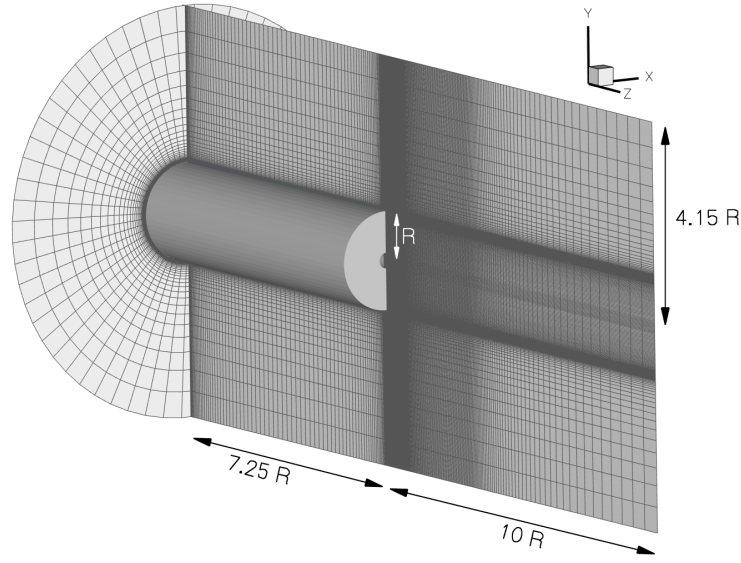


Figure 3.8: Computational domain and the structured 4-block grid used for axisymmetric base flow simulations.

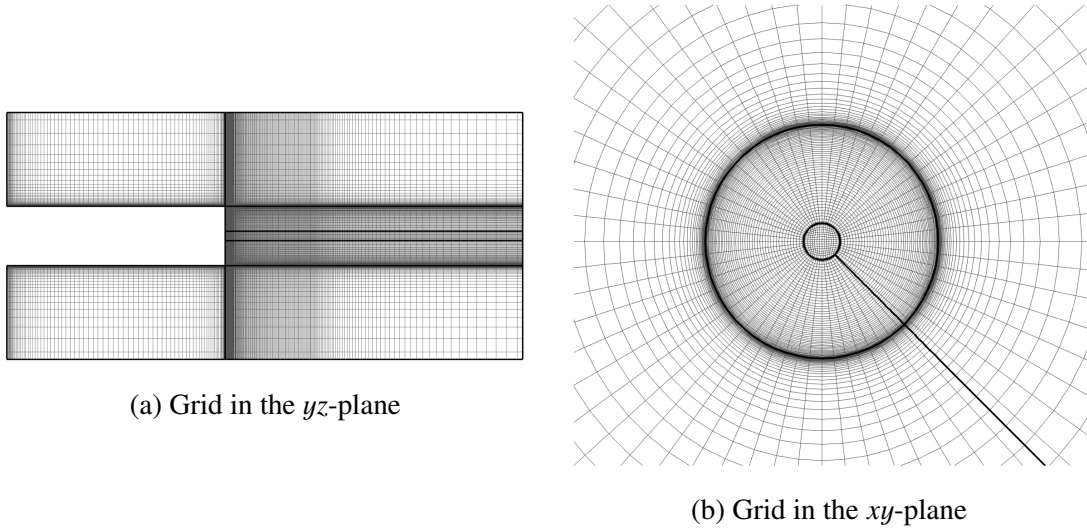


Figure 3.9: Computational grid for the axisymmetric base flow computations as seen in different planes.

and this case has been used for turbulence model validation purposes by a number of researchers.

Figure 3.8 shows the computational domain and the grid used for the computations. Grid sections in the yz and xy planes are shown in Fig. 3.9. The length of incoming boundary layer block is chosen by matching the boundary layer characteristics 1 mm before the base corner with the experiment. The exit boundary is placed at a distance of $10R$ from the base whereas the top boundary is located at $4.15R$ from the base center. Freestream conditions are directly specified at the inlet while a no-slip condition is used

at the cylinder and base walls. At the exit and top boundaries, extrapolation condition is used.

A structured multi-block grid is used for the simulations and the grid is generated using the commercial software *Gridgen ver15*. The grid consists of 4 blocks; block 1 and 2 spans the cylindrical domain after the base while block 3 serves for the development of the incoming boundary layer and block 4 is the parallel extension of block 3. The grid is clustered near the walls and the incoming boundary layer grid is directly extended without adapting it to the free-shear layer development. Studies with different grids wherein the grid lines are aligned with the free-shear layer and the reattachment shock wave showed almost no difference with the present grid solutions. A cell size of 1×10^{-6} m is used near the cylinder wall in the wall-normal direction whereas the size of the cell near the base wall is 1×10^{-5} m. The grid consists of about 1.65 million cells and grid size is based on a detailed grid converge study as presented in the Appendix F.

3.8 Axisymmetric underexpanded jet computations

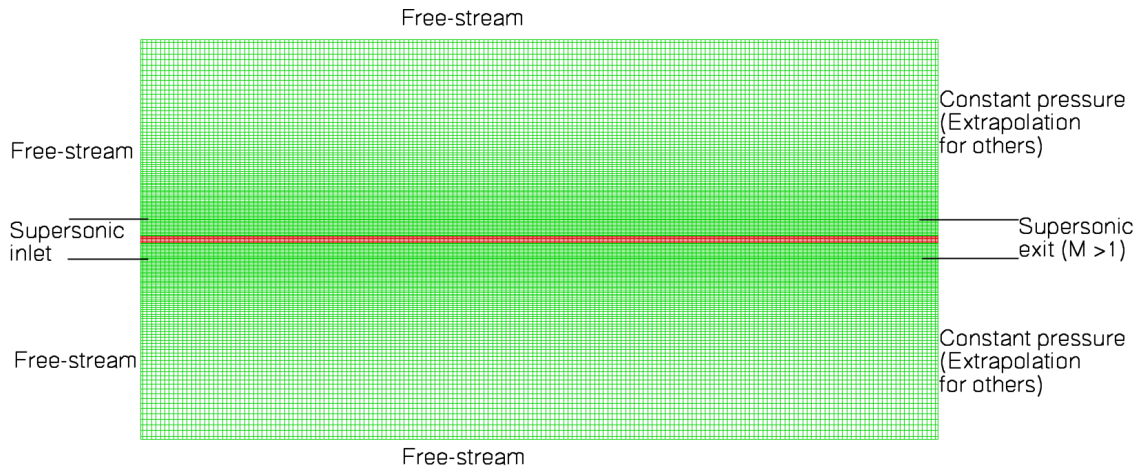
The supersonic underexpanded jet used to evaluate the turbulence models in this work was studied by Seiner and Norum [155, 156]. This jet differs from other supersonic underexpanded jets available in the literature in that the nozzle used had a convergent-divergent shape which gave flat exit mean flow profiles. In the experiments, a Mach 2 cold jet is issued into atmospheric still air at a nozzle pressure ratio (NPR) of 1.45, where NPR is defined as the ratio between nozzle exit jet static pressure to ambient static pressure. A cold jet is considered in the present study instead of a hot jet for simplicity in the computations as ideal gas simulations without considering chemical reactions serve the purpose. Also, the major features of the reverse flow observed in the multi-jet interaction phenomenon (discussed in the next section) are similar for hot or cold jets (without base burning) and this is supported by experimental data [141]. The jet and free stream conditions used in the simulations are given below :

- **Jet conditions at the nozzle exit** : $M_j = 2$, $P_j = 1.446 P_\infty$, $T_j = 0.5556 T_\infty$
- **Free-stream conditions** : $M_\infty = 0.01$, $P_\infty \sim 0.99 \text{ bar} = 0.99 \times 10^5 \text{ Pa}$, $T_\infty = 288 \text{ K}$

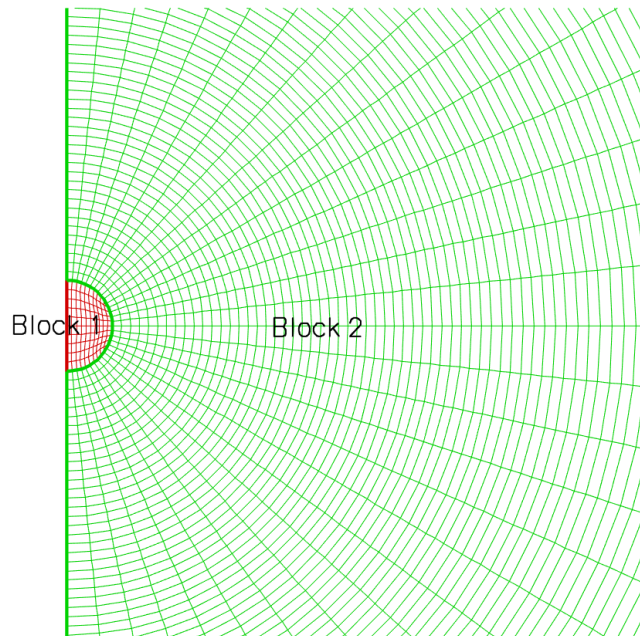
The nozzle exit diameter D is considered to be 50 mm. The flowfield is assumed to be axisymmetric, as per [36, 125–127]. Hence, a semicircular grid is considered for the 3D computations. A structured 2-block grid generated using the *Gridgen ver15* software is used for the simulations, and a small sized first block is used inside the nozzle radius to avoid grid singularity at the center (see Fig. 3.10). The computational domain covers

$20D$ in the axial direction and $5D$ in the radial direction from the jet centerline, and this is based on the previous studies [36]. The grid used for the simulations consists of more than 2.4 million cells, and it is based on a careful grid convergence study. The cells are uniformly spaced in the axial directions whereas a tan hyperbolic stretching is used in the radial direction. The cell size in the axial direction is about 3.75 mm whereas the minimum cell size in the radial direction is about 0.67 mm near the jet centerline.

The supersonic jet conditions are directly specified at the nozzle exit. A very low Mach number of 0.01 is considered for the freestream and other flow properties are di-



(a) Grid in yz plane



(b) Grid in xy plane

Figure 3.10: Structured 2-block grid used for the underexpanded jet simulations along with the boundary conditions used.

rectly specified in the freestream region. Computations with smaller M_∞ values, like 0.001 and 0.0001, give similar results but require very small CFL numbers for convergence. Also, a higher M_∞ value such as 0.1 results in larger shock-cell sizes which are inconsistent with the experiment. Similar trends for freestream Mach number variation on the results are observed in [127]. In the exit boundary, variables are directly extrapolated from the interior for the supersonic region with Mach number greater than 1. On the other hand, for regions with $M < 1$, a constant pressure boundary condition is used and an extrapolation condition is used for the other variables. Computations with characteristic boundary conditions are also explored similar to the previous studies [36]. However, convergence was found to be significantly difficult with such boundary conditions, especially on finer grids.

3.9 Multi-jet interaction simulations

The available wind tunnel data of Musial and Ward [131] is used for the validation purposes. This case has been chosen as it appears to be the only set of experiments where both the base pressure and base heat flux are measured and well documented. The experiments were conducted on a wind tunnel model for several four-nozzle clustered configurations, using bell shaped and conical nozzles. Three different nozzle extension ratios (nozzle extension length from the base to diameter, L/D_e) and different nozzle spacing ratios (nozzle spacing to nozzle exit diameter, D_s/D_e) were considered for a range of nozzle pressure ratio (NPR) and freestream Mach number in the experiments.

The present investigation focuses on the experimental configurations that used bell nozzles with D_s/D_e of 2.18 and a L/D_e ratio of 0.7015 (see Fig. 3.11 and Table 3.2). The simulations are carried out for two NPR conditions; low altitude case with NPR of 990 and a freestream Mach number of 2.5 and high altitude case with NPR of 3760 and a freestream Mach number of 3.5. Here, NPR is defined as the ratio between chamber pressure and ambient static pressure. Perfect gas (thermally and calorically perfect) assumption is used in this study, as previous studies [8, 142] show that reasonable predictions for base pressure and base heat flux can be obtained using these assumptions. From one-dimensional isentropic relations, nozzle exit conditions can be obtained for the given nozzle area ratio ($A_e/A_* = 12$).

The nozzle flow is modelled using the ratio of specific heats $\gamma = 1.15$, as per Mehta et al.[]. Using isentropic relations, we get Mach number at the nozzle exit $M_j = 3.25$, $P_j/P_c = 0.0114$, $T_j/T_c = 0.558$, where the subscripts 'c' and 'j' represent the total conditions and jet static conditions at nozzle exit, respectively. Now, using chamber pressure

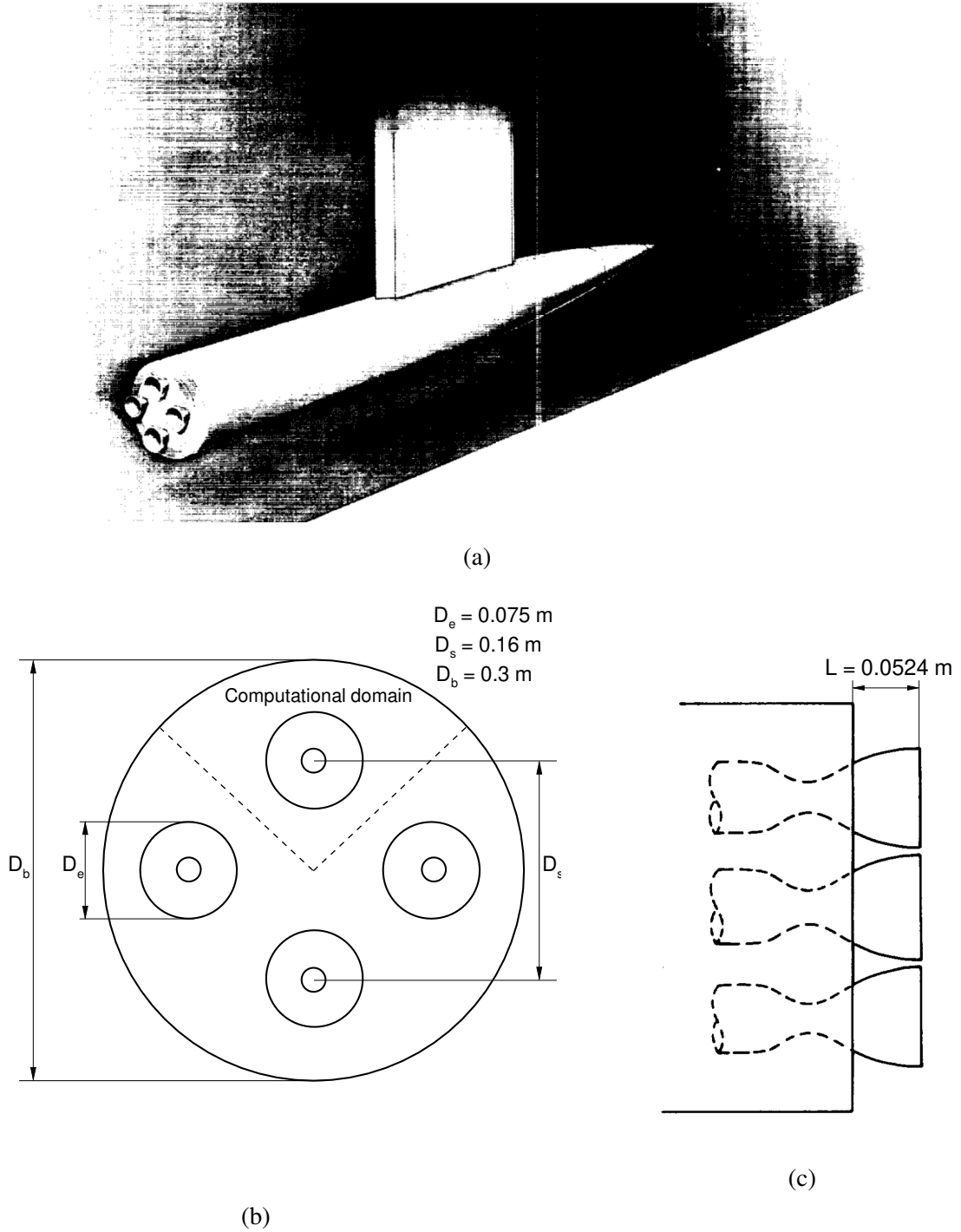


Figure 3.11: (a) Base model used for the experiments conducted by Musial and Ward [131], and (b) Base plate with four nozzles showing the symmetry lines where base pressure and heat flux were measured [131] (c) Geometric configuration showing the nozzle extension considered.

$P_c = 4.137 \times 10^6$ Pa, we get $P_j = 47.162$ kPa, and using chamber temperature $T_c = 3227.778$ K gives $T_j = 1801$ K and $\rho_j = 0.0912$ kg/m³. Assuming, $T_\infty = 300$ K for the two cases as per Zhou et al. [146] and as P_c is given and a constant, we can find

Table 3.2: Geometry and flow parameters for the multi-jet interaction cases.

Geometry parameter	Value
Base diameter, D_b	0.305 m
Nozzle exit diameter, D_e	0.075 m
Nozzle spacing ratio, D_s/D_e	2.18
Nozzle extension ratio, L/D_e	0.7
Nozzle area ratio	12
Flow conditions	Value
Base wall temperature, T_d	338.9 K
Chamber pressure, P_c	4.14 MPa
Chamber temperature, T_c	3227.8 K

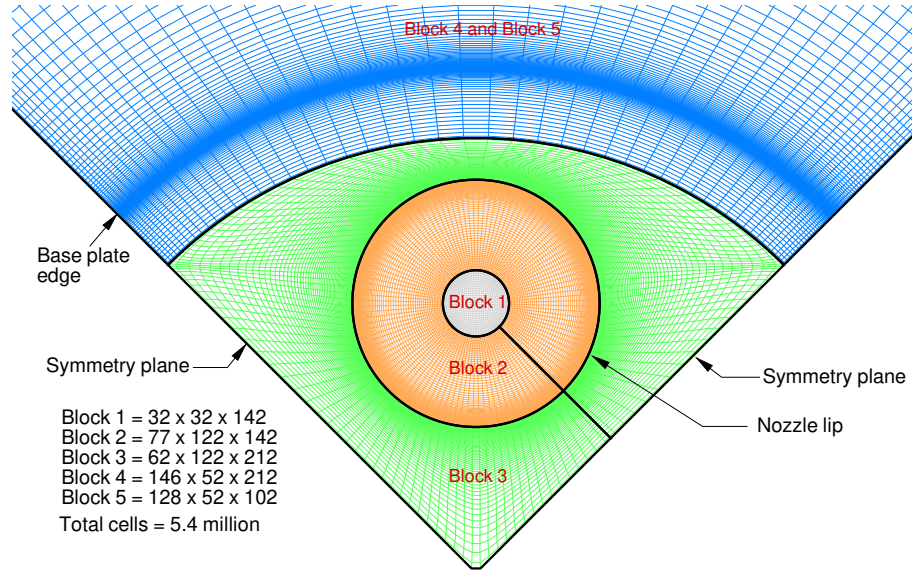
Table 3.3: Chamber pressure to ambient static pressure ratio (P_c/P_∞), Nozzle exit pressure to ambient static pressure ratio (P_j/P_∞), and freestream conditions

S. No.	P_c/P_∞	P_j/P_∞	P_∞ (kPa)	M_∞	$Re_\infty (\times 10^5)$
1	990	11.3	4.2	2.5	5.2
2	3760	42.9	1.1	3.5	1.9

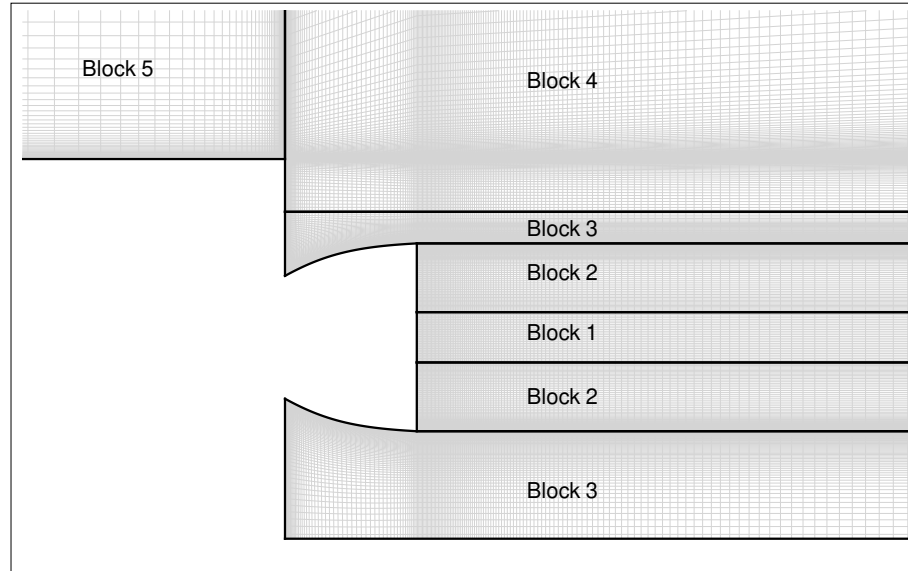
$P_\infty = 4178.78$ Pa for $P_c/P_\infty = 990$ and $P_\infty = 1100.26$ Pa for $P_c/P_\infty = 3760$. Now, freestream density ρ_∞ for the two NPR cases can be obtained using the ideal gas relation, and the values are 0.0485 kg/m^3 for low altitude and 0.0128 kg/m^3 for the high altitude case. The flow conditions for the two test cases are given in Table 3.3. The wall is assumed to be isothermal maintained at a temperature of $T_{wall} = 338.889$ K, as per Musial and Ward [131]. Radiation heat transfer is neglected in the present computations since it is found to be negligible compared to the convective heat transfer for this case [8].

From symmetry considerations, only $1/8^{th}$ sector of the base can be used for the CFD simulations. However, due to issues with grid generation, a quarter grid is used for the simulations (see Figs. 3.11(b) and 3.12). A three-dimensional structured multi-block body fitted grid is generated using the commercial software *Gridgen ver15*. The maximum extension of the domain in the transverse direction is about $3.4D_e$, whereas it extends to about $10D_e$ in the streamwise direction downstream of the base plane and about $8.2D_e$ upstream of the base plane, where $D_e = 0.075$ m is the diameter of the nozzle.

A 5-block structured grid is used for the computations and the grid consists of a total of about 5.4 million interior cells (see Fig. 3.12). A number of grid topologies with different meshing strategies along with different domain structures, such as a grid with two half nozzles instead of a single full nozzle, are tried out in this work, and they gave similar results. Note that a grid convergence study has not been performed for these test cases because of the constraints on the computational resources. However, based on the experience from the grid convergence studies conducted for the other test cases, the grid



(a)



(b)

Figure 3.12: Surface grids showing the computational domain used for the multi-jet simulations : two views

sizes are chosen to be sufficiently finer near the important flow structures like the shock waves, shear layers, recirculation regions, etc. Also, base pressure is found to be relatively insensitive to the grid size than base heat transfer rates for these test cases; see Mehta et al.[8] and Nallasamy et al.[142].

Table 3.4: Table of co-ordinates for Bell-shaped nozzle contour [131].

Table of co-ordinates	
$x \text{ (m)}$	$y \text{ (m)}$
0	0.0108585
0.0127	0.0163068
0.0254	0.023622
0.0381	0.029845
0.0508	0.0338836
0.0635	0.036068
0.079248	0.037338

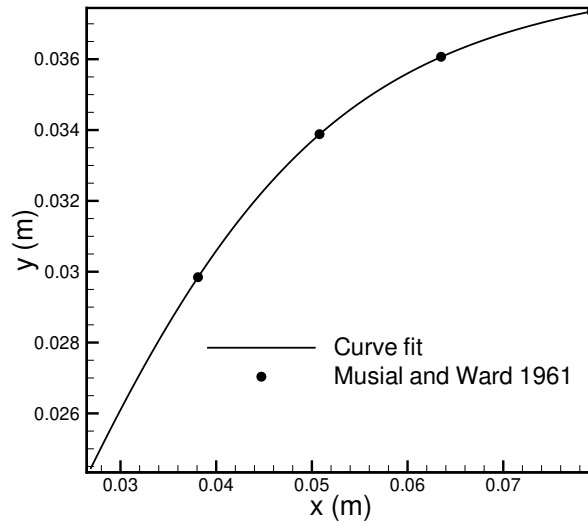


Figure 3.13: Comparison of the curve fit and co-ordinates given by Musial and Ward [131].

Musial and Ward [131] has provided some of the xy co-ordinates of the bell nozzle contour and they are given in Table 3.4. Using those, a curve is fitted as :

$$y = (3.434 \times 10^6)x^7 - (1.094 \times 10^6)x^6 + (1.197 \times 10^5)x^5 - (3.203 \times 10^3)x^4 - (3.046 \times 10^2)x^3 + (18.14)x^2 + (0.2515)x + 0.01086 \quad (3.68)$$

Using the above equation, the nozzle contour is obtained as shown in Fig. 3.13. This curve is then exported in the *Gridgen* software and a 3D bell-shaped nozzle contoured grid is made as shown in Fig. 3.14.

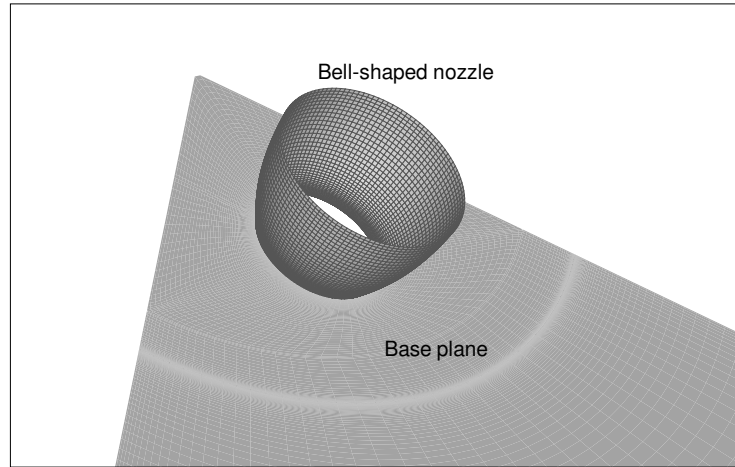


Figure 3.14: Grid showing bell nozzle used for the multi-jet simulations.

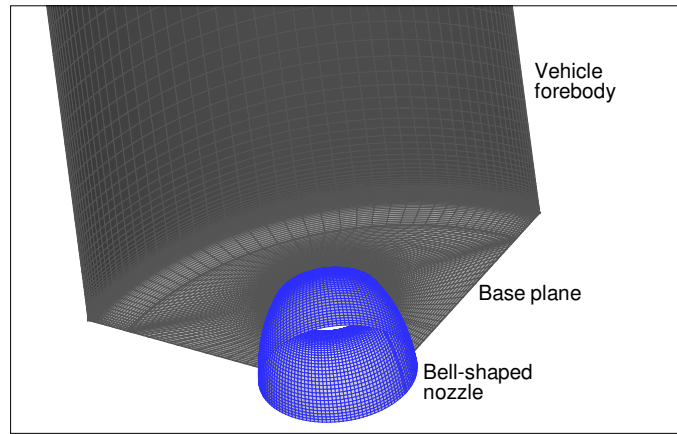


Figure 3.15: Cylindrical body along with the bell-shaped nozzle used for the multi-jet computations.

On the base surface, no slip condition for the velocity and constant wall temperature boundary condition are specified. At the inflow boundary upstream of the base, uniform inflow conditions are prescribed. The simulation methodology of Nallasamy et al.[142] is adopted in this work. The nozzle internal flow is not solved. Instead, the nozzle exit flow conditions are derived from one-dimensional isentropic flow corresponding to the stagnation pressure and temperature, as per earlier discussions. Shear profiles for mean and turbulence variables are prescribed using an exponential function. The flow properties at the top and exit boundary are extrapolated from those of the interior domain. Symmetry boundary conditions are prescribed on the nozzle, plume impingement, and base wall symmetry planes. Fig.3.16 shows the three-dimensional computational domain along with the boundary conditions used in the simulations.

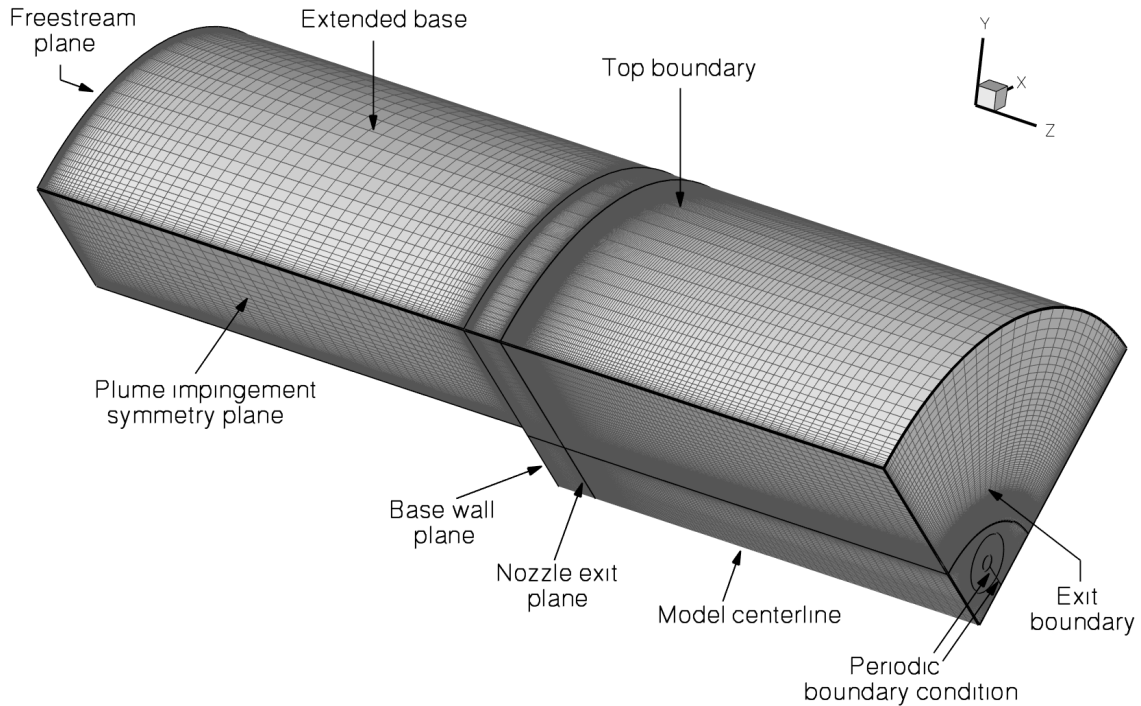


Figure 3.16: 3D view of the multi-block grid used for the multi-jet computations along with the boundary conditions employed.

3.10 Summary

This chapter discusses in detail the simulation methodology adopted for the one-, two-, and three- dimensional RANS simulations carried out in this work. The canonical shock-turbulence interaction cases are computed in a 1D RANS set-up and the governing equations including the initial and boundary conditions are discussed. For the two- and three-dimensional mean flow simulations, a well-validated in-house CFD code is used. It uses a second order modified Steger-Warming flux splitting approach for computing the inviscid fluxes and second-order central differencing for the viscous fluxes and turbulent source terms. An implicit time integration algorithm is used to reach steady-state solutions. The validation test cases cover a wide range of flow conditions from supersonic to hypersonic speeds and attached to fully separated flows that display SBLIs and SSLIs. A number of geometric configurations are considered including compression ramp, impinging/reflected shock, backward facing step, cylindrical blunt body, underexpanded nozzle, and four-nozzle clustered rocket engine. Details of the computational domain and structured grids used for the simulations are also presented.

Chapter 4

Modeling of shock-turbulence interaction

A shock wave is often encountered in high-speed flows and it modulates the turbulence passing through it causing an amplification of turbulence kinetic energy (TKE) [30, 47]. Interestingly, the interaction also has an effect on the shock wave and the upstream turbulent fluctuations cause unsteady shock oscillations. It can be hypothesized that the amount by which shock experiences unsteady motion depends on shock strength and hence the upstream flow Mach number M_1 [61]. As the shock oscillates, additional stresses are expected to arise as a result of its stretching and straining. However, conventional two-equation eddy viscosity turbulence models like k - ϵ , k - ω , and Menter SST k - ω consider a shock wave to be rigid and neglect any oscillations when it interacts with turbulence. This is one of the major reasons of their failure in case of high-speed flows involving shock waves, as pointed out by Sinha et al. [30].

4.1 Physics based shock-unsteadiness modification to k - ϵ model

The standard k -equation [57] for a compressible steady one-dimensional mean flow through a normal shock simplifies to

$$\bar{\rho}\tilde{u}\frac{\partial k}{\partial x} = -\bar{\rho}\widetilde{u_i''u_i''}\frac{\partial \tilde{u}}{\partial x} - \bar{\rho}\epsilon + \overline{p'\theta'} \quad (4.1)$$

where $k = \frac{1}{2}\widetilde{u_i''u_i''}$ is the turbulent kinetic energy, ρ is the density, u is the component of velocity in the stream-wise direction x , p is the pressure and $\theta = \partial u_j / \partial x_j$ is the dilatation. Here, overbar and tilde represent Reynolds and Favre-averaged quantities, and the prime

and double-prime represent the Reynolds and Favre fluctuations respectively. The source terms correspond to the production and dissipation mechanisms, where ϵ is a sum of the solenoidal part ϵ_s and the dilatational part ϵ_c . We neglect the viscous and turbulent diffusion terms, as per [30]. The normal Reynolds stress $\bar{\rho}\widetilde{u''u''}$ is modeled using the Boussinesq assumption as

$$\bar{\rho}\widetilde{u''u''} = -\frac{4}{3}\mu_T\frac{\partial\tilde{u}}{\partial x} + \frac{2}{3}\bar{\rho}k \quad (4.2)$$

where μ_T is the eddy viscosity and is given by $\mu_T = c_\mu\bar{\rho}k^2/\epsilon_s$ and $c_\mu = 0.09$ is a model constant.

Sinha et al. [30] studies the physics of canonical STI to suggest improvements in the k - ϵ model. A transport equation for k is derived in the reference frame of the shock by assuming that the shock undergoes small deviations from its mean position. The transport equation has an extra term that contains the effect of unsteady shock oscillations. Linear analysis shows that this term has a damping effect on the amplification of TKE through the shock. It is modeled with the assumption that shock-unsteadiness is proportional to the turbulent fluctuations in the flow. The value of the constant of proportionality obtained using LIA results is modified such that TKE amplifications across the shock match the limiting behaviour for $M_1 \rightarrow 1$ and $M_1 \rightarrow \infty$. Thus, the modeled k -equation takes the form

$$\bar{\rho}\tilde{u}\frac{\partial k}{\partial x} = -\frac{2}{3}\bar{\rho}k\frac{\partial\tilde{u}}{\partial x} + \frac{2}{3}b'_1\bar{\rho}k\frac{\partial\tilde{u}}{\partial x} - \bar{\rho}\epsilon_s \quad (4.3)$$

where $b'_1 = b_{1,\infty}(1 - e^{1-M_1})$ is called as the shock-unsteadiness damping parameter and $b_{1,\infty} = 0.4$ is its high Mach number limiting value. Notice that the turbulent viscosity is suppressed in the production term so as to minimize the error due to the limitations in the Boussinesq approximation. In the shock region, setting μ_T equal to zero results in the isotropic form of the Reynolds stress, $\bar{\rho}\widetilde{u''u''} = \frac{2}{3}\bar{\rho}k$, which is valid for the shock-upstream flow. In case of the models with realizability constraints [28, 32] and nonlinear eddy-viscosity based formulations [64, 65], the limitations in Boussinesq approximation are taken care of by suppressing the model parameter c_μ in regions of high mean flow gradient. Eq. (4.3) is applicable to one-dimensional shock-turbulence interactions with uniform mean flow on either sides of the shock wave and represents a significant improvement over the existing models. Compressibility corrections [157], in the form of dilatational dissipation and pressure dilatation, do not improve turbulence levels significantly [30]. Further, they deteriorate model predictions in the upstream boundary layer and therefore are not preferred in SBLI flows [158]. Hence, they are not included in Eq. (4.3).

The solenoidal dissipation rate ϵ_s is defined as the product of the mean kinematic viscosity $\bar{\nu}$ and enstrophy, i.e., mean-square vorticity fluctuations $\overline{\omega'_i \omega'_i}$, and hence can be written as $\epsilon_s = \bar{\nu} \overline{\omega'_i \omega'_i}$. Sinha [91] assumes that ϵ_s is the dominant part in the turbulent dissipation rate. A transport equation for the evolution of ϵ_s is derived using linearised Navier-Stokes equations and is modeled as,

$$\bar{\rho} \tilde{u} \frac{\partial \epsilon_s}{\partial x} = -\frac{2}{3} c_{\epsilon 1} \bar{\rho} \epsilon_s \frac{\partial \tilde{u}}{\partial x} - c_{\epsilon 2} \bar{\rho} \frac{\epsilon_s^2}{k} \quad (4.4)$$

where $c_{\epsilon 1} = 1 + 0.21 M_1$ and $c_{\epsilon 2} = 1.2$ are the model parameters. The first term on the right-hand-side brings in the effect of mean compression on the vorticity fluctuations and the effect of temperature rise on the mean kinematic viscosity. It represents the mechanism by which the dissipation rate of TKE gets amplified across the shock. The second term is the dissipation mechanism whose effects are significant only in the shock-free region and can be neglected in the shock wave [30]. It governs the decay rate of turbulence on either sides of the shock wave. Eq. (4.4) is applicable to the canonical case where homogeneous isotropic turbulence passes through a normal shock and is able to predict the amplification of the TKE dissipation rate accurately. Note that the above form of the model constant $c_{\epsilon 1}$ proposed in [91] is close to that in [30] and using any of the forms makes practically no difference to the results obtained.

4.2 A new k - ϵ model

The model parameter b'_1 in Eq. (4.3) and $c_{\epsilon 1}$ in Eq. (4.4) are functions of the upstream Mach number, which makes their numerical implementation non-local on a computational mesh. We attempt to remove this dependency using the following approach.

4.2.1 Turbulence Kinetic Energy

Across the shock, the conservation of mean momentum can be simplified to

$$\bar{\rho} \tilde{u} \frac{\partial \tilde{u}}{\partial x} + \frac{\partial \bar{p}}{\partial x} = 0 \quad (4.5)$$

Rewriting the above equation as,

$$\frac{\partial \tilde{u}}{\partial x} = -\frac{\tilde{u}}{\bar{\rho} \tilde{u}^2} \frac{\partial \bar{p}}{\partial x} \quad (4.6)$$

and assuming that $\bar{\rho} \tilde{u}^2$ scales as $\bar{p}$, we get

$$b'_1 \frac{\partial \tilde{u}}{\partial x} = -c \frac{\tilde{u}}{\bar{p}} \frac{\partial \bar{p}}{\partial x} \quad (4.7)$$

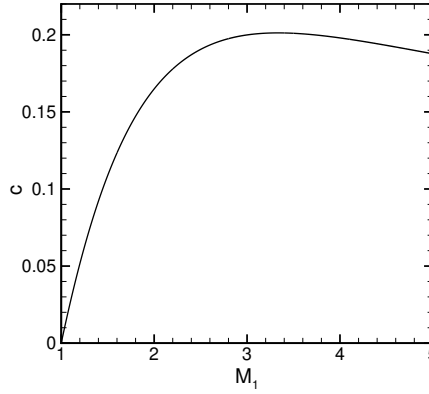


Figure 4.1: Variation of the model parameter c with upstream Mach number.

where c is a new model parameter. The above equation gives an alternative model for the shock-unsteadiness term, and leads to a new form of the k -equation,

$$\bar{\rho}\tilde{u}\frac{\partial k}{\partial x} = -\frac{2}{3}\bar{\rho}k\frac{\partial \tilde{u}}{\partial x} - \frac{2}{3}c\bar{\rho}k\frac{\tilde{u}}{\bar{p}}\frac{\partial \bar{p}}{\partial x} - \bar{\rho}\epsilon_s \quad (4.8)$$

A suitable value of the model constant c is obtained by calibrating it to the shock unsteadiness k - ϵ model [30]. The calibration is done for normal shocks with upstream Mach numbers ranging from 1 to 5.

Integrating Eq. (4.7) across the shock, we get

$$b'_1 \ln\left(\frac{\tilde{u}_2}{\tilde{u}_1}\right) = -c \ln\left(\frac{\bar{p}_2}{\bar{p}_1}\right) \quad (4.9)$$

Note that the subscripts ‘1’ and ‘2’ denote the values of the quantities upstream and downstream of the shock respectively. Using b'_1 and Rankine-Hugoniot jump conditions for $\tilde{u}$ and $\bar{p}$, we obtain the value of c as a function of the shock strength (Fig. 4.1). The model constant varies between 0 and 0.2 for Mach numbers in the range 1 to 5. A positive c and a positive pressure gradient at the shock results in a negative contribution of the corresponding source term in Eq. (4.8). Thus, it brings down the amplification of TKE in the shock region. This is physically consistent with the function of the shock unsteadiness modification of Sinha et al. [30]. A value of $c = 0.194$ is recommended as it gives the best possible match with the shock unsteadiness k - ϵ model. Neglecting the TKE dissipation rate in the shock region and integrating Eq. (4.8) across the shock, we get

$$\left(\frac{k_2}{k_1}\right) = \left(\frac{\tilde{u}_1}{\tilde{u}_2}\right)^{2/3} \left(\frac{\bar{p}_2}{\bar{p}_1}\right)^{-2c/3} \quad (4.10)$$

Similarly, neglecting the turbulent dissipation from Eq. (4.3) and integrating it across the shock yields,

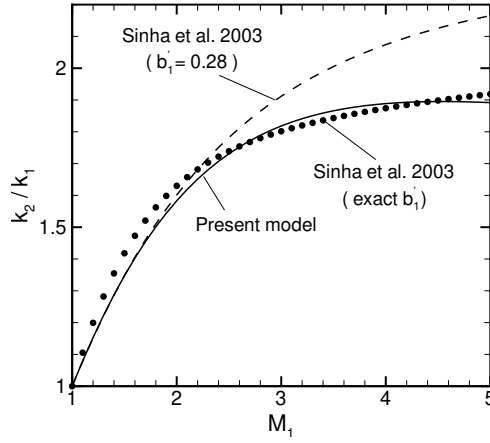


Figure 4.2: TKE amplification for varying upstream Mach number as obtained for the present model with $c = 0.194$, shock-unsteadiness model [30] with exact b'_1 and $b'_1 = 0.28$.

Table 4.1: Error (in %) in TKE amplification for the present model with $c = 0.194$ and Sinha et al. [30] with $b'_1 = 0.28$, computed with respect to the shock-unsteadiness model presented in [30].

M_1	Present Model ($c = 0.194$)	Sinha et al. [30] ($b'_1 = 0.28$)
1.5	-4.98	-4.98
2.5	-0.45	2.5
3.5	1.24	8.79
4.7	-0.64	12.36
5	-1.4	12.85

$$\left(\frac{k_2}{k_1}\right) = \left(\frac{\tilde{u}_1}{\tilde{u}_2}\right)^{2(1-b'_1)/3} \quad (4.11)$$

Here, k_2/k_1 represents the jump in TKE across the shock wave. Fig. 4.2 shows the TKE amplifications across the shock obtained by plotting Eq. (4.10) for a range of upstream Mach numbers. The results are compared with the TKE amplifications obtained using Eq. (4.11). With the recommended value of c , the present model underpredicts the TKE amplification upto 5% in the range $1 < M_1 < 2.5$ and slightly overpredicts at higher Mach numbers up to 4.5. Overall, the error with respect to the shock-unsteadiness k - ϵ model of Sinha et al. [30] is less than 5% (See Table 4.1).

Another approach to make the shock-unsteadiness k -equation local is to choose a constant value of the model parameter b'_1 from its range of 0 to 0.4. Fig. 4.2 shows the

TKE amplification obtained using Eq. (4.11) with a constant value of $b'_1 = 0.28$. This constant value is chosen to match with the TKE amplification for $M_1 = 1.5$ obtained using the newly proposed model. Both the shock-unsteadiness k - ϵ and the present model with constant model parameters underpredict TKE amplification by about 5 % at $M_1 = 1.5$. As against the present model, the error in prediction gets significantly larger for the shock-unsteadiness model with constant b'_1 as the upstream Mach number increases (See Table 4.1). Thus, the new formulation improves predictions for a constant model parameter $c = 0.194$ as against taking a constant value of b'_1 .

4.2.2 Turbulent dissipation rate

Similar to b'_1 in the TKE equation, the model parameter $c_{\epsilon 1}$ in the solenoidal dissipation rate equation (4.4) is also a function of the upstream Mach number. To remove this dependency, we take an alternative form of the transport equation for the solenoidal dissipation rate ϵ_s from Sinha [91]. For the sake of completeness, the key steps in the modeling of ϵ_s are discussed below. For a detailed discussion, refer to Sinha [91].

Amplification of ϵ_s across the shock can be expressed in terms of the amplification in enstrophy $\overline{\omega'_i \omega'_i}$ and the changes in the kinematic viscosity $\bar{\nu}$,

$$\tilde{u} \frac{\partial \epsilon_s}{\partial x} = \tilde{u} \frac{\partial}{\partial x} (\overline{\nu \omega'_i \omega'_i}) = \overline{\tilde{u} \omega'_i \omega'_i} \frac{\partial \bar{\nu}}{\partial x} + \tilde{u} \bar{\nu} \frac{\partial}{\partial x} (\overline{\omega'_i \omega'_i}) \quad (4.12)$$

The amplification in enstrophy $\overline{\omega'_i \omega'_i}$ across the shock is modeled using LIA. Linearized Navier-Stokes equations are written in a reference frame attached to the unsteady shock wave to derive transport equations for the vorticity components. These are combined to obtain an equation for the enstrophy evolution across a time-averaged shock wave. A budget of this equation is computed using the results from LIA and DNS data to identify the dominant terms responsible for the amplification of the enstrophy across the shock. Closure approximations are then proposed for the unclosed correlations in the dominant source terms resulting in the following equation.

$$\tilde{u} \frac{\partial}{\partial x} (\overline{\omega'_i \omega'_i}) = -c_w \overline{\omega'_i \omega'_i} \frac{\partial \tilde{u}}{\partial x} \quad (4.13)$$

where $c_w = 1.55$. This equation governs the production of enstrophy due to the mean flow compression across the shock and is used to derive an equation for ϵ_s .

The dynamic viscosity $\bar{\mu} = \bar{\nu} \bar{\rho}$ is assumed to be a function of mean temperature $\tilde{T}$, such that $\bar{\mu} \propto \tilde{T}^\alpha$ where $\alpha = 0.76$ is a constant. The change in kinematic viscosity across the shock can thus be calculated as,

$$\frac{\partial \bar{v}}{\partial x} = -\frac{\bar{\mu}}{\bar{\rho}^2} \frac{\partial \bar{\rho}}{\partial x} + \frac{1}{\bar{\rho}} \frac{\partial \bar{\mu}}{\partial x} = \bar{v} \left[\frac{1}{\tilde{u}} \frac{\partial \tilde{u}}{\partial x} + \frac{\alpha}{\tilde{T}} \frac{\partial \tilde{T}}{\partial x} \right] \quad (4.14)$$

Combining Eqs. (4.13) and (4.14), we get

$$\bar{\rho} \tilde{u} \frac{\partial \epsilon_s}{\partial x} = (1 - c_w) \bar{\rho} \epsilon_s \frac{\partial \tilde{u}}{\partial x} + \alpha \bar{\rho} \epsilon_s \frac{\tilde{u}}{\tilde{T}} \frac{\partial \tilde{T}}{\partial x} - c_2 \bar{\rho} \frac{\epsilon_s^2}{k} \quad (4.15)$$

where viscous dissipation with $c_2 = 1.2$ is added to match the turbulence decay in DNS data on each side of the shock wave. The first two terms on the right-hand-side of the above equation are responsible for the amplification in ϵ_s across the shock, while the last term causes its decay on either sides of the shock wave. The equation neglects the effects due to viscous and turbulent diffusion and can be assumed to hold for high Reynolds number flows.

Analytical values of the amplification in the turbulent dissipation rate in the inviscid limit can be obtained by neglecting the dissipation of ϵ_s from Eq. (4.15) and then integrating it across the shock,

$$\left(\frac{\epsilon_{s2}}{\epsilon_{s1}} \right) = \left(\frac{\tilde{u}_1}{\tilde{u}_2} \right)^{(c_w-1)} \left(\frac{\tilde{T}_2}{\tilde{T}_1} \right)^\alpha \quad (4.16)$$

Comparing the jumps in the turbulent dissipation across the shock obtained using Eq. (4.16) and those obtained from Eq. (4.4),

$$\left(\frac{\epsilon_{s2}}{\epsilon_{s1}} \right) = \left(\frac{\tilde{u}_1}{\tilde{u}_2} \right)^{(2/3)c_{\epsilon 1}} \quad (4.17)$$

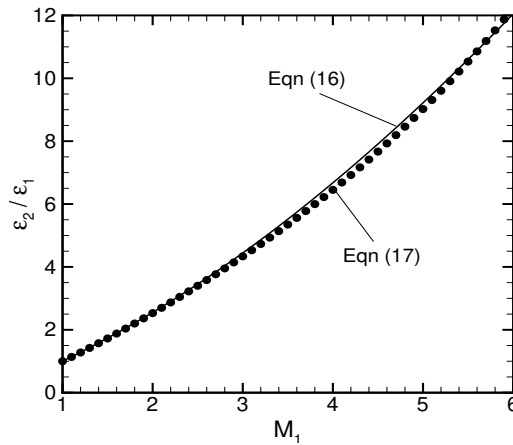


Figure 4.3: Amplification in the turbulent dissipation rate for different upstream Mach numbers as obtained using Eqs. (4.16) and (4.17).

shows that the results are almost identical for a range of upstream Mach numbers (see Fig. 4.3). Unlike TKE jump, the amplification of turbulent dissipation rate across the shock increases steadily with the increase in the upstream Mach number.

4.2.3 Non-conservative errors at a shock wave

The k and ϵ_s equations given by Eqs. (4.8) and (4.15) respectively are non-conservative in nature. This is due to the fact that derivatives in the source terms on the right-hand-side of both the equations are multiplied by the flow variables. Sinha et al. [75] studies the numerical issues involved while solving the non-conservative form of the shock-unsteadiness k - ϵ equations. It is pointed out that for shock capturing simulations, like those in this work, the non-conservative nature of the source terms can result in large numerical error at shock waves. The error can be minimized by casting all the terms in conservative form. The convection terms in the Eqs. (4.8) and (4.15) can be written in a conservative form using the Reynolds-averaged mass conservation equation. In a finite-volume approach, the convective fluxes are evaluated at cell interfaces. The corresponding truncation error involves higher-order derivatives of the flow variables. These derivatives take large values in the shock wave, but contributions from adjacent cells cancel at the common interfaces when all the finite volume cells spanning the shock wave are taken together. The net error across a shock wave is therefore a function of the higher-order derivatives evaluated in a region of smooth flow solution, outside the shock wave. On successive grid refinement, the leading-error term vanishes and the solution tends towards the exact integral. A detailed numerical error analysis at the shock wave can be found in the Appendix of [75] and is not repeated here.

The source terms in both the k and ϵ_s equations involve flow derivatives that are evaluated using a symmetric second-order discretization in the current CFD implementation. The corresponding truncation error involves third and higher-order derivatives of velocity, pressure and temperature. On integration over a finite volume cell, the error term thus involves integration of these higher order derivatives. The leading error term scales as $1/\delta^3$, where δ is the computed shock thickness, and hence the error takes large values in the region of a flow discontinuity. When all the finite volume cells spanning the shock wave are taken together, these error terms from the individual cells add up and appear as numerical internal volume sources. Their overall contribution across the shock wave therefore involves higher-order velocity, pressure and temperature derivatives evaluated in the region of flow discontinuity. Unlike the conservative flux terms, the telescopic effect of error cancellation between adjacent cells is absent in the non-conservative source

terms. This leads to errors that are comparable or larger than the physical production at the shock waves. Also, the numerical error at shock waves does not decrease in magnitude with successive grid refinements as observed for smooth solutions.

The characteristic of the numerical error obtained in simulations using non-conservative shock-unsteadiness k - ϵ model is reported in Sinha et al. [75]. Non-conservative nature of the source terms causes non-physical trends in the k and ϵ_s amplifications. In particular, the computed TKE amplification shows a non-monotonic trend with increasing Mach number, which is in contrast to the variations observed in DNS and LIA. The results are also qualitatively and quantitatively different from the exact integration of the governing equations. To restrict the truncation errors at shock waves, Sinha et al. [75] proposes an alternative form of the shock-unsteadiness k - ϵ equations obtained by a transformation of the turbulence variables. Conserved quantities involving k and ϵ are formulated and transport equations for these new variables are derived. These new turbulence model equations are free of the non-conservative source terms and show dramatic improvement in numerical accuracy. Here, we follow a similar approach for Eqs. (4.8) and (4.15) to derive the corresponding conservative forms.

4.2.4 Conservative formulation

From Eq. (4.10), we can write $k_2 u_2^{2/3} p_2^{2c/3} = k_1 u_1^{2/3} p_1^{2c/3}$ such that the quantity

$$f = k u^{2/3} p^{2c/3} \quad (4.18)$$

is conserved across the shock. A transport equation for f can be derived by differentiating f w.r.t. x ,

$$\begin{aligned} \rho u \frac{\partial f}{\partial x} &= \rho u^{2/3+1} p^{2c/3} \frac{\partial k}{\partial x} + \frac{2}{3} \rho k u^{2/3} p^{2c/3} \frac{\partial u}{\partial x} \\ &+ \frac{2}{3} c \rho k u^{2/3+1} p^{2c/3-1} \frac{\partial p}{\partial x} \end{aligned} \quad (4.19)$$

Note that from now onwards, the overbars on ρ and p and tilde on u and T are dropped to avoid potential conflict with the sign of the exponent. Multiplying the TKE equation (4.8) by $u^{2/3} p^{2c/3}$, we get

$$\begin{aligned} \rho u^{2/3+1} p^{2c/3} \frac{\partial k}{\partial x} &= -\frac{2}{3} \rho k u^{2/3} p^{2c/3} \frac{\partial u}{\partial x} - \frac{2}{3} c \rho k u^{2/3+1} p^{2c/3-1} \frac{\partial p}{\partial x} \\ &- \rho \epsilon_s u^{2/3} p^{2c/3} \end{aligned} \quad (4.20)$$

which on substituting in Eq. (4.19) and using mass conservation across the shock, we get

Table 4.2: Normalized values of k and ϵ at the inlet station for different Mach number cases.

M	$k_0 \times 10^2$	$\epsilon_0 \times 10^4$	DNS source
1.5	2.88	11.3	Larsson and Lele [58]
2.0	2.76	10.7	–
2.5	2.69	10.4	Larsson and Lele [58]
2.75	2.66	10.3	–
3.0	2.64	10.2	–
3.5	2.61	10.0	Larsson and Lele [58]
4.25	2.57	9.9	–
4.7	2.56	9.8	Larsson and Lele [58]

$$\frac{\partial}{\partial x}(\rho u f) = -\rho \epsilon_s u^{2/3} p^{2c/3} \quad (4.21)$$

This equation is mathematically identical to the TKE equation (4.8), but the source terms with non-conservative derivative have been eliminated. Due to this, the corresponding discretization error exhibits a telescoping effect at the cell interfaces. We thus get a well-behaved numerical solution at a flow discontinuity, as shown in the results.

Similarly, from Eq. (4.16), the conserved quantity across the shock turns out to be

$$g = u^{c_w-1} \epsilon_s T^{-\alpha} \quad (4.22)$$

and the corresponding transport equation is

$$\frac{\partial}{\partial x}(\rho u g) = -c_2 \rho \frac{\epsilon_s^2}{k} u^{c_w-1} T^{-\alpha} \quad (4.23)$$

Substituting $\epsilon_s = g u^{1-c_w} T^\alpha$ and $k = f u^{-2/3} p^{-2c/3}$, Eqs. (4.21) and (4.23) can be cast into the following form :

$$\frac{\partial}{\partial x}(\rho u f) = -\rho g (u^{\frac{5}{3}-c_w} T^\alpha p^{2c/3}) \quad (4.24)$$

$$\frac{\partial}{\partial x}(\rho u g) = -c_2 \rho \frac{g^2}{f} (u^{\frac{5}{3}-c_w} T^\alpha p^{2c/3}) \quad (4.25)$$

The above equations are similar to the original Eqs. (4.8) and (4.15), except for the absence of the production terms. The dissipation terms retain their original form with an additional factor of $(u^{\frac{5}{3}-c_w} T^\alpha p^{2c/3})$.

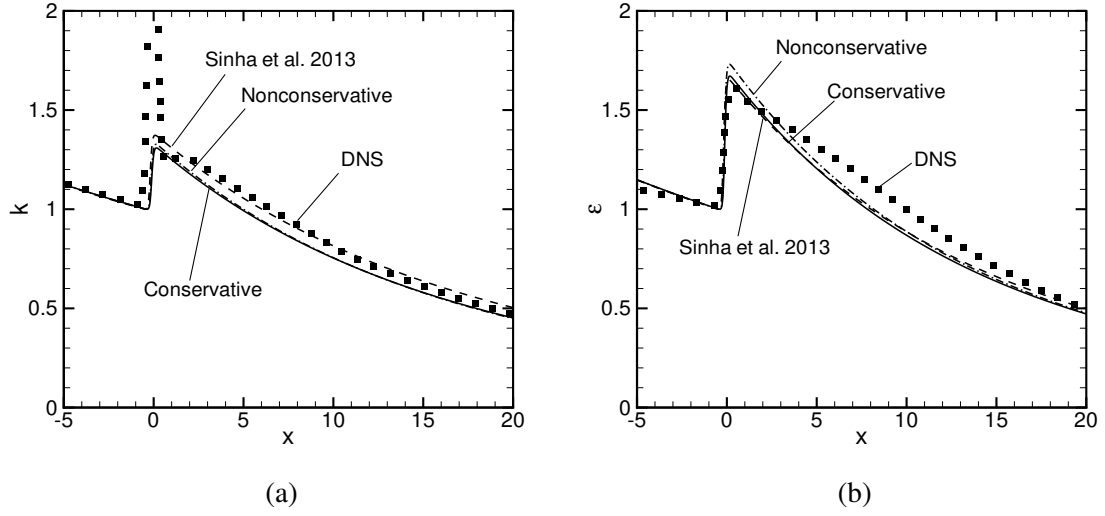


Figure 4.4: Streamwise variation of TKE and turbulent dissipation rate in the Mach 1.5 shock/homogeneous turbulence interaction shows that all model predictions are comparable to the DNS data.

4.3 Comparisons with DNS

We next evaluate the accuracy of the proposed turbulence model equations, both the conservative and non-conservative formulations. The results are compared with the DNS data of Larsson et al. [58] and conservative form of shock-unsteadiness k - ϵ model [75]. We note that the k - ϵ model proposed in [75] is based on the physics-based shock-unsteadiness correction of Sinha et al. [30] and is cast in a conservative form to eliminate numerical errors at shock discontinuities. It has been shown to match the DNS data well in the previous work. By comparison, the current k - ϵ model emulates the physics of the shock-unsteadiness model with constant turbulence model parameters. This is unlike the [75] model which has model parameters as function of the upstream Mach number, and hence lead to difficulty in CFD implementation for complex STBLI flows.

Evolutions of TKE and its dissipation rate along the streamwise direction are plotted for different upstream Mach numbers in Figs. 4.4 - 4.6. As earlier, the TKE and dissipation rate are normalized by their values just before the shock wave, and the simulations are performed using 1600 grid points. The TKE and dissipation rate profiles show decay on either sides of the shock and amplifications in their values can be seen only in the shock region. This is because the production of TKE is due to the mean velocity and mean pressure gradients, and the production of dissipation rate is due to the mean velocity and mean temperature gradients. The DNS data also shows high values of TKE at the shock. These are due to unsteady shock oscillations, and are not captured by the turbulence models.

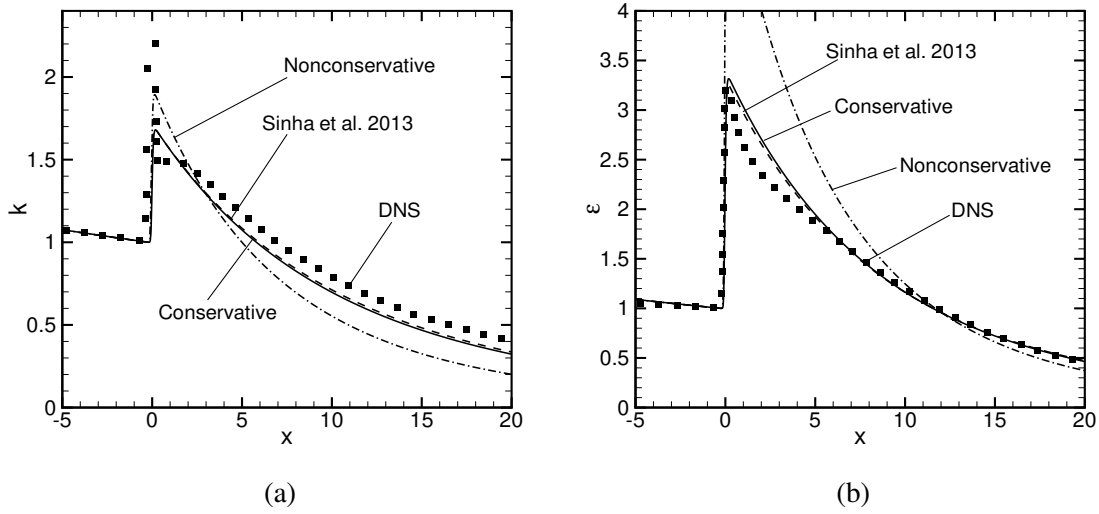


Figure 4.5: Spatial evolution of TKE and turbulent dissipation rate in the shock/homogeneous turbulence interaction for $M_1 = 2.5$.

The pre-shock decay of TKE and dissipation rate for the three test cases obtained using all the k - ϵ models match well with the DNS data. For the Mach 1.5 case, in Fig. 4.4, the model predictions match the DNS amplification of k and ϵ fairly well, with minor discrepancies in the post-shock evolution of the turbulence field. As noted earlier, the current model slightly underpredicts the TKE amplification as compared to [75] and DNS data at this Mach number. Further, both the conservative and non-conservative forms yield practically identical solutions for such weak shock interactions. There is a slight overprediction of the turbulent dissipation rate amplification by the non-conservative equations, as compared to the conservative form.

The effect of the numerical errors due to the non-conservative production terms in the model equations is prominent in the higher Mach number cases. The non-conservative amplification of turbulent dissipation rate dramatically increases with Mach number. It is higher by about 67% at Mach 2.5, whereas for the $M = 3.5$ case, the overprediction is over 425%. The TKE amplifications are also overpredicted for these cases (see Figs. 4.5 and 4.6). The excessive amplification of ϵ across the Mach 3.5 shock results in a steeper decay of both TKE and dissipation rate in the downstream flow. The turbulence quantities drop to negligible values behind the shock wave. Thus, the post-shock TKE and dissipation rate for the non-conservative equations are in complete disagreement with the results obtained using the conservative form, Sinha et al. [75] and DNS. The discrepancy increases with shock strength, and is because of the large numerical errors caused due to the non-conservative derivatives present in the model equations.

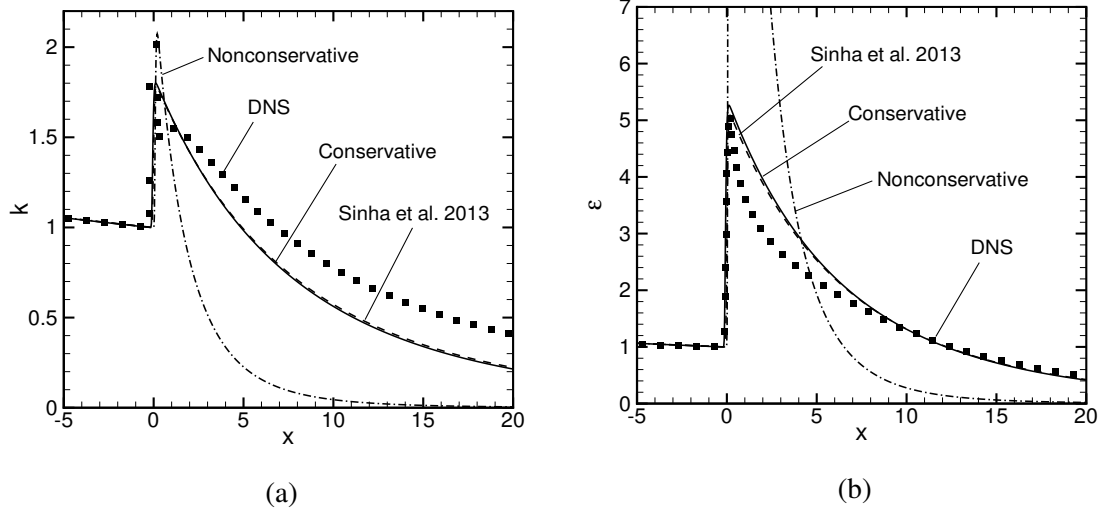


Figure 4.6: Streamwise variation of TKE and turbulent dissipation rate in the shock/homogeneous turbulence interaction for $M_1 = 3.5$.

The numerical results obtained using the conservative form of the proposed k - ϵ turbulence model are in excellent agreement with the physics-based model developed previously [30, 75]. Both the amplification across the shock and the downstream decay match DNS data closely at Mach 2.5 (Fig. 4.5). For the Mach 3.5 case (Fig. 4.6), the shock-jumps in k and ϵ are reproduced accurately, but the dissipation rate is over-predicted in the region $0 < x < 10$. This results in a faster decay of TKE behind the shock in this region. Further downstream, the conservative model Eqs. (4.24) and (4.25) predict a lower TKE than DNS, but the decay rate seems to match the DNS data well.

4.4 Extension to realistic flows

For application to realistic flows involving shock waves, the model equations (4.8) and (4.15) can be cast in a general tensor form as,

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho u_j k}{\partial x_j} = -\frac{2}{3} \rho k \frac{\partial u_i}{\partial x_i} - \frac{2}{3} c_p k \frac{u_i}{p} \frac{\partial p}{\partial x_i} - \rho \epsilon \quad (4.26)$$

$$\frac{\partial \rho \epsilon}{\partial t} + \frac{\partial \rho u_j \epsilon}{\partial x_j} = (1 - c_w) \rho \epsilon \frac{\partial u_i}{\partial x_i} + \alpha \rho \epsilon \frac{u_i}{T} \frac{\partial T}{\partial x_i} - c_2 \rho \frac{\epsilon^2}{k} \quad (4.27)$$

where we drop overbars and tilde, assume $\epsilon \approx \epsilon_s$ and the temporal derivative is introduced for unsteady flows. The equations have a form similar to the mean flow equations, and they can be solved either coupled or decoupled in a RANS code. It can be shown that the above equations are invariant under rotation of the co-ordinate axes. The conservative formulation in Eqs. (4.24) and (4.25) can also be generalized in a similar way, by replacing

the velocity dependence of the right-hand-side terms by equivalent expressions involving fluid density.

For turbulence interacting with an oblique shock, Eqs. (4.26) and (4.27) are written in a rotated co-ordinate frame such that the shock-normal direction coincides with the x' - axis,

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho u_n k}{\partial x'} = -\frac{2}{3} \rho k \frac{\partial u_n}{\partial x'} - \frac{2}{3} c_p k \frac{u_n}{p} \frac{\partial p}{\partial x'} - \rho \epsilon \quad (4.28)$$

$$\frac{\partial \rho \epsilon}{\partial t} + \frac{\partial \rho u_n \epsilon}{\partial x'} = (1 - c_w) \rho \epsilon \frac{\partial u_n}{\partial x'} + \alpha \rho \epsilon \frac{u_n}{T} \frac{\partial T}{\partial x'} - c_2 \rho \frac{\epsilon^2}{k} \quad (4.29)$$

where u_n is the shock-normal velocity component and the shock-parallel derivatives are neglected. The above equations have the same form as Eqs. (4.8) and (4.15), and thus the oblique-shock interaction is equivalent to a normal shock interaction in the shock-normal direction. Integrating the above equations across the shock wave, we get amplifications in k and ϵ as,

$$\left(\frac{k_2}{k_1} \right) = \left(\frac{\rho_2}{\rho_1} \right)^{2/3} \left(\frac{p_2}{p_1} \right)^{-2c/3} \quad (4.30)$$

$$\left(\frac{\epsilon_2}{\epsilon_1} \right) = \left(\frac{\rho_2}{\rho_1} \right)^{c_w - 1} \left(\frac{T_2}{T_1} \right)^\alpha \quad (4.31)$$

where $\rho_1 u_{n,1} = \rho_2 u_{n,2}$ across the shock is used. Similarly, the amplifications in k and ϵ across an oblique shock for the shock-unsteadiness k - ϵ model is given by,

$$\left(\frac{k_2}{k_1} \right) = \left(\frac{\rho_2}{\rho_1} \right)^{\frac{2}{3}(1-b'_1)} \quad (4.32)$$

$$\left(\frac{\epsilon_2}{\epsilon_1} \right) = \left(\frac{\rho_2}{\rho_1} \right)^{(2/3)c_{\epsilon 1}} \quad (4.33)$$

Fig. 4.7 plots the amplifications in TKE and turbulent dissipation rate across weak oblique shocks for varying flow deflection angles θ and upstream Mach numbers M_1 . The results are presented for a range of upstream Mach numbers up to 5 and the solutions are obtained for attached shocks. Detached shock cases are discussed separately. Note that the shock-normal Mach number M_{1n} is considerably lower than M_1 for oblique shocks with low deflection angles. This suggests that the model predictions are valid over a much larger range of upstream Mach numbers than that presented in Fig. 4.7. For $\theta = 15^\circ$, for example, hypersonic flows up to Mach 15 fall within the calibration range $1 \leq M_{1n} \leq 5$ considered in Section 4.2.1.

For a given deflection angle θ , TKE amplification increases with increasing Mach number (Fig. 4.7a). The current model gives qualitatively the same trend as the shock-unsteadiness k - ϵ model, but the predictions are lower in magnitude. The largest differences observed for $\theta = 15^\circ$ and $M_{1n} \simeq 2 - 3$ are within the 5% margin reported in Table

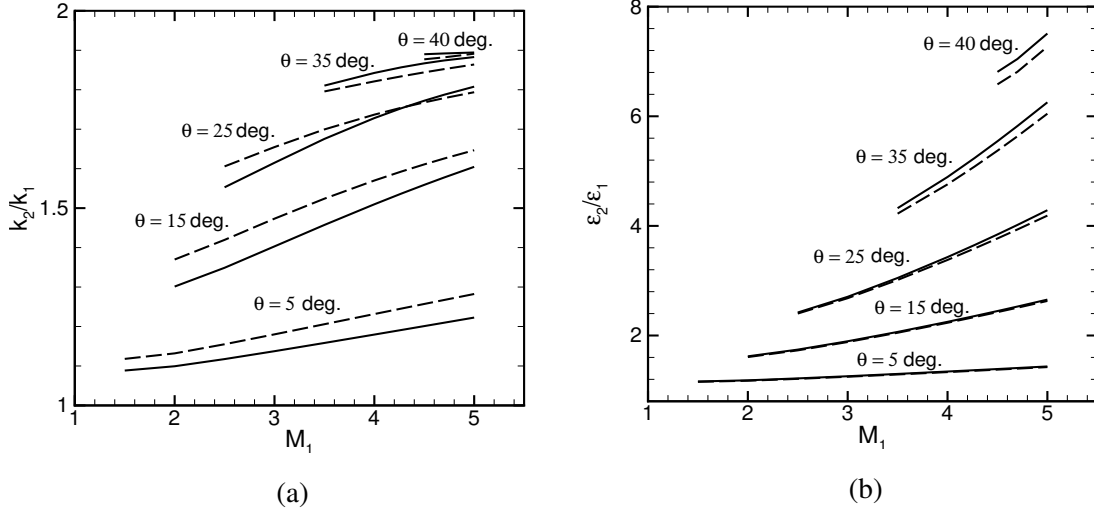


Figure 4.7: Amplifications in (a) TKE and (b) turbulent dissipation rate across weak oblique shocks obtained using the present model (solid lines) and the shock-unsteadiness $k-\epsilon$ model (dashed lines).

4.1. The two models are closer at higher deflection angles, where TKE amplification tends to approach its hypersonic asymptotic value. The ϵ -data presented in Fig. 4.7b also shows a close match between the current model and the shock-unsteadiness $k-\epsilon$ model for weak oblique shocks. The current predictions are higher than those of the previous model for all Mach numbers and flow deflection angles with a maximum difference of about 3.5%.

In case of strong oblique shocks, the model predictions for different flow deflection angles are close to each other. The k and ϵ amplifications are also comparable to the normal shock results presented in Figs. 4.2 and 4.3, and are not repeated here. As expected, the differences between the current model and the shock-unsteadiness $k-\epsilon$ model predictions are well within 5% for all Mach numbers and flow deflection angles considered.

The model prediction in realistic flows with Mach stems and lambda shocks [3, 159] would be as per the results shown for normal and oblique shocks, respectively. In case of STBLI flows, the mean flow gradients in the upstream boundary layer results in a curved shock with strength varying with distance from the wall. Application of the generalized model equations (4.26) and (4.27) would give turbulence amplification as per the local strength and inclination of the shock wave. The k and ϵ results would follow the $M_1 - \theta$ dependence presented in Fig. 4.7. Also, the model should be restricted to the supersonic part of a boundary layer, and not applied to the near-wall subsonic region where shock waves cannot penetrate.

Implementation of the current model in realistic flow simulations require a shock-identifying function, such that the model is applied in high-compression regions and we

revert back to the standard turbulence model outside of shock waves. A function based on the mean dilatation has proved effective for the shock-unsteadiness model applied to different STBLI configurations at varying Mach numbers [27, 34]. See, for example, the impingement of an oblique shock on a flat plate boundary layer, where the shock-unsteadiness model is applied in regions covered by the incident and transmitted shocks, as well as the separation and reattachment shock waves [35]. Implementation of the proposed model in a general CFD code will follow identical steps with appropriate modification required for the pressure and temperature gradient terms. A detailed validation of the model against experimental data for different geometric configurations, with varying freestream and wall boundary conditions, is presented in the subsequent sections.

The behaviour of the current model is expected to be similar to the shock-unsteadiness model, which suppresses turbulent kinetic energy amplification at shock waves. A lower turbulence at the separation shock leads to an early separation of the turbulent boundary layer [35], and thus alleviates one of the key limitation of standard $k-\epsilon$ models [1, 2]. Matching the separation bubble size leads to an accurate prediction of the shock topology in the interaction region [27], including the location and inclination of different shocks, as well as their interaction with each other. The simulations are thus able to predict the experimental data for surface pressure and skin friction coefficient better than conventional models [34]. Improvements in wall heating rates, however, are limited, and require additional modeling as proposed in recent works [160, 161]. Overall, the models based on shock-unsteadiness damping have proved their potential in a variety of SBLI configurations in supersonic and hypersonic flows. The results are also consistent across different one- [91] and two-equation turbulence models [27], and the same is expected for the new $k-\epsilon$ model proposed in this paper.

The shock-unsteadiness term in the TKE equation is derived using the conservation equations at an oscillatory shock wave. On the other hand, the shock unsteadiness effect does not appear explicitly in the enstrophy and ϵ equation written at the shock wave ([91]). On account of this, introducing an extra shock-unsteadiness like term in the ϵ and ω equations is not considered in this work. Also, changing only the TKE equation maintains consistency with the earlier works [27, 34, 35].

The local SU $k-\epsilon$ model equations are to be solved in the regions of high-compression regions, i.e., shock waves and everywhere else in the flow-field, a conventional turbulence model, such as the standard $k-\epsilon$ or $k-\omega$ model, with good performance in the compressible boundary layer flows has to be solved. In the earlier works of Sinha et al. [34], Pasha and Sinha [35] and [27], the non-local shock-unsteadiness model has been successfully used with a standard $k-\epsilon$ or $k-\omega$ model set-up. However, in these works, only

the k -equation of the shock-unsteadiness model has been implemented and the shock-unsteadiness ϵ equation is not solved. This was done in order to make the CFD implementation of the shock-unsteadiness model simple. Also, with the implementation of the shock-unsteadiness k equation only, significant improvements in the results of the k - ϵ and k - ω models have been observed.

In this work, we will follow the methodology adopted in the earlier works involving the shock-unsteadiness model, and we will solve only the local form of the k -equation along with either the standard k - ϵ or k - ω model. The local SU ϵ equation will not be solved. Following sections discuss the blending of the local SU k model equation with the standard k - ϵ model of Launder and Sharma [79] and the standard k - ω model of Wilcox [80].

4.5 Local shock-unsteadiness k - ϵ model

To apply the local shock-unsteadiness modification to the k - ϵ turbulence model, an approach similar to that used in the works of [34], [35], and [27] is followed. The shock-unsteadiness modification is applied only in the regions of shock waves in the flow-field and everywhere else the standard model is used. To identify the shock waves in a flow-field, a shock-detecting function f_s is used, which varies smoothly from zero in shock-free regions to a value of one in shock waves. Then, using f_s , a suitable blending function of the form c'_μ is devised such that the production term of the TKE equation takes a combined form of the first two terms in equation (4.26) when $f_s = 1$ (shock regions). It reverts back to the standard production term outside of shock waves. The modified production term is given by,

$$P'_k = c'_\mu \mu_T \left(2S_{ij}S_{ji} - \frac{2}{3}S_{ii}^2 \right) - \frac{2}{3}\bar{\rho}kS_{ii} \quad (4.34)$$

where,

$$\begin{aligned} \mu_T &= c_\mu f_\mu \bar{\rho} \frac{k^2}{\epsilon} \quad ; \quad c'_\mu = 1 - f_s \left(1 + \frac{2c}{3} \frac{\bar{u}_i}{\bar{p}} \frac{\partial \bar{p}}{\partial x_i} \frac{1}{c_\mu f_\mu S \zeta} \right) \\ \zeta &= \frac{Sk}{\epsilon} \quad ; \quad c_\mu = 0.09 \quad ; \quad f_\mu = \exp \left[\frac{-3.4}{(1 + 0.02R_t)^2} \right] \quad ; \quad R_t = \frac{\bar{\rho}k^2}{\mu\epsilon} \end{aligned} \quad (4.35)$$

The blending function c'_μ for the local SU model, thus, has no dependence on the upstream quantities, which is not the case with the non-local SU model given in [34]. The blending function c'_μ devised for the non-local SU model in [34] is a function of b'_1 and thus it depends on the flow quantities upstream of a shock. Here, mean strain-rate and its magnitude is respectively given by,

$$S_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad \text{and} \quad S = \left(2S_{ij}S_{ji} - \frac{2}{3}S_{ii}^2 \right)^{1/2} \quad (4.36)$$

Following shows the form of the production term in the shock regions when $f_s = 1$. At a shock wave, the blending function reduces to,

$$c'_\mu = -\frac{2c}{3} \frac{u_i}{p} \frac{\partial p}{\partial x_i} \frac{1}{c_\mu f_\mu S \zeta} \quad (4.37)$$

Therefore, the modified production term becomes

$$\begin{aligned} P'_k &= -\frac{2}{3} \mu_T c \frac{u_i}{p} \frac{\partial p}{\partial x_i} \frac{1}{c_\mu f_\mu S \zeta} S - \frac{2}{3} \rho k S_{ii} \\ &= -\frac{2}{3} \mu_T c \frac{u_i}{p} \frac{\partial p}{\partial x_i} \frac{1}{c_\mu f_\mu \frac{S k}{\epsilon}} S - \frac{2}{3} \rho k S_{ii} \\ &= -\frac{2}{3} f_\mu c_\mu \frac{\rho k^2}{\epsilon} \frac{u_i}{p} \frac{\partial p}{\partial x_i} \frac{1}{c_\mu f_\mu \frac{k}{\epsilon}} - \frac{2}{3} \rho k S_{ii} \\ P'_k &= -\frac{2}{3} c \rho k \frac{u_i}{p} \frac{\partial p}{\partial x_i} - \frac{2}{3} \rho k S_{ii} \end{aligned} \quad (4.38)$$

which is exactly the form of the local SU k -equation.

4.6 Local shock-unsteadiness k - ω model

The k -equation of the k - ω model is also similarly modified, in-line with the works of [34], [35], and [27]. A blending function c'_μ , similar to that discussed for the k - ϵ model set-up, is devised and the modified form of the production term of the k -equation for the k - ω model is given by,

$$P'_k = c'_\mu \mu_T \left(2S_{ij} S_{ji} - \frac{2}{3} S_{ii}^2 \right) - \frac{2}{3} \rho k S_{ii} \quad (4.39)$$

where the eddy viscosity and blending function are given by,

$$\mu_T = \frac{\bar{\rho} k}{\omega} ; c'_\mu = 1 - f_s \left(1 + \frac{2c}{3} \frac{\bar{u}_i}{\bar{p}} \frac{\partial \bar{p}}{\partial x_i} \frac{1}{S \zeta} \right) ; \zeta = \frac{S}{\omega} \quad (4.40)$$

The shock detecting function f_s for the two-dimensional supersonic compression corner cases is taken from [34] and is given by,

$$f_s = \frac{1}{2} - \frac{1}{2} \tanh \left(\frac{5S_{ii}}{S} + 3 \right) \quad (4.41)$$

On the other hand, for the Schulein's Mach 5 incident shock wave turbulent boundary layer interaction cases, f_s is taken from the work of [35] and is given by,

$$f_s = \frac{1}{2} - \frac{1}{2} \tanh \left(\frac{12S_{ii} \delta_0}{U_\infty} + 4 \right) \quad (4.42)$$

where, δ_0 is the incoming boundary layer thickness and U_∞ is the free-stream velocity.

4.7 Numerical error at shock waves

Turbulence model equations can pose severe numerical challenges at shock waves. This is because of the production term in conventional models that is proportional to the square of the mean flow velocity gradient. The production term is often artificially limited to get physically realistic results. On the other hand, the shock-unsteadiness model uses physical arguments to suppress eddy viscosity at shock waves. This significantly reduces the numerical error in the high gradient region of shock waves and results in a realistic value of TKE across a shock (see Figure 7 in [30]). The additional damping provided by the shock-unsteadiness term is able to match DNS data over a range of shock Mach numbers.

The shock-unsteadiness turbulence model, when implemented using different numerical methods, can give different results on an identical computational grid. This is because different numerical methods employ varying degree of numerical dissipation to capture the shock discontinuity. The effect of numerical dissipation on shock-turbulence modeling has been studied earlier in the context of canonical shock-turbulence interaction [60]. It is found that a highly dissipative scheme will give a lower amplification of the TKE compared to a numerical method with low numerical dissipation, on an identical computational grid. On refining the grid, the post-shock TKE value increases and comparable values are obtained with a dissipative scheme on a much finer grid. The main point is that, grid refinement assumes additional significance in the context of shock-turbulence interaction and a sufficiently refined mesh should give identical results for different numerical schemes. See the study of shock-boundary layer interaction on a double-cone geometry by [162].

Numerical error analysis of turbulence models at a shock wave is presented in [75], where a large error in TKE amplification is observed across strong shock waves. The error is small at lower Mach numbers in the range $M < 2$ [61]. These results are for normal shock waves and can be interpreted for oblique shocks in SBLI in terms of the Mach number normal to the shock wave (M_{1n}). For the present cases, M_{1n} values are in the range of 1-2 for the separation shocks [34, 35], which are important for predicting separation bubble size.

Also, note that the above discussion is for inviscid scenario representing Reynolds number $Re \rightarrow \infty$. For finite Re representative of realistic SBLI flows, the numerical error at the eddy viscous shock waves is also shown to be minimal for the SU model [163]. In the present study, we use a low-dissipation version of the Steger-Warming numerical method to capture the shock waves and the results presented in the paper are obtained after

a detailed grid refinement study. Similar results are expected even with other numerical schemes, once their respective grid convergence is achieved.

The local shock-unsteadiness k - ϵ and k - ω models are applied to the compression ramp and the oblique shock impingement SBLI flows. The models are validated for six different flow cases; three each from the two flow configurations considered. The validation of the local SU models is done by comparing the obtained results for the surface pressure and skin friction coefficient with those obtained using the non-local variant of the SU model. The non-local model results have been extensively compared with experimental data in earlier works [34, 35]. Here, we present comparisons with the experimental measurements for the strongest interaction cases in the two flow configurations.

4.8 2D supersonic compression corner simulations

Three compression ramp flows with free-stream Mach number of $M_\infty \simeq 2.85$ and ramp angles of 16° , 20° , and 24° are computed to test the performance of the local SU k - ϵ and k - ω models. The Reynolds number is about $7 \times 10^7 \text{ m}^{-1}$, and the momentum thickness of the incoming boundary layer is about $1.2 \times 10^{-3} \text{ m}$ for all these three flow cases. The three ramp angles span the range from near incipient separated flow (in the case of 16° ramp angle) to fully separated flows (in the case of 20° and 24° ramp angles). The experiments were reported by Settles and Dodson and the experimental data is taken from [164]. The experimental uncertainties are quoted as approximately $\pm 2\%$ for wall pressure, and $\pm 15\%$ for skin friction coefficient.

The mean flow and turbulence profiles at the inlet boundary of the compression ramp and incident oblique shock simulations are obtained from separate flat plate simulations using the standard k - ϵ and k - ω models. The value of momentum thickness is matched to obtain the inlet profiles. For the three compression ramp cases, the momentum thickness of the boundary layer at the inlet is around 1.2 mm . On the other hand, the free-stream temperature and density are taken to be in the range of $100 - 102 \text{ K}$ and $0.82 - 0.88 \text{ kg/m}^3$, respectively. The wall temperature for all the three compression ramp cases of 24° , 20° and 16° is taken to be in the range of $274 - 282 \text{ K}$. Readers are referred to [34] for the exact free-stream and wall temperature conditions used in the computations of the three compression ramp cases. For these simulations, k , ϵ and ω values at the boundary layer edge in the inlet profile are prescribed as the free-stream conditions at the top boundary. An extrapolation condition for the mean and turbulence variables is used at the top and exit boundaries.

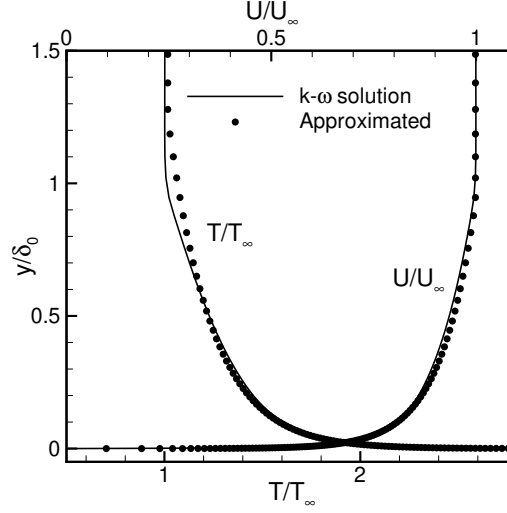


Figure 4.8: Comparison between the $k-\omega$ model solution and the approximated logarithmic function velocity and temperature profiles just upstream of the separation shock shows a good match.

A body-fitted structured grid used for all the computations, similar to that used in Sinha et al.[34]. The grid has grid points clustered near the wall and in the region of compression corner. The grid is exponentially stretched away from the wall and in the upstream and downstream directions of the compression corner. The y^+ value for the first grid point from the wall, i.e., y_2^+ was never greater than 0.6 for all the turbulence models and the cases considered. We use 170×200 grid points for the $k-\epsilon$ model and 120×110 grid for the $k-\omega$ simulations. These are based on an extensive grid refinement study carried out in [34] for the same geometry.

4.8.1 Numerical implementation

As discussed earlier, the original SU model requires the information of upstream shock-normal Mach number M_{1n} to calculate the model parameter b'_1 . For the two-dimensional compression ramp flows, the flow upstream of the interaction region is well characterized. The stream-wise velocity $\tilde{u}_1$ and temperature $\tilde{T}_1$ profiles upstream of the separation shock can be approximated using a logarithmic function of the form $(a + b \log_e y)$, where a and b are arbitrary constants and they assume different values for $\tilde{u}_1$ and $\tilde{T}_1$ profiles. Here, y is the distance from the wall. Figure 4.8 shows that the mean velocity and temperature profiles just upstream of the separation shock wave obtained using the $k-\omega$ model solution and the logarithmic function compare well. The upstream stream-wise Mach number M_1 can thus be computed as a function of y . Now, a unit vector normal to the shock wave can be obtained by taking gradient of pressure divided by its magnitude.

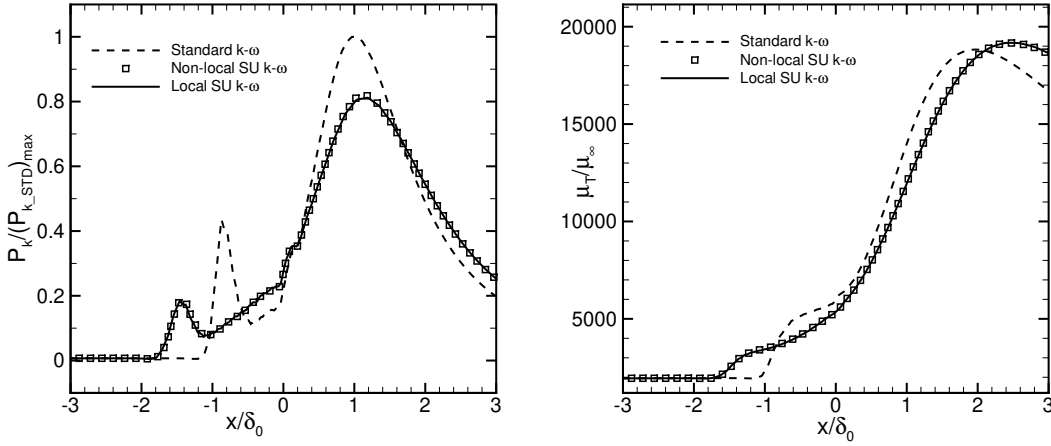


Figure 4.9: Variation of (a) normalized TKE production and (b) normalized eddy viscosity for different versions of the $k-\omega$ model along a streamline originating at $y = 0.2\delta_0$ in the incoming boundary layer for the 24° case.

Thus, in the regions of shock waves as identified by f_s , the Mach number normal to the shock wave M_{1n} can be obtained by taking a dot product with the unit pressure gradient vector. Mathematically, it can be written as

$$M_{1n} = M_1 \left[\frac{\partial \bar{p} / \partial x}{\sqrt{(\partial \bar{p} / \partial x)^2 + (\partial \bar{p} / \partial y)^2}} \right] \quad (4.43)$$

The local value of M_{1n} is then used to evaluate the shock-unsteadiness damping parameter b'_1 at each point in the shock, which is then used to calculate the blending function c'_μ .

The need to identify the upstream flow direction and compute upstream shock-normal Mach number M_{1n} makes the CFD implementation of the original SU model complex. The degree of complexity in identifying the upstream flow direction and thereby computing M_{1n} is different for different SBLI flows. The complexity increases for the flows where the upstream flow is not well characterized or in the presence of multiple shock waves. In such complex scenarios, the method presented above for computing M_{1n} may not work and another alternative method would be needed. The local SU model, on the other hand, has a constant model parameter c , and computation of the blending function c'_μ does not require any information of the shock-upstream mean flow quantities. As a result, complications in the CFD implementation of the original SU model, because of the additional computation of the non-local quantity M_{1n} , are thus avoided. It is shown in the subsequent sections that this simplification in the CFD implementation of the local SU model over the original SU model comes without affecting the accuracy of predictions.

Figure 4.9 shows the variation of normalized TKE production and eddy-viscosity for different versions of the $k-\omega$ model along a streamline originating at $y = 0.2\delta_0$ in

the incoming boundary layer. The maximum value of the standard $k-\omega$ model's TKE production is used to normalize the TKE production term of all the models and free-stream molecular viscosity is used to normalize the eddy-viscosity values. There is an exact match of the TKE production and eddy-viscosity values between the non-local and local SU modified $k-\omega$ models. The different $k-\omega$ models follow a similar trend for the TKE production and eddy-viscosity variation along the streamline. The TKE production and eddy-viscosity get amplified across the separation shock and continue increasing in the shear layer enclosing the separated flow region at the corner. Their highest values are attained in the reattachment region, followed by a decrease in the boundary layer recovery region on the ramp. The SU modification reduces the TKE production and eddy-viscosity by about 55% and 40%, respectively across the separation shock along the considered streamline as compared to the standard model. Also, the SU modified models predict about 18% less TKE production across the reattachment shock as compared to the standard model. The $k-\epsilon$ simulations also show a similar variation of TKE production and eddy-viscosity along streamlines, and is not shown here.

4.8.2 Comparison with experiment

24° case

The normalized surface pressure distribution and the skin friction coefficient computed using the standard, shock-unsteadiness (SU) and the local shock-unsteadiness versions of the $k-\epsilon$ and $k-\omega$ models are shown in figures 4.10 and 4.11, respectively. The streamwise distance s along the plate and the ramp is normalized by the incoming boundary-layer thickness δ_0 , where $s = 0$ represents the corner. It can be seen that the local forms of the shock-unsteadiness $k-\epsilon$ and $k-\omega$ models yield identical results for the surface pressure and skin friction coefficient as that of the SU $k-\epsilon$ and $k-\omega$ models of [34].

The simulation results are compared with the experimental data of [164]. The shock-unsteadiness modification results in a better prediction of the wall pressure as compared to their standard counterparts. The pressure rise location at the separation shock as predicted by both the shock-unsteadiness versions of the $k-\epsilon$ and $k-\omega$ model matches well with the experimental data. The shock-unsteadiness modification results in a pressure plateau higher than that in the corresponding standard model and reproduces the experimental pressure rise on the ramp accurately.

The skin friction plots in figures 4.11(a) and 4.11(b) show that the standard models predict a later separation and reattachment than the experiment. The C_f values on the ramp are highly overpredicted for the standard $k-\epsilon$ model, whereas the overprediction is

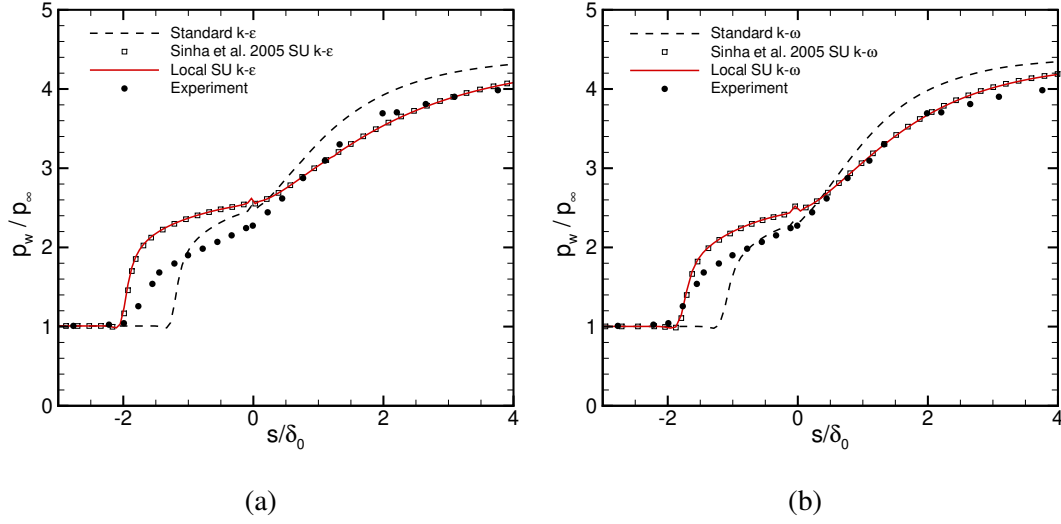


Figure 4.10: Comparison of wall pressure obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 24° case.

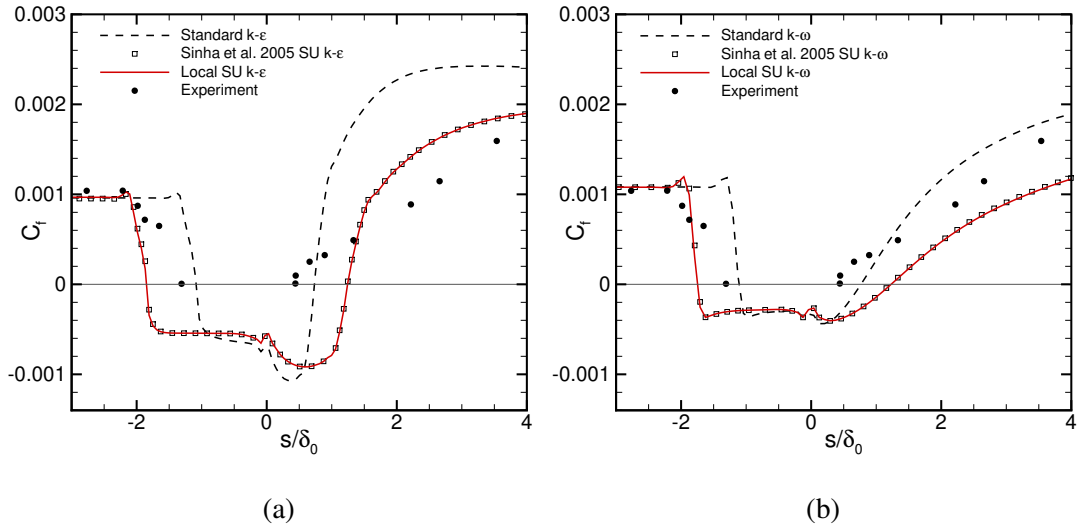


Figure 4.11: Comparison of skin friction coefficient obtained using the (a) $k-\epsilon$ and (b) $k-\omega$ model variants with the experimental data for 24° case.

less for the standard $k-\omega$ model. The drop in C_f at $s \simeq 2\delta_0$, as predicted by the modified $k-\omega$ and $k-\epsilon$ models, agree well with the experimental measurements, but the size of the separation region is overpredicted. Finally, the modification reduces C_f on the ramp as compared to the standard models.

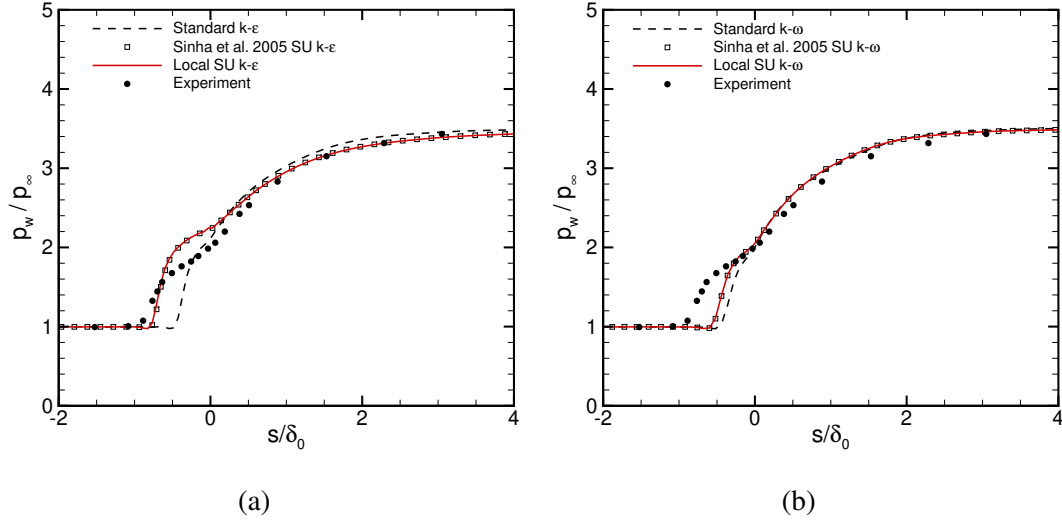


Figure 4.12: Comparison of wall pressure obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 20° case.

20° case

The local form of the shock-unsteadiness model is next tested for the 20° compression ramp flow. Figure 4.12 shows the simulation results for the wall pressure along with the experimental data by [164]. In this case also, the local SU $k-\epsilon$ and $k-\omega$ models exactly match with the results obtained using the corresponding non-local models. Similar to the 24° case, the location of the separation shock pressure rise as predicted by the standard $k-\epsilon$ and $k-\omega$ models is far downstream of the experimental data. The shock-unsteadiness correction moves the separation shock upstream. The effect of the modification is small in the $k-\omega$ simulation, whereas the modified $k-\epsilon$ model results are closer to the experimental separation shock location. Also, note that the modified $k-\epsilon$ model results in a higher pressure plateau in the separation region, similar to that in the 24° case. All of the models agree well with the pressure measurements on the ramp.

The simulation results for the skin friction coefficient are shown in figure 4.13. Again, the local forms of the SU models yield identical results as that given by their non-local counterparts. The results for all the turbulence models follow a pattern similar to that obtained in the 24° ramp flow. The standard $k-\epsilon$ and $k-\omega$ models predict a smaller separation region than in the experiment. The C_f values on the ramp are highly overpredicted by the two standard models, and the overprediction is more for the standard $k-\epsilon$ model. The shock-unsteadiness modification to the $k-\epsilon$ model matches the drop in C_f values at $s \approx -\delta_0$ better than in case of SU $k-\omega$ models. The location of the separation shock is also better predicted by the shock-unsteadiness variants of the corresponding

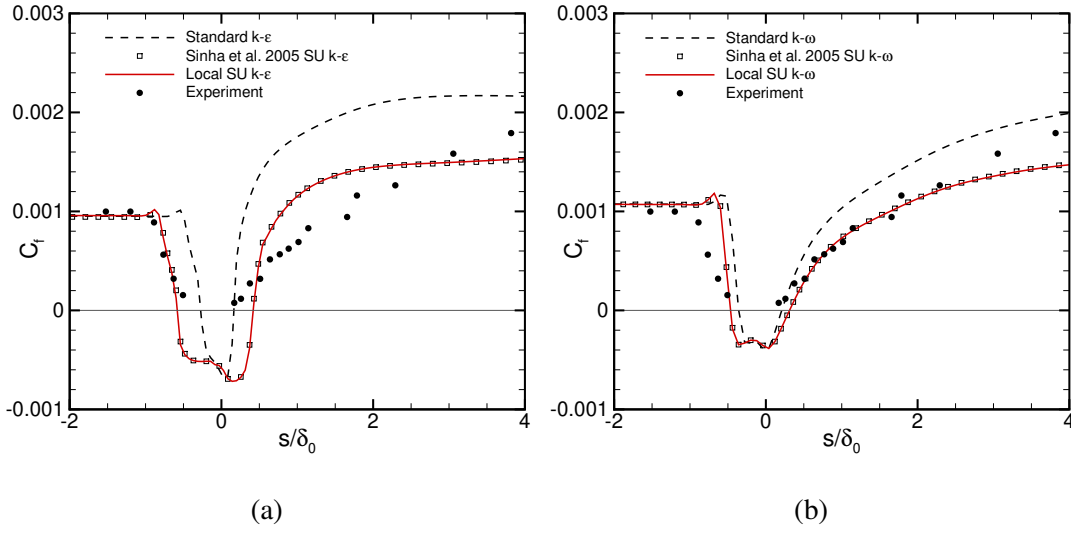


Figure 4.13: Comparison of skin friction coefficient obtained using the (a) $k-\epsilon$ and (b) $k-\omega$ model variants with the experimental data for 20° case.

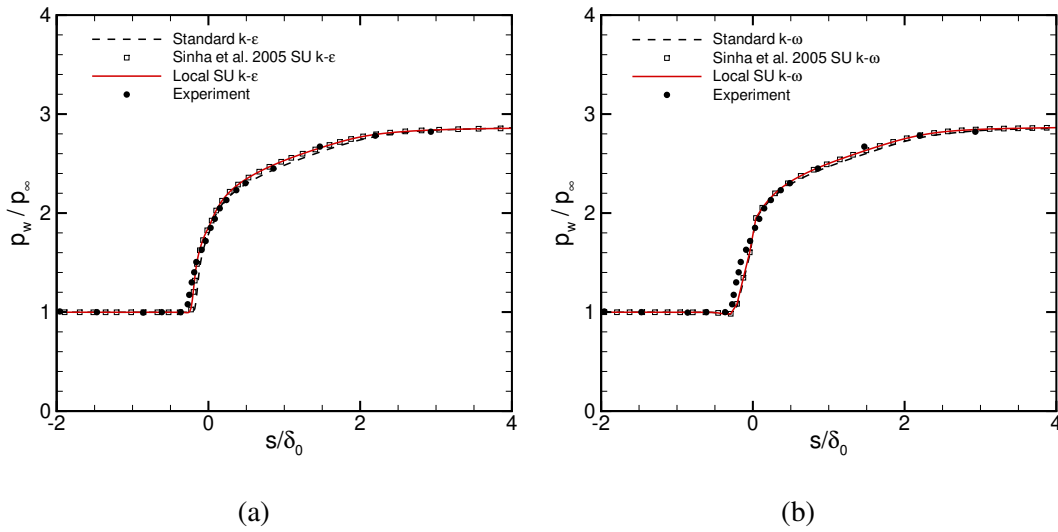


Figure 4.14: Comparison of wall pressure obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 16° case.

standard models. The modified $k-\epsilon$ and $k-\omega$ models result in a lower value of C_f on the ramp as compared to the standard models and the SU $k-\omega$ model predictions overlap with the experimental data in the recovering boundary layer up to $x = 2.5\delta_0$.

16° case

The flow over a 16° compression ramp is simulated using the standard and modified models, and the results are shown in figures 4.14 and 4.15. The flow, in this case, is near

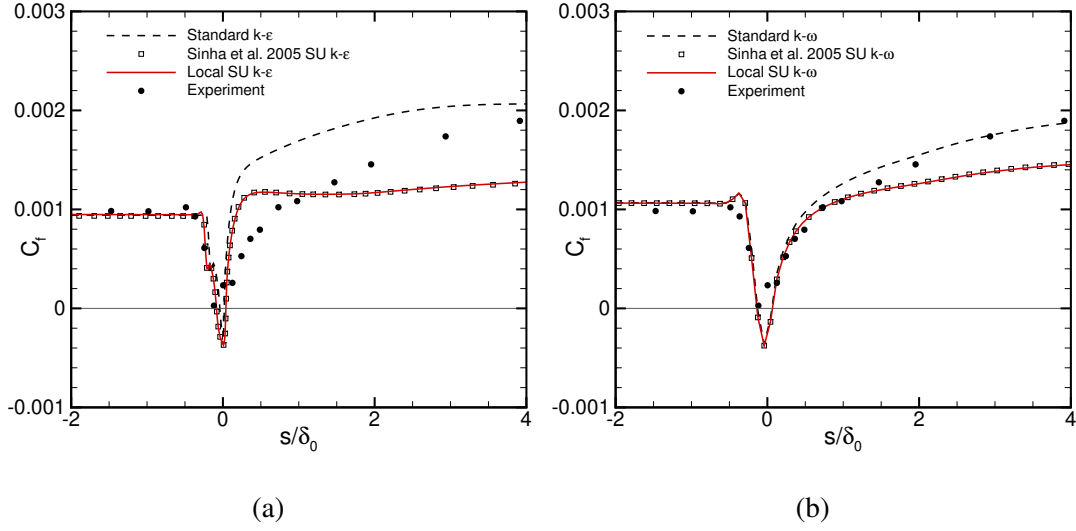


Figure 4.15: Comparison of skin friction coefficient obtained using the (a) $k-\epsilon$ and (b) $k-\omega$ model variants with the experimental data for 16° case.

incipient separation so that the pressure variation (figure 4.14) does not have the distinct pressure plateau that is observed at higher ramp angles. In this case, the standard $k-\epsilon$ and $k-\omega$ models almost exactly reproduce the experimental pressure measurements. The results obtained using the modified versions of the $k-\epsilon$ and $k-\omega$ models also match well with the experimental data and with the results obtained using the standard models. This shows that the effect of the shock-unsteadiness correction on the pressure distribution is negligible for small ramp angles, where the standard models predictions are close to the experimental data.

The skin friction coefficient (figure 4.15) as computed using the different turbulence models show a small region of separated flow at the corner. The standard and modified model results are identical on the plate ($s < 0$) and agree well with the experimental data. However, the model predictions differ in the recovery region on the ramp. The standard $k-\epsilon$ model overpredicts the skin friction, whereas the standard $k-\omega$ model matches the experimental data. The shock-unsteadiness modification results in a lower value of C_f on the ramp for both the models. Once again, the results for the local form of the SU models exactly match with the results obtained using their non-local counterparts.

4.9 Impinging oblique SBLI simulations

The selected test case consists of a Mach 5 flow with oblique shock impinging on a turbulent boundary layer developed over a flat plate. Three different shock generator angles

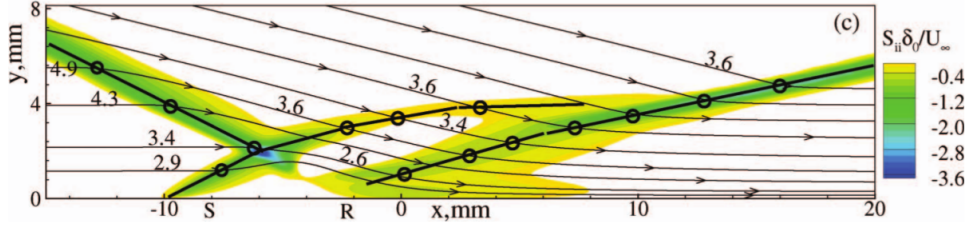


Figure 4.16: Distribution of normalized mean dilatation in the computed flow-field for 14° shock generator angle. Here, δ_0 is the incoming boundary layer thickness and U_∞ is the free-stream velocity. Points are identified in the shock waves and upstream Mach number is obtained along the corresponding streamlines. (Taken from [150]).

are considered and the details of experimental conditions can be found in Section 3.4 of Chapter 3. A structured Cartesian grid is used for the computations and the simulation methodology adopted is similar to the works of Pasha and Sinha [35]. Extensive grid convergence study, similar to that in [35], but with the local form of the shock-unsteadiness $k-\epsilon$ and $k-\omega$ models is carried out. Details are omitted for the sake of brevity. The skin-friction coefficient (C_f) is found to be the most sensitive parameter for grid convergence and the solutions are said to be grid converged if the change in C_f is less than 5% for the successively refined grids. The grid convergence study suggests that the local SU $k-\epsilon$ model requires a grid with 300×360 finite volume cells. On the other hand, the local SU $k-\omega$ model requires a grid with 300×400 cells. A distance of 1×10^{-6} m of the first cell center from the wall is sufficient to give a grid converged solution for all the cases. This gives a y_2^+ that never goes above 0.67 for all the turbulence models and the cases considered.

4.9.1 CFD implementation

Implementation of the shock-unsteadiness model for the cases involving oblique shock wave impinging on a turbulent boundary layer over a flat plate is more complex than the earlier case of two-dimensional compression corner. This is because of the presence of a number of intersecting shock waves in the interaction region; see Figure 4.16 taken from [150]. Obtaining the local variation of shock-normal Mach number for all these shock-waves is a challenging task. The methodology adopted in the work of [35] for the 14° shock generator angle is described here. At first, preliminary computations are performed and the different shock waves in the flow-field are identified using the normalized mean dilatation, as shown in Figure 4.16. The values of upstream mean Mach number at different shocks are calculated manually by identifying an upstream location along the local streamline. The upstream Mach number values, in conjunction with the

local shock-normal direction, give the value of M_{1n} at grid points in the shock wave. Now, using the values of M_{1n} to obtain the exact variation of b'_1 upstream of all these shock waves and incorporating it into the CFD code is a cumbersome task. Therefore, a representative average value of M_{1n} , and hence that of b'_1 is used for the computations.

An alternate method involves identification of the upstream state by taking a maximum of Equation (4.43) for each finite volume cell and its neighbors. Then, distributing this maximum to the cell and its neighbors should give the value of M_{1n} . However, such method may not work well in case of multiple shock waves and their interactions, like the case of incident oblique shock impinging on a turbulent boundary considered here. Also, the number of neighbour cells taken for the approximation may also affect the accuracy of the predictions.

By comparison, the local form of the model is based on a constant value of the shock-unsteadiness model parameter c , and hence does not need the variation of the upstream shock-normal Mach number. Preliminary computations to identify the different shock waves in the flow are, therefore, not required for the local SU model. This reduces the computational effort significantly. Further, we save on human effort involved in graphically identifying the local streamline and manually obtaining the value of M_{1n} at different points along shock waves.

4.9.2 Comparison with experiment

14° case

Figure 4.17 presents the normalized pressure contours along with the contours of the shock detecting function f_s in the computed flow-field obtained using the local SU $k-\epsilon$ model. The detailed shock structures along with the limiting streamline representing the separation bubble can be clearly seen in figure 4.17(a), where **S** and **R** represent the separation and reattachment points, respectively. The shock detecting function f_s , which is directly taken from [35], is also shown to capture all the shocks in the flow. The region of non-zero f_s represents the region in the flow field where the local SU $k-\epsilon$ model is active.

Figures 4.18 and 4.19 show the comparisons of wall pressure and skin friction coefficient on the flat plate obtained using the different variants of $k-\epsilon$ and $k-\omega$ models with that of the experimental data of [93]. As can be seen from both the figures, the results for the local SU $k-\epsilon$ and $k-\omega$ models match well with that of the results obtained using their respective non-local models of [34].

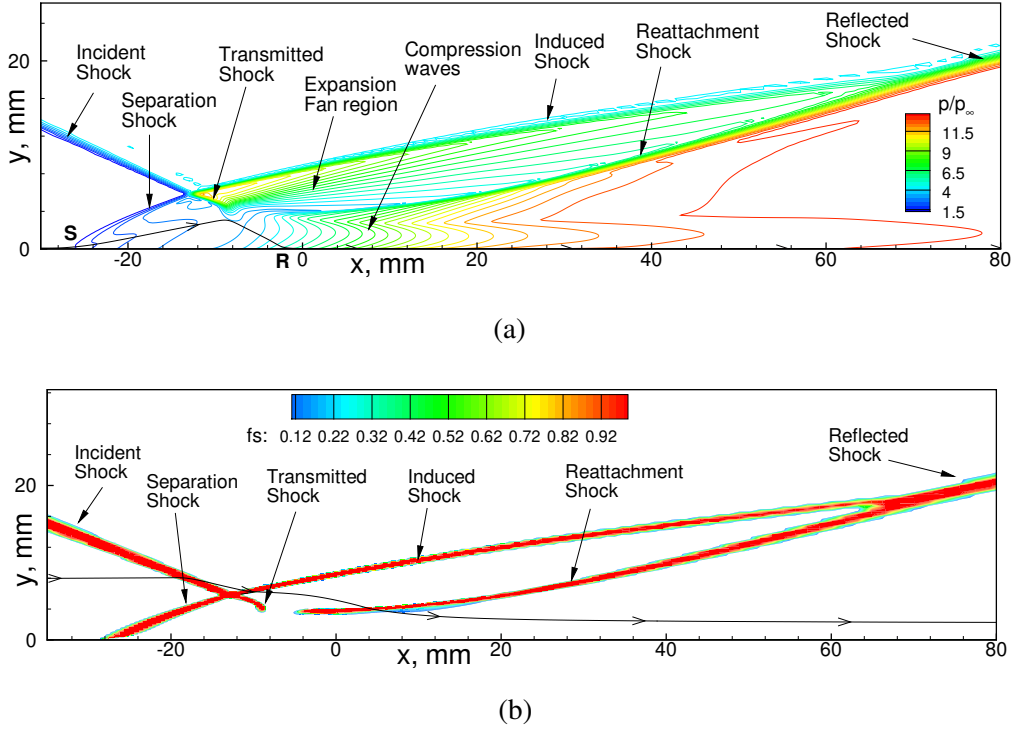


Figure 4.17: (a) Normalized pressure and (b) shock detecting function f_s contours computed using the local shock-unsteadiness $k-\epsilon$ model for 14° shock generator angle.

The surface pressure plot in figure 4.18 shows that both the standard $k-\epsilon$ and $k-\omega$ models predict the location of the separation shock downstream of the experiments. The pressure plateau, as well as the total pressure rise, is also underpredicted as compared to the experimental data. There is also a mismatch in the pressure rise prediction after the reattachment point when compared with the experimental data. On the other hand, the shock-unsteadiness modified $k-\epsilon$ and $k-\omega$ models give improved wall pressure predictions and shift the separation shock location closer to the experiment. The total pressure rise is underpredicted by the SU $k-\epsilon$ models, but the pressure rise after the reattachment point matches quite well with the experiment. The SU $k-\omega$ models give an excellent match with the experimental data for the pressure variation after the reattachment point.

Figure 4.19 shows that both the standard $k-\epsilon$ and $k-\omega$ models predict a later separation than the experiment, and highly underpredict the separation bubble size, as indicated by the length for which $C_f < 0$. The C_f values after the reattachment point are also not in agreement with the experimental data for both the standard models. The shock-unsteadiness modification improves the predictions of the separation point and reduces C_f in the recovery region post-reattachment. The SU $k-\omega$ model matches the experimental C_f variation up to about $x = 15$ mm, and asymptotes to a constant value further downstream. The standard $k-\omega$ predictions appear to be closer to the experimental data at

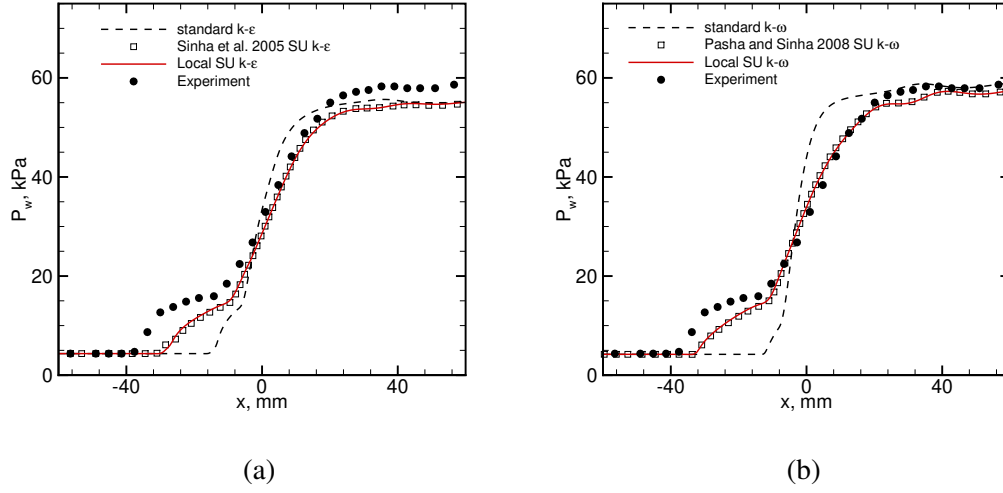


Figure 4.18: Comparison of wall pressure as obtained using the different variants of (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 14° case.

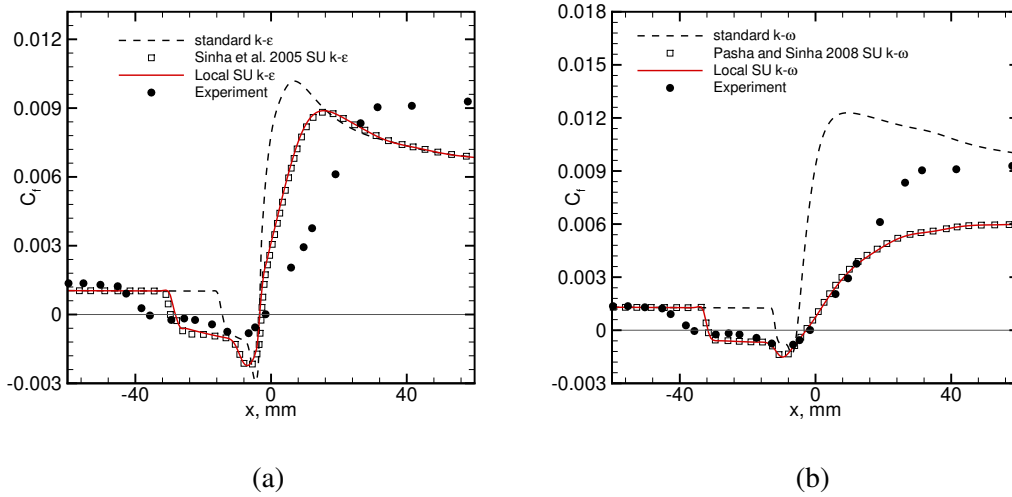


Figure 4.19: Comparison of skin friction coefficient as obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 14° case.

$x \approx 50$ mm, but computations with a longer domain [35] show that the standard model predictions eventually decrease to values comparable to the SU $k-\omega$ model.

10° case

The simulation results for the wall pressure and skin friction coefficient for the 10° case obtained using the different variants of $k-\epsilon$ and $k-\omega$ models are shown in figures 4.20 and 4.21. For this case also, the local forms of the SU $k-\epsilon$ and $k-\omega$ models give almost identical results as that obtained using the corresponding non-local SU models in [35].

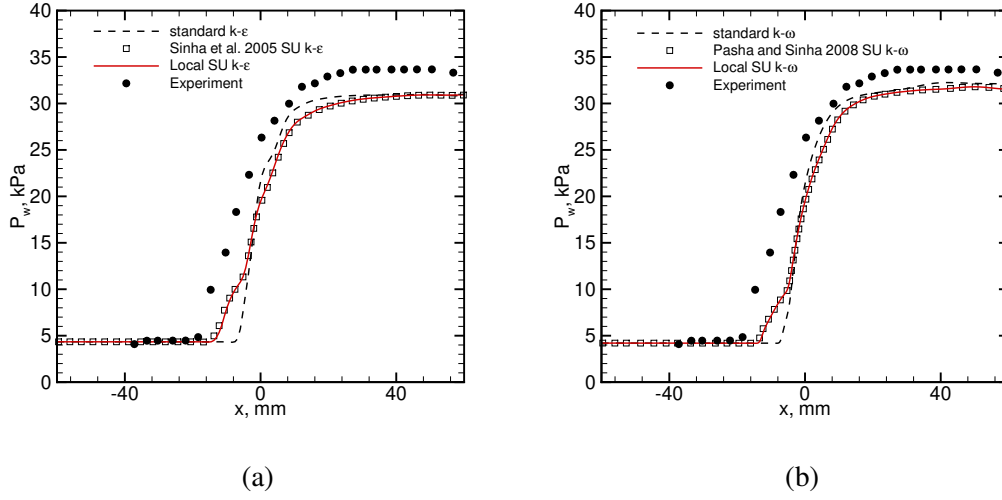


Figure 4.20: Comparison of wall pressure as obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 10° case.

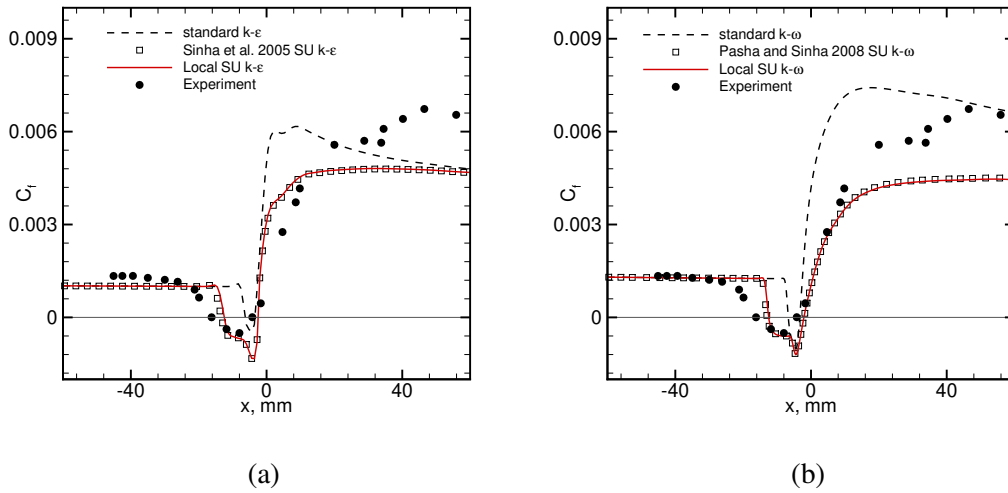


Figure 4.21: Comparison of skin friction coefficient as obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 10° case.

Comparison with experimental data follows the trend described in the 14° case. Specifically, a very small separation bubble ($C_f < 0$) and a later pressure rise is predicted by the standard models. As a result, the separation and reattachment shocks in the numerical solutions are not distinctly visible. By comparison, the solution obtained by the shock-unsteadiness models predicts a sizable separation bubble and the shock topology (similar to that shown for the 14° case) matches the experimental schlieren image closely.

Note that the size of the separation bubble, which is equivalent to the extent of upstream influence of the interaction, decreases with a reduction in the shock strength. This

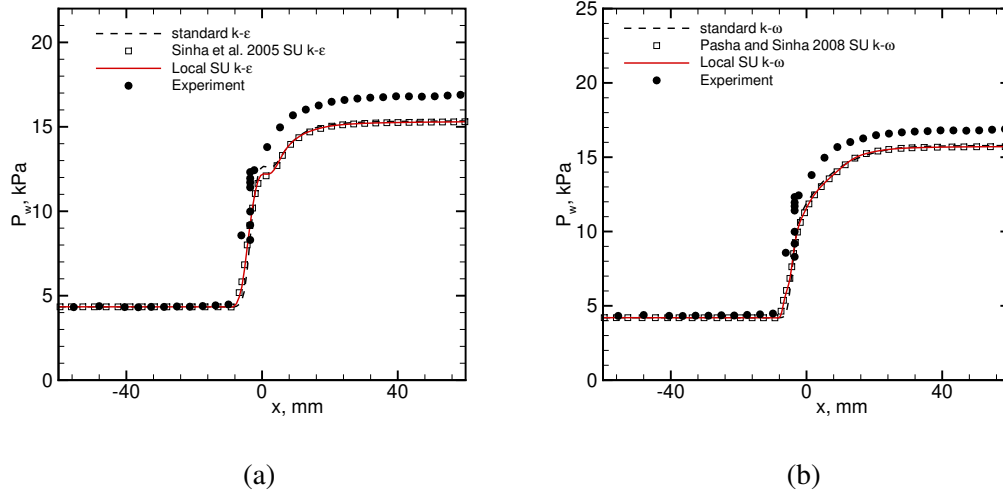


Figure 4.22: Comparison of wall pressure as obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 6° case.

is physically expected, and is visible in the experimental data for both the compression ramp and oblique shock impingement cases. The numerical results obtained by the standard $k-\epsilon$ and $k-\omega$ models follow this trend. The same is true for the shock-unsteadiness modified $k-\epsilon$ and $k-\omega$ models. It is particularly interesting to note that the effect of the shock-unsteadiness correction, as judged by the difference between the separation bubble size between the standard models and the corresponding SU models, also decreases with shock strength. This is physically consistent with the fact that the shock-unsteadiness correction is proportional to the mean dilatation (non-local SU) and the mean pressure gradient (local SU) across the shock. It is expected to vanish in the limit of Mach number decreasing to unity.

6° case

For the 6° case, boundary layer separation was not identified in the experimental measurements; the shock structure reported in the schlieren image [93] is akin to the inviscid reflection of the incident oblique shock wave. On the other hand, the shock-unsteadiness turbulence models predict incipient separation. Once again, the local form of shock-unsteadiness model results are able to match the non-local SU model predictions, while making the model implementation more robust. There are no major differences between the standard and the shock-unsteadiness $k-\omega$ models for the wall pressure profile (See figure 4.22(b)), the SU $k-\omega$ models show better predictions for C_f after $x = 0$ mm and the results match well with the limited experimental data available (See figure 4.23(b)). The

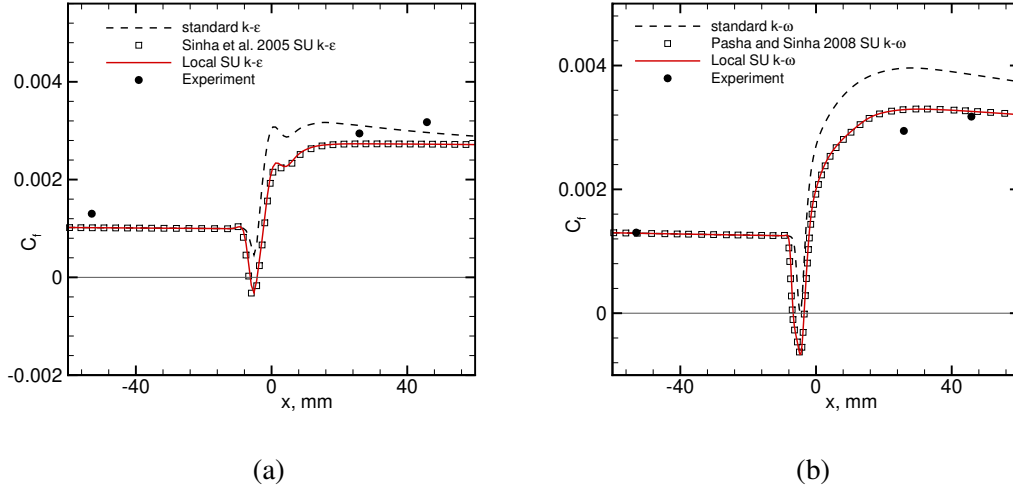


Figure 4.23: Comparison of skin friction coefficient as obtained using the different variants of the (a) $k-\epsilon$ and (b) $k-\omega$ models with the experimental data for 6° case.

SU $k-\epsilon$ model results (figures 4.22(a) and 4.23(a)) show marginal flow separation that is smaller in size when compared with the SU $k-\omega$ results.

4.10 Comparison of the computational time required

The local form of the shock-unsteadiness model is numerically robust and easy to use compared to the non-local shock-unsteadiness models used in literature. It can also lead to a significant saving in the computational effort. A comparison of the computational time required to reach a steady-state solution using the different variants of the $k-\epsilon$ and $k-\omega$ models, including the standard, local SU, and non-local SU versions, is shown in Table 4.3. We use the Mach 5 oblique shock impingement cases with shock generator angles of 14° and 10° . These are computationally more intensive than the supersonic compression corner flows.

The data in Table 4.3 suggests that the non-local SU $k-\epsilon$ and $k-\omega$ models require almost twice the number of time-steps, for both the cases of 14° and 10° , in comparison with the corresponding local SU models. This is because, the non-local SU model requires the information of M_{1n} , which is obtained by performing initial computations with the standard model. The converged solution of the standard model is then used as the starting point for the simulation using the non-local SU model. On the contrary, the local SU model can be directly used for performing the computations, starting from a solution initialized to freestream conditions. For example, both the local and non-local SU $k-\epsilon$ models require 45,000 time-steps to converge for the 14° case, with comparable

Table 4.3: Comparison of the computational time (on 32 processors) required by the different variants of $k-\epsilon$ and $k-\omega$ turbulence models for the 14° and 10° incident oblique shock wave turbulent boundary layer interaction cases.

Turbulence model		Shock generator angle, β	Time-steps required (in thousands)	Computing time per iteration (in milliseconds)	Total Computing time (in mins.)
$k-\epsilon$	Standard	14°	40	59.7	39.78
	Non-local SU		40 + 45	60	84.96
	Local SU		45	60.4	45.28
$k-\omega$	Standard	14°	6	65	6.5
	Non-local SU		6 + 8	65.1	15.2
	Local SU		8	65.2	8.7
$k-\epsilon$	Standard	10°	17	59.8	16.94
	Non-local SU		17 + 18	60.6	35.34
	Local SU		20	61.3	20.42
$k-\omega$	Standard	10°	6	65	6.5
	Non-local SU		6 + 4	65.4	10.9
	Local SU		6	66	6.6

values of the Courant-Friedrichs-Lewy (CFL) number. However, the non-local SU model computation requires additional 40,000 time-steps to get the standard $k-\epsilon$ solution that is used as the initial condition for the subsequent application of the SU correction. Similar observations can be made for the 10° case, as well as the computations with the different $k-\omega$ models for both the shock generator angles.

In terms of the computational time required per time-step, all the variants of the models are comparable, with the shock-unsteadiness models being marginally more expensive than the respective standard $k-\epsilon$ and $k-\omega$ models. This is because of the additional computations performed to evaluate the shock detecting function f_s and the shock-unsteadiness damping parameters. Thus, the total time required to reach a steady-state solution for the local SU $k-\epsilon$ and $k-\omega$ models are only 15-20% higher than the corresponding standard models. On the other hand, the non-local SU models require about twice the time of the

standard model for both the test cases. The local SU models, therefore, result in almost 50% savings of the computational time for the 14° and 10° cases, respectively, in comparison with its non-local counterpart. This saving in the computational time comes without affecting the accuracy of the results obtained. Both the local and non-local models are based on identical shock-physics and give almost identical values of surface pressure and skin-friction coefficient, as shown in the previous sub-sections.

A comparison between the computational efficacy of the $k-\epsilon$ and $k-\omega$ family of models shows that the different versions of the $k-\omega$ model require significantly lesser computational time as compared to the corresponding versions of the $k-\epsilon$ model. This is because of the higher CFL numbers that the $k-\omega$ models can handle in comparison with the $k-\epsilon$ models. Note that the time required per time-step is comparable for the two models. The $k-\epsilon$ model equations are relatively stiff and are difficult to converge in comparison with the $k-\omega$ model. This can be seen from the Table 4.3; the standard $k-\epsilon$ model requires about 40,000 iterations for convergence in comparison with the 6000 iterations required by the standard $k-\omega$ model, for the 14° case. Similar observations can be made for the 10° case. Another thing to note here is that, as the shock strength increases from the 10° case to the 14° case, convergence of the $k-\epsilon$ models is significantly affected, which is not the case with the $k-\omega$ models.

4.11 Summary

This chapter describes the canonical STI problem in detail. The development of a physically sound and numerically robust form of shock-unsteadiness (SU) model, called as local SU model, for application to high-speed turbulent flows with shock waves is presented. A new physically-consistent source term is proposed to model the shock-unsteadiness damping of turbulent kinetic energy. The turbulent dissipation rate equation is based on the physical effects of mean compression and enhancement of fluid viscosity across a shock wave. Unlike previous physics-based turbulence models, the current model parameters are independent of the shock strength and hence do not require prior knowledge of the upstream mean flow Mach number. This can be of significant advantage while applying the model equations to realistic flows with multiple shock waves and their interaction with boundary layers. The model equations are also cast in a conservative form so as to eliminate numerical error at flow discontinuities. The model is tested extensively in canonical shock-turbulence interaction and found to match available DNS data well over a range of Mach numbers. The superior numerical characteristics, in terms of grid convergence of the solution, is also demonstrated. The local SU model is then systematically

extended to realistic flows in the standard k - ϵ and k - ω model frameworks. Application of the model to a range of SBLI cases in the supersonic and hypersonic regime with attached to fully separated flows is also successfully demonstrated.

Chapter 5

Development of an anisotropic SST turbulence model for shock-dominated flows

In this chapter, we start with the SST model of Menter and propose modifications to include non-linear Reynolds stress-strain relation and shock-physics based correction applicable to shock-dominated turbulent flows. The model development strategy followed here is not intended to formulate a new non-linear constitutive stress-stress model but to use an already existing relationship in the SST model framework. The modelling process is not specific to the choice of the non-linear stress-strain relationship and can be used with any existing non-linear constitutive models. Major focus is to get a correct representation of the structure parameter behaviour in the shock/turbulence interaction regions along with correct prediction of anisotropy in normal Reynolds stresses. During the model development, care is taken to retain the boundary layer predicting capabilities of the standard SST model.

5.1 Menter SST k - ω model

The family of k - ϵ and k - ω two-equation eddy viscosity turbulence models are generally the most popular choices for the engineering predictions of flows occurring in aerospace-related applications. The standard k - ω model is known to perform better for boundary layer flows, however the standard k - ϵ model predicts the spreading rates more accurately in case of free-shear layers. Also, it is known that standard k - ω model has a very strong sensitivity to the arbitrary freestream values specified for ω outside the boundary layer [37, 38]. The turbulent eddy viscosity in boundary layers and freeshear layers can be

changed by more than 100% by simply reducing the freestream value of ω . The standard k - ϵ model does not suffer from this deficiency. Thus, a desired choice of a turbulence model would be to use the standard k - ω model near the wall in the boundary layer and standard k - ω model away from the wall and in the free-shear layers. Menter SST k - ω model is based on this idea and the transport equations for this model are a combination of the standard k - ϵ and k - ω models in a k - ω modelling framework. The model equations are shown in Chapter 2 and the derivation of the transport equations is shown in Appendix C.

Menter SST model employs a linear relation between the Reynolds stress and the mean strain-rate tensors known as Boussinesq model, as discussed in Chapter 2. Menter's analysis showed that the ability to correctly model adverse pressure gradient boundary layer flows and separation largely depends on the constitutive model, and the traditional linear Boussinesq model is not sufficient for this purpose. Menter developed an extension of the Boussinesq model that prevents the shear stress anisotropy from exceeding a certain limit. The SST model is not a non-linear model in the tensorial sense, i.e., it relies on a tensorially linear relation between Reynolds stress and the strain rate tensors. However, the coefficient of this relationship, i.e. eddy viscosity μ_T , depends on a vorticity invariant as shown below,

$$\mu_T = \frac{\bar{\rho} a_1 k}{\max(a_1 \omega, \Omega F_2)} = \begin{cases} \bar{\rho} k / \omega & \text{Standard form} \\ \bar{\rho} a_1 k / \Omega & \text{SST modified form} \end{cases} \quad (5.1)$$

The function F_2 switches eddy viscosity smoothly between the SST modified form active inside boundary layers and the standard form everywhere else. Here, $\Omega = \sqrt{-2II_\Omega}$ is the vorticity magnitude with the second invariant of vorticity given by $II_\Omega = \Omega_{ij}\Omega_{ji}$ and vorticity $\Omega_{ij} = (1/2)(\partial \tilde{u}_i / \partial x_j - \partial \tilde{u}_j / \partial x_i)$. Despite the modification to the definition of μ_T , the linear stress-strain relation cannot reproduce anisotropy in Reynolds stresses, especially in case of SBLIs wherein anisotropy is a key feature [71, 95]. On account of this, for the new model development, the Boussinesq linear stress-strain relation is discarded and instead a non-linear stress-strain constitutive model is used in the standard SST model. This is done without changing the model coefficients of the original SST model.

5.2 Quadratic eddy viscosity formulation

Previous studies have shown that the range of applicability of the two-equation eddy viscosity models can be extended with the use of more advanced constitutive models employing non-linear relations between the Reynolds stress and the mean velocity gradient and the turbulence scales. In fact, a quadratic stress-strain relationship is atleast required

to obtain Reynolds stress anisotropy and to reproduce the effects arising because of it. EARSMs are an interesting and promising modelling avenues which provide non-linear stress-strain relationships. In this approach, part of the higher-order description of physical processes on the Reynolds stress transport model level is transferred onto the two-equation modelling level. EARSM approach is considered to be a suitable type of constitutive modelling for the present work in order to reproduce Reynolds stress anisotropy correctly.

Following Wallin and Johansson [70], a general relationship between Reynolds stress and mean strain-rate and vorticity tensors can be written in terms of an effective turbulent viscosity and an extra anisotropy term as,

$$\tau_{ij} = 2\mu_T S_{ij} - \frac{2}{3}\rho k \delta_{ij} - \rho k a_{ij}^{(ex)} \quad (5.2)$$

In this work, we use the quadratic EARSM of Rung et al. [73] for which the extra anisotropy tensor is given by,

$$a_{ij}^{(ex)} = -2C_\mu^* \tau^2 \left[\beta_2' (S_{ik} \Omega_{kj} - \Omega_{ik} S_{kj}) - \beta_3' \left(S_{ik} S_{kj} - \frac{1}{3} S_{kl} S_{kl} \delta_{ij} \right) \right] \quad (5.3)$$

where $\tau = 1/(C_\mu \omega)$ is the turbulence time scale and $C_\mu = 0.09$. The choice of the EARSM is made based on the model's simplicity and ability to correctly predict anisotropy in normal Reynolds stresses in case of non-equilibrium compressible flows. Eddy-viscosity for this EARSM is similar to the standard form of μ_T for a linear eddy viscosity based k - ω model in Eqn. (5.1) with an additional factor of C_μ^*/C_μ multiplied to it,

$$\mu_T = \frac{C_\mu^* \rho k}{C_\mu \omega} \quad (5.4)$$

where C_μ^* is the anisotropy parameter and it is a function of invariants of the mean strain-rate and vorticity tensors. The model parameters and model constants are given in Chapter 2.

5.3 Log layer analysis of EARSM

Log layer is defined as the portion of the boundary layer sufficiently distant from the surface that molecular viscosity is negligible relative to eddy viscosity, yet close enough for convective effects to be negligible [57]. Neglecting the convection and pressure gradient (zero-pressure-gradient) and the molecular viscosity relative to the eddy viscosity, we have from the x -momentum equation,

$$\frac{\partial}{\partial y} \left[\mu_T \frac{\partial \tilde{u}}{\partial y} \right] = 0 \quad (5.5)$$

Integrating across the log layer thickness, we get

$$\mu_T \frac{\partial \tilde{u}}{\partial y} = \tau_{xy} = \text{constant} \quad (5.6)$$

As Bradshaw's relation hold good in a log layer and noting that Reynolds shear stress is the principal contributor to TKE production, we have

$$\tau_{xy} = \bar{\rho} a_1 k \quad (5.7)$$

Equating Eqns. (5.6) and (5.7), we get

$$\frac{\partial \tilde{u}}{\partial y} = \frac{\bar{\rho} a_1 k}{\mu_T} \quad (5.8)$$

Now, for the EARSIM, eddy viscosity is given by Eqn. (5.4) and substituting it in Eqn. (5.8) we get,

$$\frac{\partial \tilde{u}}{\partial y} = \frac{C_\mu}{C_\mu^*} a_1 \omega \quad (5.9)$$

We generalize the velocity gradient in the above equation in terms of the magnitude of mean velocity gradients $D = \sqrt{(\partial \tilde{u}_i / \partial x_j)(\partial \tilde{u}_i / \partial x_j)}$ and noting that $\tau = 1/(C_\mu \omega)$, we get

$$a_1 = C_\mu^* \tau D \quad (5.10)$$

5.4 Modified Structure parameter

Menter [37] assumes a general linear relationship for the principal Reynolds stress, $\tau_{principal} = \mu_T \Omega$, where the vorticity magnitude represents the mean velocity gradients. This is equated to Bradshaw's assumption,

$$\tau_{principal} = \bar{\rho} a_1 k \quad (5.11)$$

which states that principal turbulent stress is proportional to turbulence kinetic energy in a turbulent boundary layer, to arrive at the SST modification in Eqn. (5.1). Here, the structure parameter a_1 assumes a constant value of 0.31.

We follow a strategy similar to Menter and assume the following quadratic relation between the principal stress and mean velocity gradient,

$$\tau_{principal} = \mu_T D + C_\mu^* \bar{\rho} k (\tau D)^2 (\beta_2' - \beta_3') \quad (5.12)$$

Here, $D = \sqrt{(\partial \tilde{u}_i / \partial x_j)(\partial \tilde{u}_i / \partial x_j)} = \sqrt{(S'^2 + \Omega^2)/2}$ is the magnitude of deformation rate tensor $\partial \tilde{u}_i / \partial x_j$ and $S' = \sqrt{2S'_{ij}S'_{ij}}$ with $S'_{ij} = (1/2)(\partial \tilde{u}_i / \partial x_j + \partial \tilde{u}_j / \partial x_i)$. Note that, we have chosen to use the magnitude of deformation rate tensor as against the vorticity magnitude

Ω used by Menter. In this way, we are retaining the physics incumbent in the vorticity magnitude in the principal stress as in Menter's proposal. Moreover, it ensures that the dominant nature of the strain-rate is captured in the vicinity of shock waves.

Equating the above equation to Bradshaw's relation (Eqn. (5.11)), gives us

$$a'_1 = C_\mu^* \tau D \left[1 - \tau D (\beta'_2 - \beta'_3) \right] \quad (5.13)$$

where we use $a_1 = C_\mu^* \tau D$ obtained from a log-layer analysis of the EARSM equations for an equilibrium zero-pressure-gradient turbulent boundary layer (shown in Section 5.3). Thus, the structure parameter for this model is a function of the mean velocity field in the form of the magnitude of the deformation-rate tensor and, unlike the SST model, is not a constant. The anisotropy parameter C_μ^* also appears in the above equation and it responds to the changes in strain rate and vorticity magnitude.

It is interesting to note that the constant value 0.31 of the structure parameter used in SST model is based on incompressible equilibrium boundary layer data. However, it is also found to be a good approximation in adverse-pressure-gradient subsonic and transonic boundary layers. For incompressible high Reynolds number flows with Re_θ (θ being the momentum thickness) in the range of 2573 – 57720, Smits and Dussauge [45] found that a_1 lies in the range of 0.28 – 0.34. Experimental [47] and DNS [46] studies of compressible zero-pressure-gradient boundary layer also found that a_1 assumes values similar to incompressible flows. However, in case of non-equilibrium compressible flows, such as SBLIs, experiments show that a_1 is higher than 0.31 in the interaction region and the increase is significant.

Rose and Murphy [47] perform experiments for an impinging shock configuration (with a free stream Mach number around 4 and $Re_\delta = 8.7 \times 10^4$) to calculate a_1 . The experiments involve near incipient shock-induced separation. Measurements reveal that the structure parameter increases to a value as high as 0.44 in the interaction region. UFAST SBLI impinging shock experiments [51, 153] with Mach 2.3 flow also show that a_1 reaches a value as high as 0.4 in the SBLI region. Ardonneau [165] and Smits and Muck [48] make similar observations for a series of compression corner SBLI experiments. Thus, there is a definite evidence that the physics of compressible flows with adverse pressure gradient is significantly different than that of incompressible separated flows. As a matter of fact, in an incompressible boundary layer, a_1 decreases in response to adverse pressure gradient, as observed by Bradshaw [166], whereas it increases in SBLI flows [50].

5.5 Analysis at a shock wave : SU limiter

The structure parameter given in Eqn. (5.13) is a function of the deformation-rate tensor and hence strain-rate, which can take large numerical values at a shock wave. In fact, the strain rate can increase with grid refinement as $1/\Delta$, where Δ is the grid resolution at a shock wave. As a result, the structure parameter can assume unphysical values. So, we use the physics of shock-turbulence interaction to prescribe possible limiting values of the effective structure parameter at a shock wave. This follows from a physical manner than the empirical stress limiter C_{lim} found in the literature [54]. The following analysis is based on Sinha et al. [30] shock-unsteadiness (SU) model.

It is shown that the standard form of TKE production grossly overpredicts turbulence amplification at a shock wave. The shock-unsteadiness model alters the production term based on the damping effect of an unsteady shock wave, and the results match DNS data for canonical shock-turbulence interaction. Shock-unsteadiness model's production term is given by

$$P'_k = -\frac{2}{3}\bar{\rho}k(1 - b'_1)S_{ii} = \tau_{eq}|S_{ii}| \quad (5.14)$$

where S_{ii} is the mean dilatation, and b'_1 is the shock unsteadiness parameter described below. The production of TKE in Eqn. (5.14) is written in terms of the magnitude of dilatation and an equivalent Reynolds stress,

$$\tau_{eq} = (2/3)\bar{\rho}k(1 - b'_1) = s_u\bar{\rho}a_1k \quad (5.15)$$

which brings in the effect of unsteady shock oscillations in response to incoming turbulent fluctuations. Thus, the equivalent stress τ_{eq} acts on the dilatation S_{ii} and results in the correct production of TKE at a shock wave. The equivalent stress τ_{eq} is proportional to TKE at a shock wave, with $s_u = \frac{2}{3}(1 - b'_1)/a_1$. This is similar to Bradshaw's relation given in Eqn. (5.11). However, at a shock wave the equivalent stress $\tau_{eq} > \bar{\rho}a_1k$, as $s_u > 1$. Here, the shock-unsteadiness damping parameter $b'_1 = 0.4(1 - e^{1-M_{1n}})$ is a function of upstream shock-normal Mach number M_{1n} .

We now convert the equivalent stress τ_{eq} into an effective eddy viscosity to get,

$$\mu_T = \frac{\bar{\rho}a_1''k}{D} \quad \text{with} \quad a_1'' = s_u a_1 \quad (5.16)$$

which has a form similar to the SST modified eddy viscosity in Eqn. (5.1), with a modified structure parameter at a shock wave. Figure 5.1 shows the variation of s_u and a_1'' for a range of Mach numbers. The shock-unsteadiness model parameter s_u assumes a value of about 2 for weak shock waves and asymptotes to a value of 1.29 for $M_{1n} \rightarrow \infty$. This means that the modified structure parameter a_1'' varies from 0.62 for Mach 1 to 0.4 for

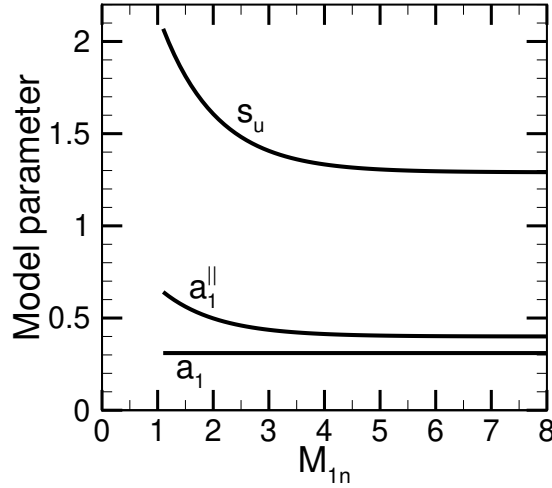


Figure 5.1: Variation of the model parameters s_u and a_1'' with upstream shock-normal Mach number M_{1n}

hypersonic flows. We choose $a_1'' = 0.4$ as the limiting value of the structure parameter at a shock wave, so as to meet the constraint for the entire range of Mach numbers. A constant value of a_1'' makes the model implementation simple, by avoiding a model parameter that depends on the upstream Mach number.

Wilcox 2006 k - ω model uses the effective structure parameter to be 0.343 and gives better results for SBLI flows than $a_1 = 0.31$ [57]. Several numerical studies using SST model [52–55] also found that using a value of the structure parameter higher than its standard value of 0.31 gives improved results for SBLI flows; however all these studies increase the structure parameter in an ad-hoc manner. Our model brings in a similar effect, but in a more formal way that is based on physical analysis at a shock wave. The current model thus differs from the ad-hoc approaches found in the literature, which use empirical eddy-viscosity limiters.

5.6 A new shear stress transport model

We now combine the developments of the previous sub-sections to present the final form of the SST model for SBLI flows. There are two key developments. First, the SST model is augmented by a quadratic relation for Reynolds stresses taken from EARSIM. This leads to a modified structure parameter a_1' in Eqn. (5.13). This is combined with the SU-modified eddy viscosity at a shock wave given in Eqn. (5.16). We thus get an eddy viscosity model applicable to turbulent boundary layers and at shock waves,

$$\mu_T = \frac{\bar{\rho} a_1^* k}{D} = \frac{\bar{\rho} a_1 k}{D(a_1/a_1^*)} \quad \text{with} \quad a_1^* = \min[a_1', a_1''] \quad (5.17)$$

This form is similar to the SST-modified μ_T in Eqn. (5.1), with a_1 replaced by the effective structure parameter a_1^* .

Obviously, Eqn. (5.17) is not desirable for the complete flowfield, since it leads to infinitely high eddy-viscosities at points where D goes to zero. So, it is supplemented with the EARSM eddy viscosity given in Eqn. (5.4) and rewritten here in a slightly different form as,

$$\mu_T = \frac{\bar{\rho} a_1 k}{(C_\mu^*/C_\mu) a_1 \omega} \quad (5.18)$$

To guarantee the selection of desired forms of the eddy viscosity in appropriate regions of the flowfield, we combine Eqns. (5.17) and (5.18) in the following manner,

$$\mu_T = \frac{\bar{\rho} a_1 k}{\max[(1 - F'_2)(C_\mu/C_\mu^*) a_1 \omega, F'_2 D(a_1/a_1^*)]} \quad (5.19)$$

similar to Eqn. (5.1) in the original SST model. Here, F'_2 is a function which identifies turbulent boundary layers and shock waves, where it assumes a value of 1, and it is identically zero in rest of the flowfield. Thus, Eqn. (5.19) is formulated such that it switches to Eqn. (5.17) inside a boundary layer and in the regions of shock waves, and reverts back to Eqn. (5.18) elsewhere. The modified eddy viscosity given in Eqn. (5.19) together with the quadratic Reynolds stress-strain relation given in Eqns. (5.2) and (5.3) together with the k and ω transport equations of the standard SST model is called as the SUQ-SST model.

As discussed earlier, the modelling strategy followed here can be easily used with any other NLVEM or EARSM in the SST model framework. For example, if EARSM by Wallin and Johansson [70] is considered for the model development instead of EARSM by Rung et al. [73], then the resulting changes in the model equations are discussed below. For a general three-dimensional mean flow, WJ EARSM has a quartic expansion of the Reynolds stresses, i.e., Eqn. (5.3) for the extra-anisotropy will have additional contributions in terms of third and fourth order powers of strain and vorticity tensors. This will then translate to a quartic relation for the structure parameter in Eqn. (5.13) instead of a quadratic one. However, rest of the equations in the new model i.e. Eqns (5.17) and (5.19) will have identical forms as the current proposed model using EARSM by Rung et al [73].

5.7 The blending function F'_2

The function F'_2 is given by,

$$F'_2 = \max[F_2, f_s] \quad (5.20)$$

where F_2 is the standard function used in Menter's SST model. It makes sure that the SST correction is active only inside a turbulent boundary layer. By comparison, the function f_s detects regions of shock waves and following form is proposed in this work,

$$\begin{aligned} arg &= 50 \frac{S_{ii}}{\tau_s} + 5 \\ \tau_s &= \frac{a}{L_s}, \quad L_s = \max(\Delta, L_\epsilon), \quad L_\epsilon = \frac{0.25 \sqrt{k}}{0.09\omega} \\ f_s &= \frac{1}{2} - \frac{1}{2} \tanh(arg) \end{aligned} \quad (5.21)$$

where, a is the local speed of sound. Here, Δ is the local representative finite-volume cell size. For structured Cartesian grids, Δ can be simply calculated as $\Delta = \sqrt{\Delta x \Delta y}$, where Δx and Δy are the local grid spacings in the x and y -direction, respectively.

The function f_s smoothly transitions to 1 in shock regions from a value of zero outside shock waves. In Eqn. (5.21), we use the mean dilatation S_{ii} normalized by a suitable time scale τ_s for shock detection. The time scale uses local representative grid size for detecting shock waves in the inviscid regions and the form is similar to that used in Bhagatwala and Lele [170]. On the other hand, for detecting shock waves penetrating inside a turbulent boundary layer, such as the separation shock foot in the impinging shock and compression corner flows described below, we use the turbulence dissipation length scale L_ϵ to determine τ_s .

Interestingly, different forms of the function f_s have been used in previous works [27, 34, 35] for computing different configurations of SBLI flows. However, a single form of shock function proposed here and given in Eqn. (5.21), which is algebraic in nature, is found suitable to correctly capture shock waves for all the high-speed flow configurations considered in this work. The function f_s proposed here requires local flow data and grid details and unlike the form in Pasha and Sinha [27, 35], it is independent of non-local flow properties, like the incoming boundary layer thickness, which may not be readily available.

5.8 Zero-pressure-gradient flat plate boundary layer

The newly developed SUQ-SST model is an extension of the standard SST model wherein a quadratic stress-strain relation is used along with a modified μ_T definition. However, despite these changes, the new SST model retains the boundary layer predicting capabilities of the original SST model. To demonstrate this, a zero-pressure gradient flat plate boundary layer is computed. Zhang et al. [167] presents a direct numerical simulation (DNS)

Table 5.1: Free stream and wall temperature conditions for the DNS case

M_∞	U_∞ , m/s	ρ_∞ , kg/m ³	T_∞ , K	T_w , K	T_w/T_r
2.5	823.6	0.1	270	568	1

database of high-speed zero-pressure-gradient turbulent boundary layers developing spatially over a flat plate with nominal freestream Mach number ranging from 2.5 to 14. These DNS cases fall within the perfect gas regime. In this work, the Mach 2.5 case is computed with freestream conditions of velocity $U_\infty = 823.6$ m/s, density $\rho_\infty = 0.1$ kg/m³, and temperature $T_\infty = 270$ K. The working fluid is air with viscosity calculated using Sutherland's law. The wall is assumed to be isothermal with a wall temperature of $T_w = 568$ K. DNS provides mean properties at a location, where boundary layer thickness $\delta = 7.7$ mm, displacement thickness $\delta^* = 2.38$ mm, momentum thickness $\theta = 0.58$ mm, and the Reynolds number based on momentum thickness is $Re_\theta = 2809.9$.

A structured cartesian mesh with 400×250 cells is used for the simulations with the grid points clustered at the plate tip and exponentially stretched away from it along the wall. Exponential stretching is also used in the wall-normal direction with the distance of the first cell centre of 1×10^{-6} m from the wall. At the solid wall, a no-slip wall boundary condition is employed, whereas at the top and exit boundaries, an extrapolation condition is used

The SUQ-SST model gives almost identical values of Reynolds shear stress as the standard SST model in a zero-pressure-gradient turbulent boundary layer. This is because the shear component of the quadratic part of Reynolds stress given in Eqn. (5.3) has a negligible contribution in an equilibrium boundary layer. Moreover, the modified structure parameter calculated using Eqn. (5.13) assumes values closer to 0.31 for most parts of the undisturbed boundary layer, thereby returning μ_T values similar to the standard SST model. Since, Reynolds shear stress principally dictates the growth and characteristics of a turbulent boundary layer, SUQ-SST model matches the standard SST model in resolving a turbulent boundary layer.

Figure 5.2(a) plots the comparison between DNS and the two SST models for the Van Driest transformed mean velocity defined as [168],

$$u_{VD}^+ = \frac{1}{u_\tau} \int_0^{\tilde{u}} \left(\frac{\bar{\rho}}{\bar{\rho}_w} \right)^{(1/2)} d\tilde{u} \quad (5.22)$$

where $u_\tau = \sqrt{\tau_w/\rho_w}$ is the friction velocity, with τ_w being the wall shear stress and ρ_w is the density at the wall. The comparisons are shown at a location with Re_θ of about

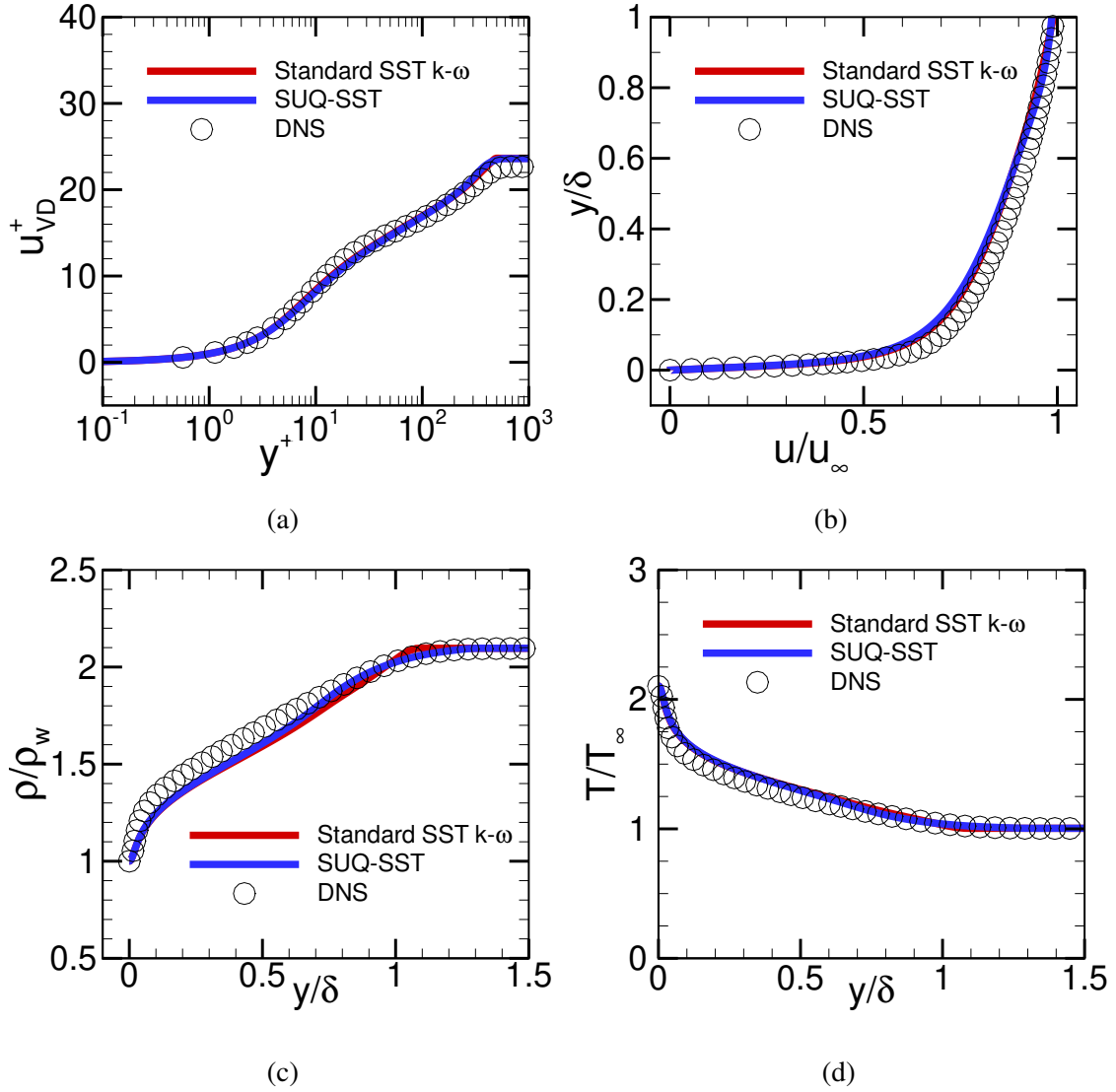


Figure 5.2: Comparison of (a) Van Driest transformed mean velocity, (b) streamwise mean velocity, (c) mean density, and (d) mean temperature obtained using the standard SST and SUQ-SST models with the DNS data.

2810. The SUQ-SST model gives almost identical result with the standard SST model and both match the DNS data well. This is also the case for the mean velocity, density, and temperature profiles in Figs. 5.2(b) - 5.2(d). This shows that modifications in terms of adding EARS features to the SST framework and calculating structure parameter using the new relation do not alter the boundary layer prediction capability of the SST model.

5.9 Summary

The chapter presents an advanced form of SST $k-\omega$ model for shock-dominated turbulent flows. The new model, called as SUQ-SST, is developed by introducing features of

EARSM in the SST framework. A quadratic relation between the Reynolds stresses and strain and vorticity tensors by Rung et al. [73] is used for this purpose. The structure parameter is modified and it depends on the local mean velocity gradients and the anisotropy parameter. The modified structure parameter increases in response to a shock-induced adverse pressure gradient, as observed in SBLI experiments. However, at a shock wave, mean velocity gradients can become significantly high that may lead to unphysical values of the structure parameter. To prevent this, the value of the structure parameter at a shock wave is limited using the shock-unsteadiness (SU) limiter. The limiting value is obtained from a Menter-like analysis of the shock-unsteadiness model of Sinha et al. [30] at a shock wave. The SU limiter is similar, but more formal and based on physical analysis, than the use of ad-hoc eddy-viscosity limiters found in the literature. Despite the modelling changes introduced to the standard SST model, the new model is shown to match the performance of the original SST model in resolving a high-speed compressible flat plate equilibrium turbulent boundary layer accurately.

Chapter 6

Application of SUQ-SST model to shock- turbulent boundary layer interaction

The standard SST model, the SUQ-SST model, and EARSM of Rung et al. [73] are applied to the SBLI test cases delineated in Chapter 3. This is to assess the performance of the SUQ-SST model in predicting shock/boundary layer interactions at different Mach numbers, for varying strength of interaction. Comparison is made with available experimental data, and with the predictions of existing models.

6.1 UFAST Mach 2.3 impinging shock case

The supersonic oblique incident/reflected shock case discussed in Section 3.3 of Chapter 3 is computed and results are compared with the available experimental data. The composite Fig. 6.1 shows the flow structure in an oblique shock impingement SBLI. Here,

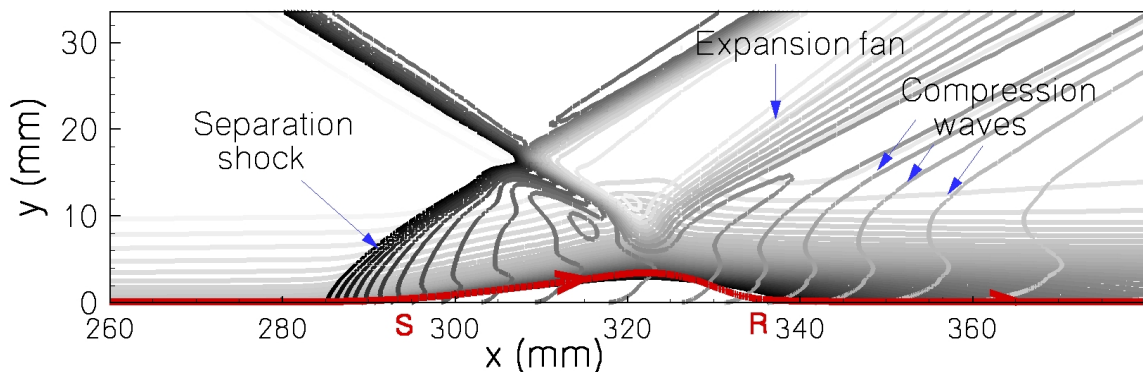


Figure 6.1: Contours of Mach number superposed upon normalized pressure to show the flow structure in the UFAST SBLI case

Mach number contours are used to show the boundary layer, whereas normalized pressure is used to depict the inviscid flow structures such as shock waves and expansion fan. The impinging shock in this case is strong enough to cause separation of the turbulent boundary layer. The limiting streamline identifies the separation **S** and reattachment **R** locations in Fig. 6.1. The separation bubble acts as a ramp and forms a separation shock. The flow turning over the separation bubble expands and eventually reattaches on the plate, forming compression waves that coalesce to form the reattachment shock.

6.1.1 Mean velocity and Reynolds stresses variations

Figure 6.2(a) shows the comparison of the computed and experimental velocity profiles at different streamwise positions. At the inflow plane for the SBLI simulations ($x = 260$ mm), all the turbulence models correctly capture the boundary layer and give excellent match with the experiment. The SUQ-SST model predicts the interaction location slightly upstream of the experimental data and hence shows more strongly decelerated boundary layer than the experiment at separation point ($x = 300$ mm). At this location, EARSM results are closer to the experimental data. Standard SST slightly under-predicts, and it is almost identical to the SSG/LRR- ω Reynolds stress model (RSM) predictions, taken from [95].

Inside the separation region ($x = 323$ mm), SUQ-SST gives the best match with experiment. Near wall region of the decelerated boundary layer is captured well by all the models. However, except for SUQ-SST, all other models show a strongly accelerated velocity profile away from the wall. A curved velocity profile with two inflection points is prominent in the SST, EARSM, and RSM results. On the other hand, SUQ-SST model gives a rather smooth velocity profile and matches well with the experiment. Similar characteristics can be found further downstream at $x = 340$ mm, with SUQ-SST model giving the best prediction.

Figures 6.2(b) and 6.2(c) plot the square root of the normal Reynolds stress components in the axial direction $(-\tau_{xx}/\rho)^{(1/2)}$ and in the wall-normal direction $(-\tau_{yy}/\rho)^{(1/2)}$. For the SST model, Boussinesq hypothesis which linearly relates the Reynolds stresses with strain-rate is used to compute the Reynolds stresses whereas a quadratic relationship between Reynolds stresses and strain-rate and vorticity tensors is used for the EARSM and SUQ-SST models. For RSM, Reynolds stresses are directly obtained by solving the individual Reynolds stress differential equations, as described in [95].

The SST model predicts similar levels of the normal Reynolds stress components at the corresponding axial locations. However, experimental data clearly shows higher levels of the streamwise Reynolds stress than the vertical component at each location. This

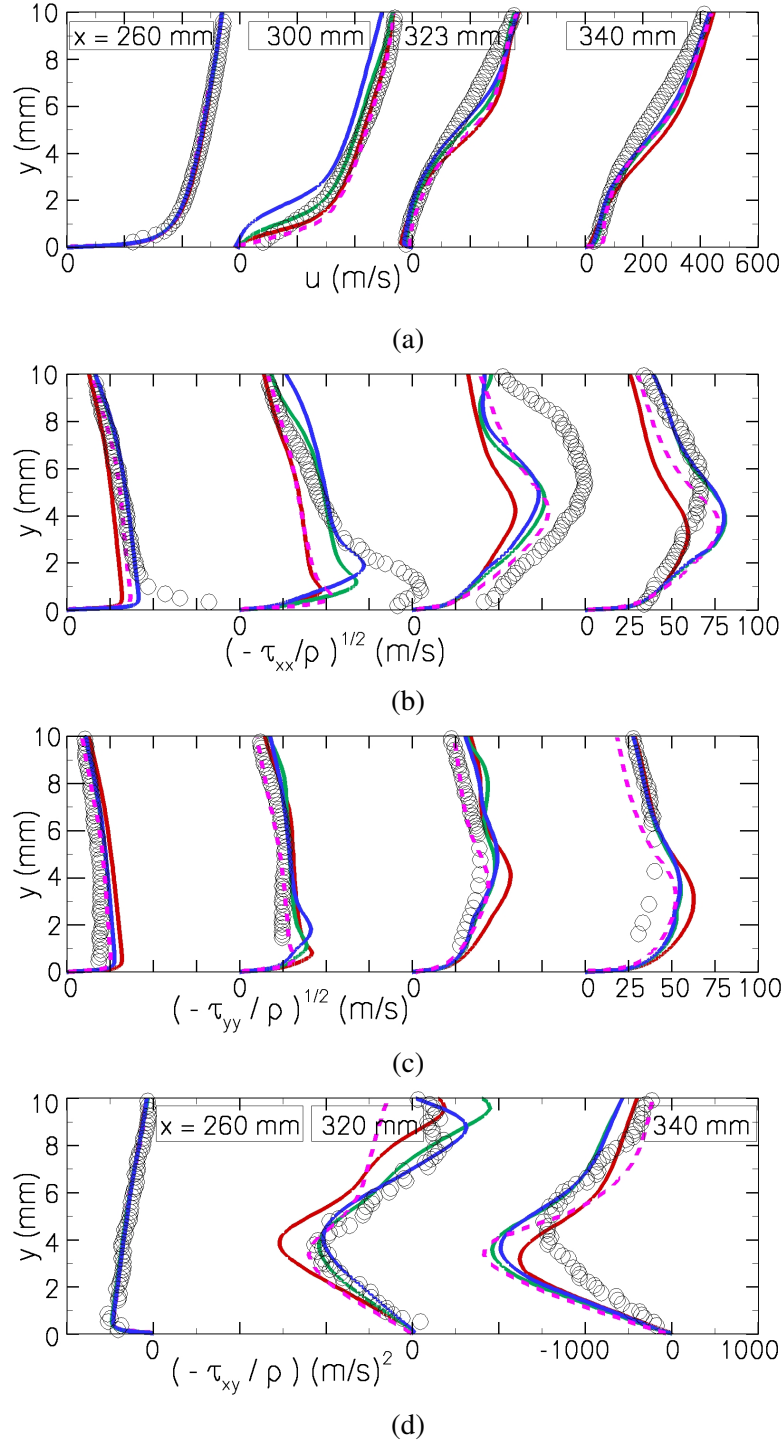


Figure 6.2: Comparison of the streamwise velocity, the streamwise, transverse, and shear components of Reynolds stresses at different streamwise positions : — Standard SST ; - - SSG/LRR- ω [95]; — EARSM; — SUQ-SST; ○ Experiment.

tendency to predict isotropic normal Reynolds stresses is the inherent limitation of the Boussinesq hypothesis based SST model. Moreover, SST model significantly underpre-

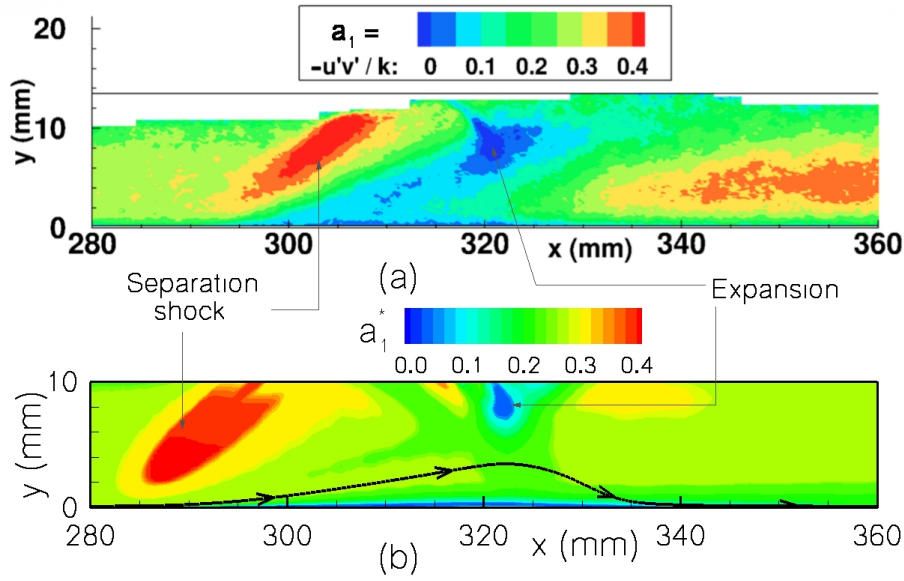


Figure 6.3: Distribution of the structure parameter in the vicinity of shock/boundary layer interaction region : (a) Experimental data [51] and (b) SUQ-SST simulation

dicts the streamwise Reynolds stress component and overpredicts the vertical component at all the four axial locations compared to the experimental data.

The improved modelling of the Reynolds stresses in EARSM, SUQ-SST, and RSM enables them to capture the Reynolds stress anisotropy. All three models predict higher levels of streamwise Reynolds stress component than the vertical component, which is consistent with the experiment. The SUQ-SST model significantly improves the normal Reynolds stress predictions compared to the standard SST model and brings the results closer to the experiment, although discrepancies still remain. The cause of these discrepancies can be attributed to the low-frequency shock oscillations, which are observed in the experiments and not accounted for in the RANS simulations.

Figure 6.2(d) shows the Reynolds shear stress comparisons within the inflow boundary layer ($x = 320$ mm) as well as in the separated flow ($x = 320$ mm) and post reattachment ($x = 340$ mm). At $x = 320$ mm, negative values of τ_{xy} in the shear layer are well reproduced by SUQ-SST, EARSM, and RSM, but significantly overestimated by the SST model. At the end of the interaction zone ($x = 340$ mm), SST gives reasonable results, whereas the other models overpredict the maximum shear stress level. On the other hand, the shear-layer thickness is underpredicted by SST and RSM, whereas SUQ-SST and EARSM predict a closer match with the experiment.

6.1.2 Structure parameter distribution

Figure 6.3(a) shows the behaviour of the structure parameter, calculated as the ratio between Reynolds shear stress and turbulence kinetic energy, in the interaction region using the UFAST experimental data [51]. The structure parameter increases to a peak value of 0.4 in the separation shock region from its equilibrium value of about 0.3 in the incoming boundary layer. It then decreases post the separation shock in the first half of the separation region ($x < 320$ mm) reaching levels below 0.1 and then again increases in the latter half ($340 > x > 320$ mm). Near the center of interaction at around $x = 320$ mm, flow expansion over the separation bubble decreases the structure parameter significantly. It again increases post reattachment ($x > 340$ mm), possibly in the reattachment compression waves, well beyond the values characteristic of equilibrium boundary layers.

Figure 6.3(b) shows the effective structure parameter distribution for the SUQ-SST model as calculated using Eq. (5.13). The model prediction in the SBLI region is qualitatively similar to the experimental data. Specifically, the model predicts an increase in the structure parameter values at the separation shock with levels almost identical to the experimental data. The model also correctly predicts a decrease in structure parameter values in the first part of the separation ($x < 320$ mm) and gives levels comparable to the experimental data in the latter part ($340 > x > 320$ mm). The reduced levels near the center of interaction at around $x = 320$ mm due to the expansion effect are also similar to that observed in the experiment. The high values post-reattachment are not reproduced by the model. The quantitative mismatch with the experimental data possibly explains the model's slight overprediction of the interaction region.

6.2 Mach 5 impinging shock cases

The different turbulence models are next applied to Mach 5 oblique shock impingement SBLI cases discussed in Section 3.4 of Chapter 3, and the results are compared in terms of the structure parameter in Fig. 6.4. The limiting streamline, showing the separation **S** and reattachment **R** points, is also identified in each case. Results are presented for the strongest interaction of 14° deflection angle; other cases show a similar trend.

SST model uses a constant value of the structure parameter $a_1 = 0.31$ inside the boundary layer (Fig. 6.4(a)) including regions of shock-induced adverse pressure gradients. This is in conflict with the experimental observations as discussed earlier. To represent the correct SBLI physics, EARSM features are introduced in the SST framework and the structure parameter is made a function of the local strain rate, as per Eq. (5.13). This modification is then able to capture the changes in structure parameter in the separation

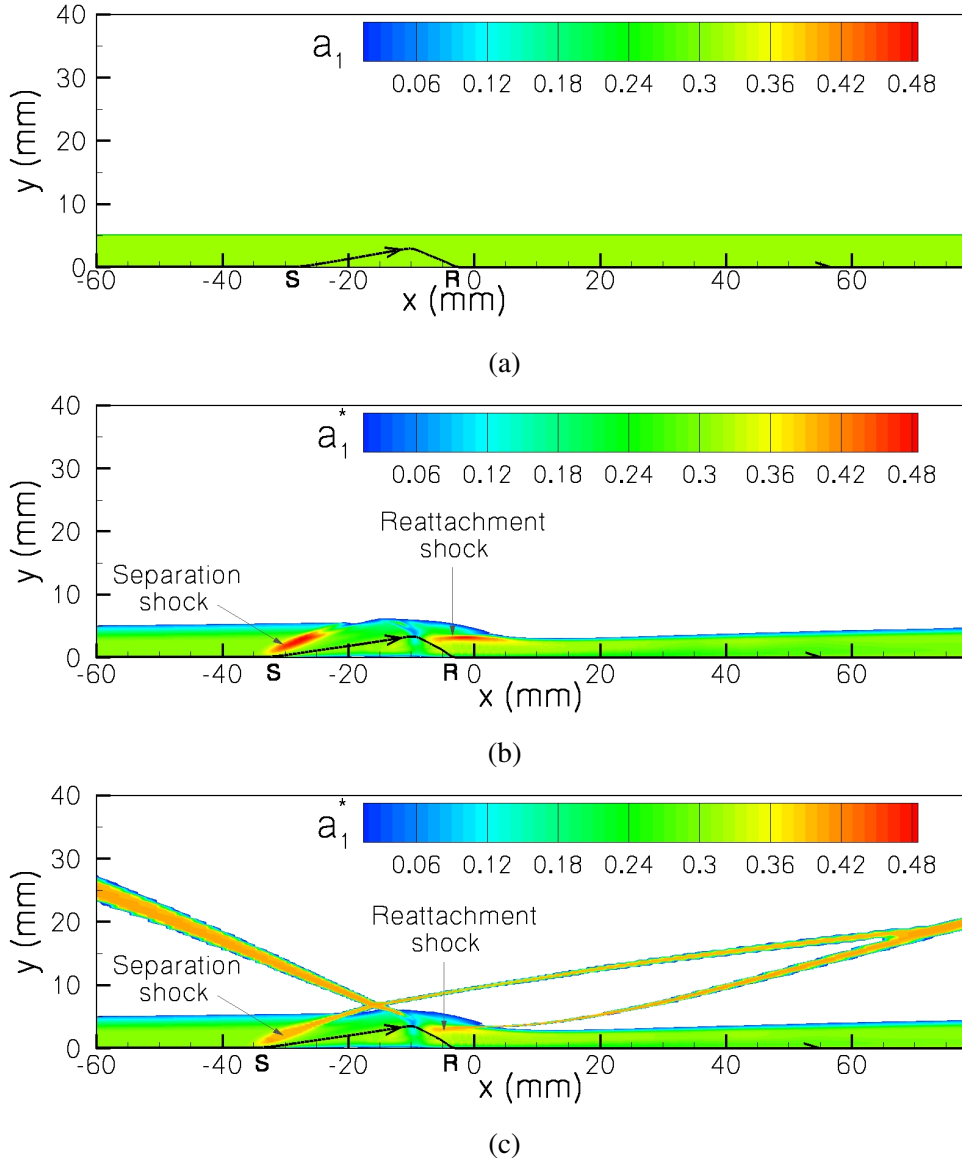


Figure 6.4: Distribution of the structure parameter in the Schulein's 14° SBLI flowfield obtained using (a) standard SST $k-\omega$ model and SUQ-SST model (b) Eq. (5.13) and (c) Eq. (5.17).

and reattachment shocks inside the boundary layer (Fig. 6.4(b)), where a peak value of about 0.48 is seen. By comparison, SUQ-SST correctly captures the changes in structure parameter in the entire shock structure (Fig. 6.4(c)), including regions outside the boundary layer. This includes the separation shock, the reattachment shock, and the other shocks shown in the figure. The structure parameter reaches a maximum value of 0.4 in the shock regions, as prescribed by the SU limiter.

The variation of the structure parameter has a direct bearing on the flow separation predicted by the model. The EARSM structure parameter given by Eq. (5.13) moves the separation point slightly upstream of the standard SST model. The SU-limiter moves

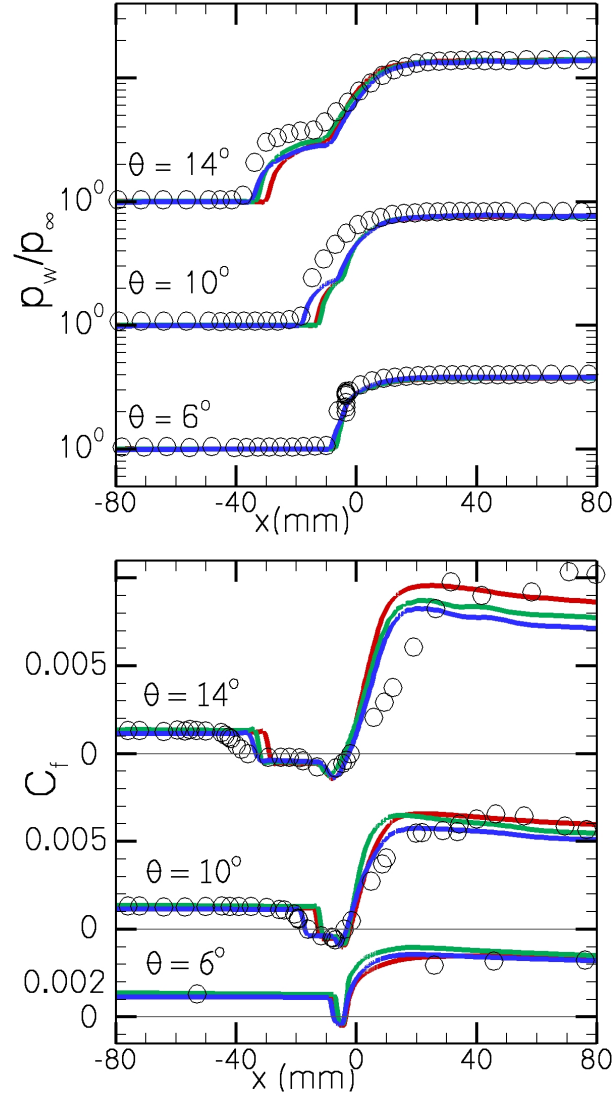


Figure 6.5: Comparison of (a) wall pressure and (b) skin friction coefficient obtained using different $k-\omega$ models with the experimental data for varying shock generator angles : — Standard SST $k-\omega$; — EARSM; — SUQ-SST; $\circ$ Experiment.

it further upstream to give a larger separation bubble. The three models give similar locations of reattachment point **R** as shown in Fig. 6.4.

Figure 6.5 compares the wall pressure and skin friction coefficient in the SBLI region obtained using the RANS models with the experimental data. The standard SST $k-\omega$ model predicts the separation shock location downstream of the experiment and significantly underpredicts the separation bubble size for the 10° and 14° cases. EARSM gives similar under-prediction for 10° , but shows improvement over the SST model for the 14° case. However, discrepancies still remain.

The SUQ-SST model improves the results significantly compared to the standard SST model. For the 10° case, the separation location is predicted accurately, both in terms

of the surface pressure rise and the zero crossing of the skin friction coefficient. A distinct pressure plateau is observed, unlike the experimental data. The skin-friction and surface pressure post-reattachment for the SUQ-SST model also agree well with the experiment. The strongest interaction, with 14° shock-generator angle, shows similar trends and SUQ-SST predictions are better compared to the standard SST model. The separation bubble size obtained using SUQ-SST is within 9% of the experiment, as opposed to 27% for SST.

For the weakest interaction case with 6° shock generator angle, all the turbulence models predict near incipient separation and agree well with experiment. There are no major differences between SST, EARM, and SUQ-SST for the wall pressure and skin friction profiles for this case. Overall, SUQ-SST consistently gives good predictions for varying strengths of interactions and match the experiment better compared to SST and EARSM.

6.3 Mach 8 compression corner cases

For an incoming flow with Mach number of about 8, the range of ramp angles considered in the experiments, discussed in Section 3.5 of Chapter 3, resulted in near incipient to fully separated flows. The incoming boundary layer is almost identical in these cases, so the effect of shock strength on the interaction length and flow separation can be studied. The strength of the shock waves also affects the distribution of the structure parameter as shown in Fig. 6.6. The SUQ-SST model captures the effect of shock strength on separation quite well and predicts near incipient separation for 27° case, mild separation for 30° case, and large separations for 33° and 36° cases, which is consistent with the experimental observations. The separation and reattachment locations are marked with **S** and **R**, respectively in Fig. 6.6.

The structure parameter increases from its equilibrium value of 0.31 and reaches a peak value of 0.4 in the separation and reattachment shocks inside the boundary layer and the ramp-induced shock in the inviscid region (Fig. 6.6). The shock-shock interaction over the ramp in the separated cases results in Edney type VI interference that emanates an expansion fan. Expansion effect reduces the values of structure parameter below 0.31 in the reattached boundary layer and the magnitude decreases with an increase in the expansion effect seen in terms of increasing ramp angle. The structure parameter eventually recovers back to its original value.

Figure 6.7 shows the wall pressure distribution for varying ramp angles as obtained using the different RANS models and experiments. For the weakest interaction i.e. 27° case, all the turbulence models give similar results and predict incipient separation match-

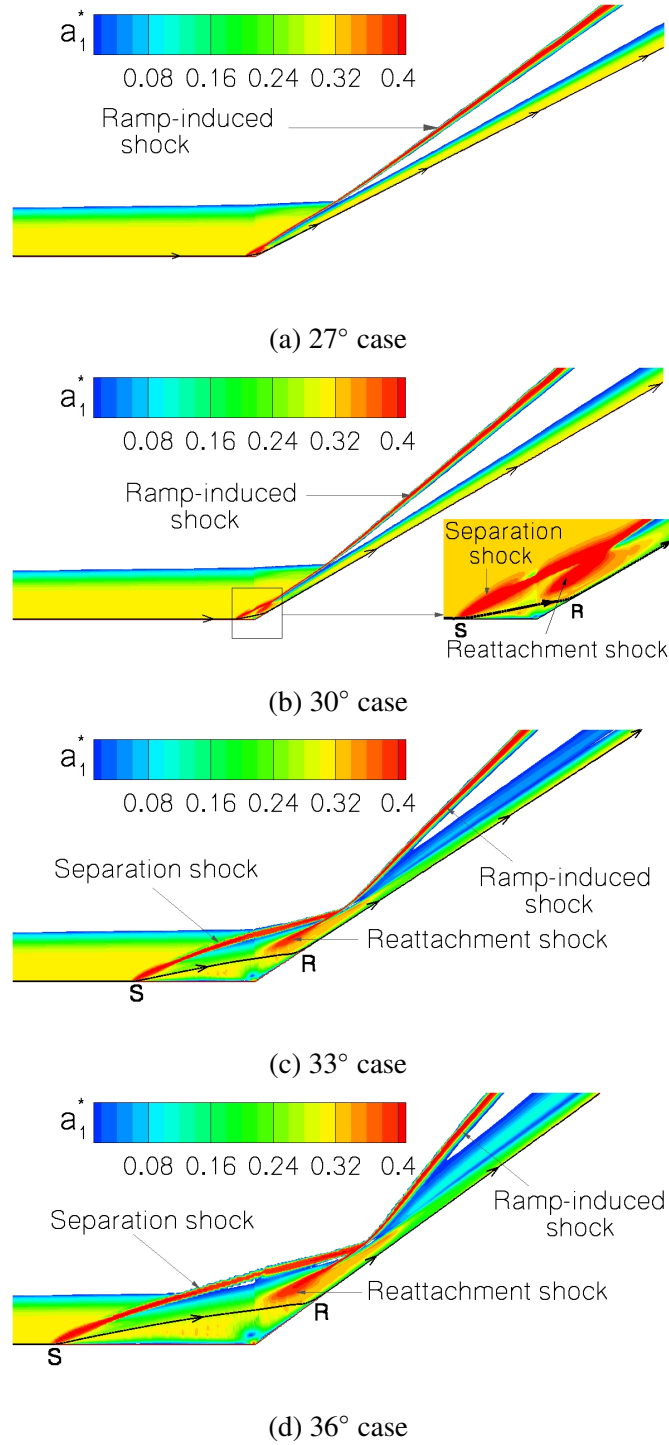


Figure 6.6: Distribution of the effective structure parameter a_1^* in the Mach 8 SBLI flow-field for different ramp angles

ing the experimental data. Experiments show mild separation for the 30° case and SUQ-SST predicts the pressure variation accurately with SST giving similar results. EARSM gives slightly delayed separation and predicts separation shock location downstream of the experiment for this case.

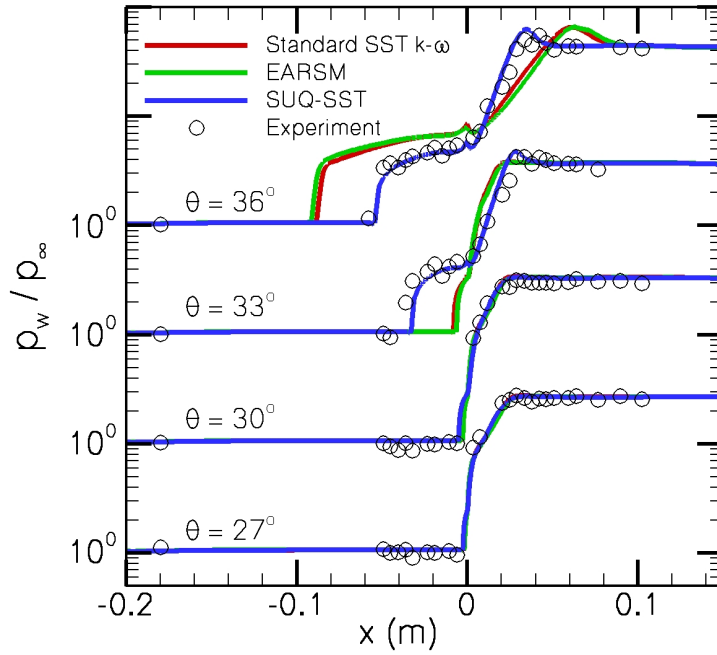


Figure 6.7: Comparison of wall pressure obtained using different $k-\omega$ models with the experimental data for the Mach 8 SBLI cases at varying ramp angles.

The 33° and 36° cases give fully separated flows and SUQ-SST predicts the separation shock location in close agreement with the experiment for both the cases. The model also predicts the pressure distribution along the ramp and the peak reattachment pressure in close agreement with the experimental data. On the other hand, SST and EARSIM give strikingly inconsistent results. Both models underpredict the separation regions considerably for the 33° case, while giving significant overprediction for the 36° interaction. The two models also underpredict the peak pressure for the 33° case, while giving the peak location much downstream of the experiment for the 36° case.

6.4 Sensitivity to the SU limiter

The SU limiter is obtained using Menter-like analysis of Sinha et al. [30] shock-unsteadiness model at a shock wave. The analysis gives the value of structure parameter in the shock regions in the range of 0.62 for weak shocks to 0.4 in high Mach number flows. The SU limiter, given by a_1'' in Eq. (5.16), brings in the effect of shock strength and decides the maximum value of structure parameter in the shock regions. Thus, it directly affects the eddy viscosity distribution in the SBLI regions and a higher a_1'' can yield comparatively smaller separation and interaction lengths, depending on the shock strength.

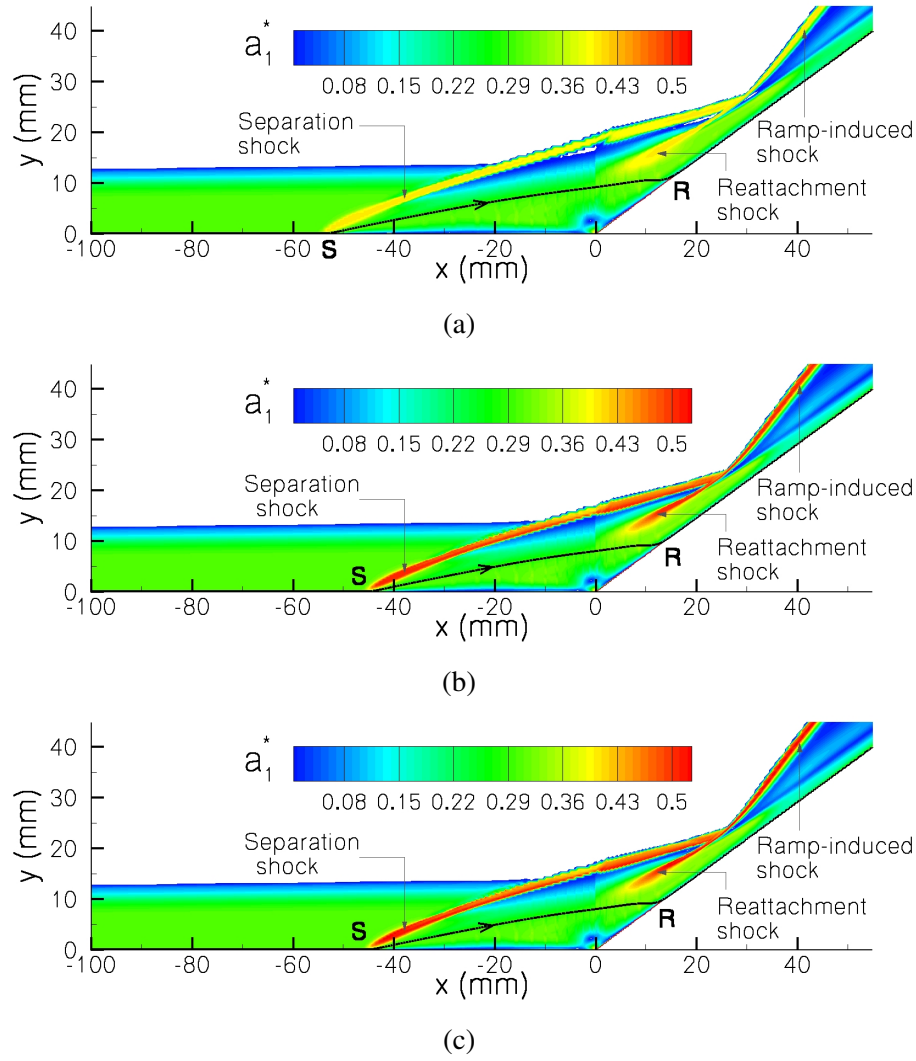


Figure 6.8: Distribution of the effective structure parameter a_1^* in the Holden's 36° SBLI case obtained using SUQ-SST model with (a) SU limiter = 0.4; (b) SU limiter = 0.51; (c) SU limiter = 0.62

Figure 6.8 shows the distribution of the effective structure parameter obtained using the SUQ-SST model using different values of the SU limiter. Specifically, the nominal value $a_1'' = 0.4$, the arithmetic mean of the extreme values $a_1'' = 0.51$, and the weak-shock limiting value $a_1'' = 0.62$, are used for the strongest interaction in the 36° Mach 8 case. All three cases presented in Fig. 6.8 show identical shock topology, where the structure parameter increases from its starting value of 0.31 in the boundary layer to the limiting value in the separation and reattachment shocks. A higher limiting value in the $a_1'' = 0.51$ solution moves the separation location **S** downstream, compared to the nominal $a_1'' = 0.4$ case. This results in a slight decrease in interaction length as seen from the wall pressure distribution (Fig. 6.9). Further increasing the SU limiter to 0.62 shows negligible change

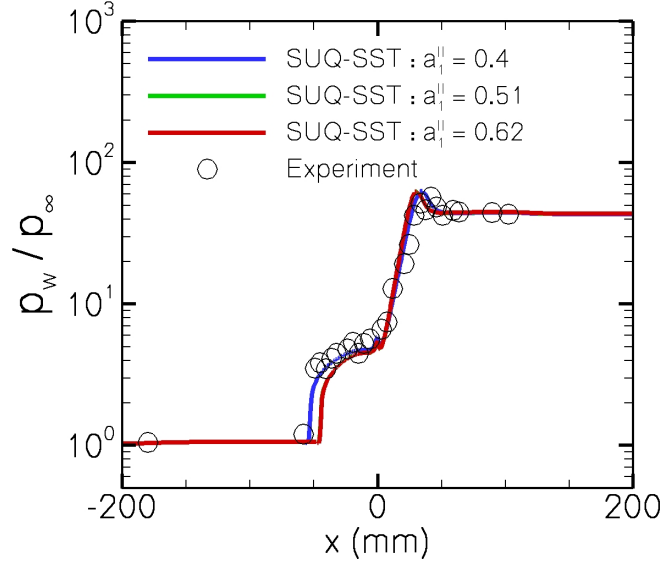


Figure 6.9: Sensitivity of wall pressure distribution to the SU limiter values for the SUQ-SST model for Holden 36° compression corner SBLI case

in the structure parameter distribution, location of the separation shock, and the surface pressure distribution.

The sensitivity of the results to the value of the SU limiter is found to be highest for the 36° case presented above. This is the strongest interaction among the cases considered in this work. The strain rates are therefore expected to be significantly higher than the other interactions, either at a lower Mach number or a lower deflection angle. For example, the SUQ-SST model predictions show almost no sensitivity to the SU limiter value for the 14° case at Mach 5, where the shock strength is about three times weaker than the 36° Mach 8 interaction.

6.5 Computational time comparisons

In comparison to the standard SST model, the SUQ-SST model requires additional computations of the quadratic part of the Reynolds stress components given in Eqns. (5.2) and (5.3), along with the EARS model parameters given in Eqns. (2.91) and (2.92). Further, the effective structure parameter is calculated using Eqn. (5.13) and the shock function f_s is obtained using Eqn. (5.21). These additional computations do not alter the computational requirement significantly. They need only about 6 - 7 % extra computational time per time-step than the standard SST model on identical grids. Moreover, the robustness of the standard SST model is not compromised by the modifications proposed in this work. The Courant-Friedrichs-Lewy (CFL) number used in SUQ-SST computations for each case are comparable to those in the case of SST and EARS models. As a result, the

number of time steps and the overall computational time required to reach steady state solution by the different models are within 10% of each other.

6.6 Summary

The SUQ-SST model is tested against a number of two-dimensional canonical SBLI flows, including UFAST Mach 2.3 experiment, Mach 5 impinging shock cases, and Mach 8 compression corner flows. The selected test cases cover a wide range of flow regime from supersonic to hypersonic speeds, and near incipient to fully separated flows. The SUQ-SST model is able to accurately capture the behaviour of the structure parameter in the SBLI regions in accordance with the UFAST experimental data. The added physics leads to significantly improved predictions of Reynolds stresses and their anisotropy compared to the standard SST model. The SUQ-SST model is also able to consistently predict the separation shock location and separation bubble size, whereas the standard SST model is shown to give inconsistent predictions of shock-induced separation across the range of flows. The new SST model thus clearly outperforms the original SST model in predicting a wide range of SBLI flows.

Chapter 7

Application of SUQ-SST model to base flows

The base flows considered in this work consist of 1) a supersonic flow over a back-facing step (BFS) and 2) a supersonic flow over a cylindrical blunt body. These flows exhibit SBLI, in case of the BFS, and SSLI, in case of the cylindrical blunt body. These geometric configurations can be considered as the canonical cases for practical applications such as a launch vehicle. Both the flow configurations show large separation and recirculation zones because of the sudden change in geometry as against the shock-induced separations seen in Chapter 6. Accurate prediction of the flowfield is important along with a correct estimation of the base pressure for these configurations.

7.1 Backward facing step

The BFS configuration chosen here, as discussed in Section 3.6 of Chapter 3, forms a part of scramjet combustor geometry and the case does not involve a transverse injection of fuel downstream of the step [99]. Mach 2 combustor inlet flow corresponds to flight Mach numbers of 5 to 7, the low Mach number range of the scramjet operation. Figure 7.1 shows the Mach number contours for the solution obtained using the SUQ-SST model; Mach levels below 1 are not shown. Mach 2 turbulent boundary layer forms over the bottom as well as the top flat surfaces. The sudden change in geometry because of the step on the bottom part causes the boundary layer to separate as a free shear layer. In the process, the flow expands around the step corner with the formation of Prandtl-Meyer expansion fan. A recirculation region of low-momentum fluid that is characterized by low velocities, low pressures, and high temperatures is formed behind the step. The flow in the recirculation bubble has a complex structure of vortices; each vortex has a distinct shape

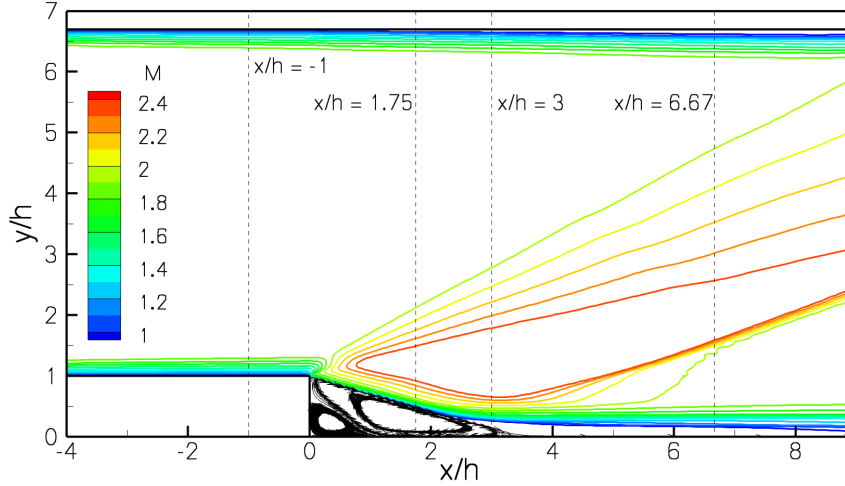


Figure 7.1: Mach number contours showing the supersonic flowfield over the BFS geometry obtained using the SUQ-SST model.

from the adjacent vortex. Such multiple vortices are also observed for the simulations with standard SST model. Similar results with the recirculation region containing multiple vortices are obtained by Khan et al. [103] using second order modified WENO and TVD numerical schemes for the identical geometry and flow conditions.

The free-shear layer separates the accelerating flow turning over the step and the recirculating flow. A reattachment shock is formed downstream of the step which turns the flow parallel to the wall. This is the region of SBLI, where the reattachment shock interacts with the re-attached boundary layer. The strength of interaction influences the flow recirculation. The standard SST and SUQ-SST models predict the length of recirculation zone behind the step to be approximately 3 step heights, which matches with the experimental observations. A lip shock resulting from the overexpansion of the flow over the expansion corner has been documented for supersonic flows over BFS [101]. However, this flow feature is not observed in the present computations or the experimental measurements for this particular case probably due to a relatively thicker boundary layer approaching the step. Overall, the computations capture the effect of expansion fan, coupled with the reattachment shock, in turning the flow over the step and then parallel to the bottom wall behind the step along with a correct recirculation bubble size. The dashed lines in Fig. 7.1 show the streamwise locations where the results obtained using the turbulence models are compared with the experimental measurements.

7.1.1 Incoming boundary layer profile

Figure 7.2 shows the comparison between the computations and the experiment for the mean streamwise velocity profile one step height before the step corner, i.e. at

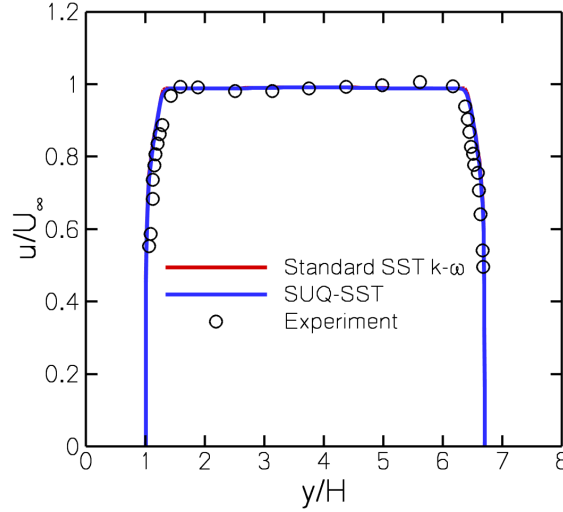


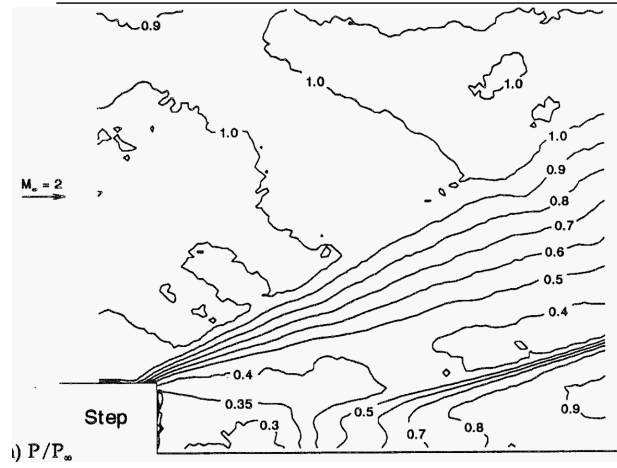
Figure 7.2: Comparison of the incoming boundary layer profile ($x/h = -1$) obtained using the standard SST, SUQ-SST turbulence models and the experimental measurements.

$x/h = -1$. Experimental measurements suggest that this location is sufficiently far upstream of the step and the boundary layer is not affected by the low pressure region in the base of the step. So, it provides a good estimate for the incoming boundary layer profile before encountering the step corner and facilitates for a proper comparison between computations and experiments. The computed boundary thickness with the standard SST and SUQ-SST models is about 1.6 mm (0.51 step heights), and it matches with the experiment.

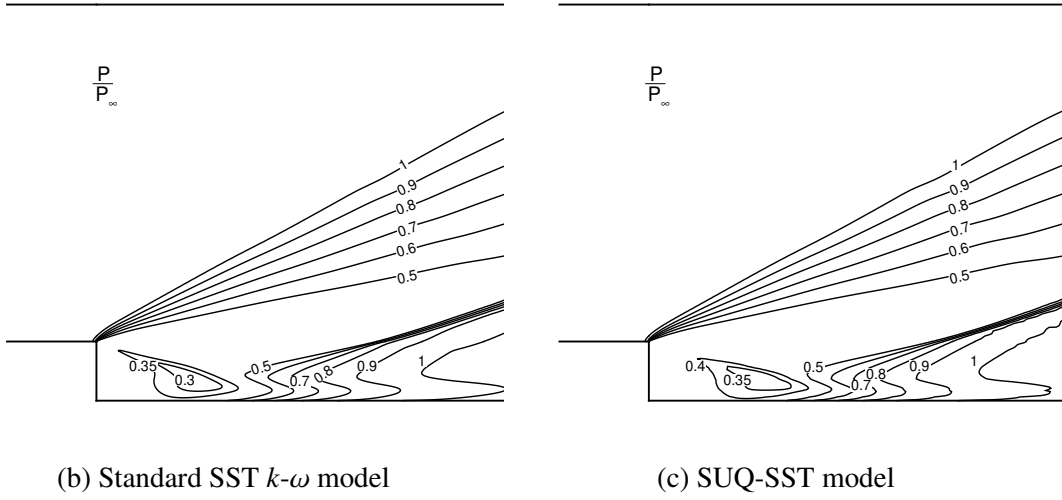
7.1.2 Comparison of pressure distribution

The comparison between the normalized pressure contours measured by the experimental PLIIF technique and those computed using the standard SST and SUQ-SST models is shown in Fig. 7.3. The purpose of this comparison is to establish the qualitative agreement of the experimental and computed results, as well as the similarity of the major features of the flowfield. The pressure contours of the expansion fan obtained using both the turbulence models emanate from its origin slightly upstream of the step corner. This observation is similar to the experimental measurements wherein the effect is more pronounced. The pressure in the shear layer as predicted by both the models is relatively constant, as expected. A steep pressure gradient in the vicinity of the recompression region is seen in the computations and measurement.

The degree of expansion and the location of the reattachment shock are in good agreement between the computations and experiment. However, pressure values in the vicinity of the wall in the recompression region are slightly higher in the computations than in the PLIIF data. There are also discrepancies in the recirculation region in terms



(a) Experimental PLIIF data

(b) Standard SST $k-\omega$ model

(c) SUQ-SST model

Figure 7.3: Comparison of mean pressure distribution in the BFS flowfield obtained using the standard SST, SUQ-SST turbulence models, and the experimental measurements.

of the distribution of pressure. These may be partly due to the issues in the experimental measurement because of the shadow cast by the step as discussed in [99], and partly because of the multiple vortices obtained in the computations. However, the computations give similar pressure levels in the recirculation region as the measurement. Overall, there is a fair agreement between the computed and measured results.

7.1.3 Comparison of temperature distribution

The normalized temperature distribution in the vicinity of the BFS obtained using the standard SST and SUQ-SST model computations and the measurement is shown in Fig. 7.4. A temperature rise in the boundary layer near the wall on the top flat surface and the bottom flat surface ahead of the step is seen in the measurement as well as the compu-

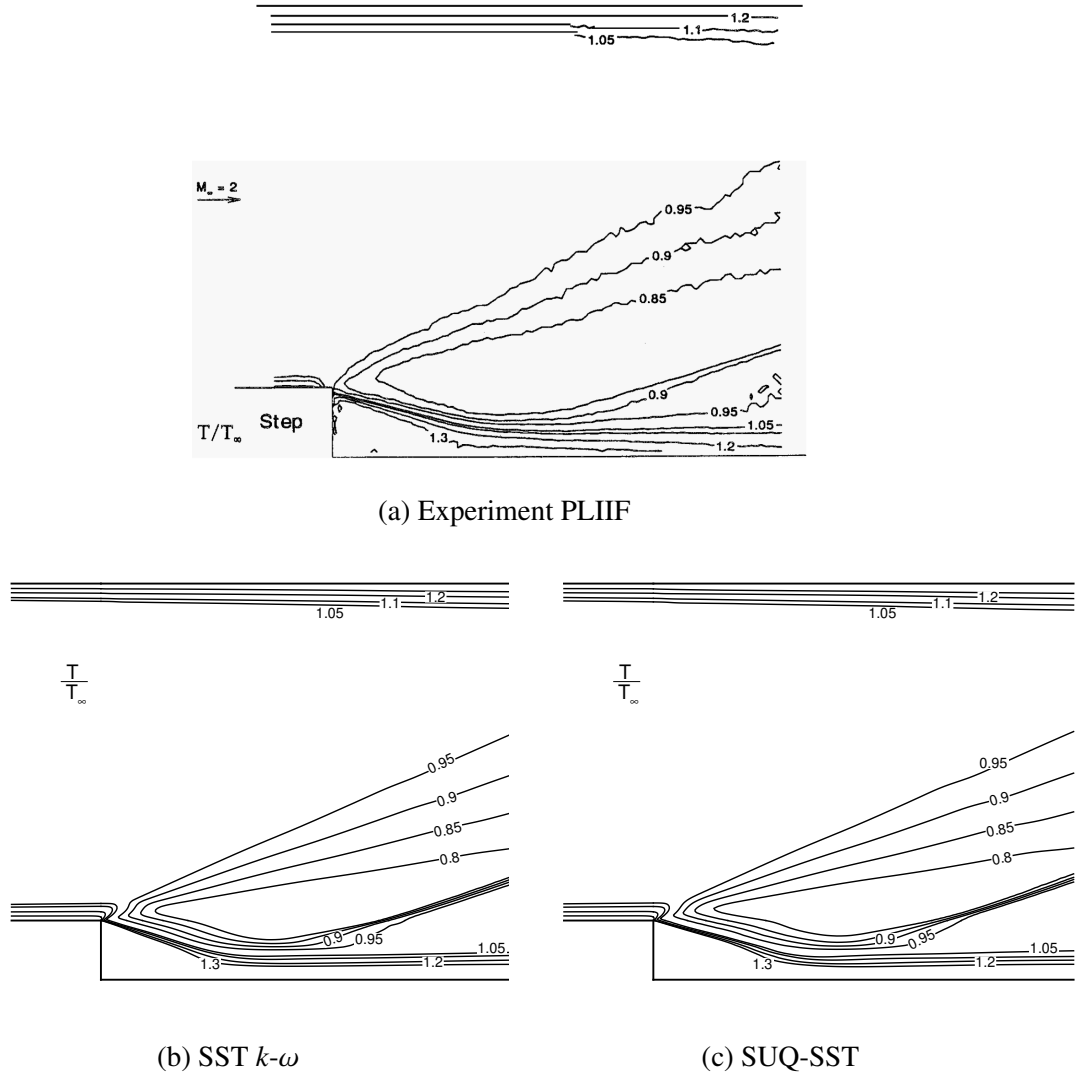


Figure 7.4: Comparison of mean temperature distribution in the BFS flowfield obtained using the standard SST, SUQ-SST turbulence models, and the experimental measurements.

tations. The temperature decreases in the expansion region and very near the step corner where the boundary layer separation takes place. The computed isotherms in the expansion fan are essentially straight lines along the expansion waves. Both the models predict a similar decrease in temperature across the expansion fan which matches fairly well with the experiment. Also, the computations show large gradients in the shear layer and the reattached boundary layer downstream of the step, similar to the measurement. The computations show a relatively steep temperature gradient across the reattachment shock causing the isotherms to coalesce which is not the case with the experiment. However, the computations give similar temperature levels in the shock region as the experiment. The results in the recirculation region are inconclusive because of the unavailability of tem-

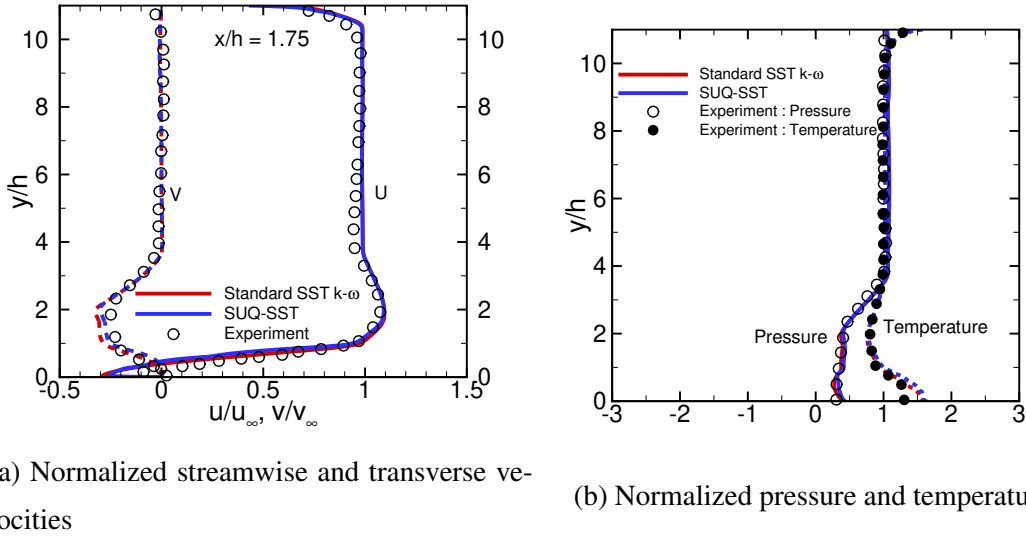


Figure 7.5: Comparison of the standard SST and SUQ-SST model predictions with the experiments at $x/h = 1.75$ (inside the separation bubble).

perature levels in the experimental data. Overall, a good agreement is observed between the measurements and both the model computations for the position of the prominent flow features and the temperature levels therein : the expansion fan, the shear layer, and the reattachment shock.

7.1.4 Comparisons of mean quantities at different streamwise locations

Profiles of streamwise velocity, transverse velocity, pressure, and temperature in the BFS flowfield obtained using the standard SST and SUQ-SST models are compared with the experimental data at different streamwise locations in Figs. 7.5, 7.6, and 7.7. The positions of these different stations are depicted by the vertical dashed lines in Fig. 7.2. The three stations consist of a location in the separation bubble ($x/h = 1.75$), a location just downstream of the experimentally observed reattachment point ($x/h = 3$), and a location far downstream of the reattachment point ($x/h = 6.67$). Here, the mean velocities u and v , the mean static pressure, and the mean static temperature are normalized using their respective freestream values. The comparisons show that the standard SST and SUQ-SST models give similar results for the mean flow for this case. The computations show a reasonable match with the experimental data at all the stations considered for the comparisons.

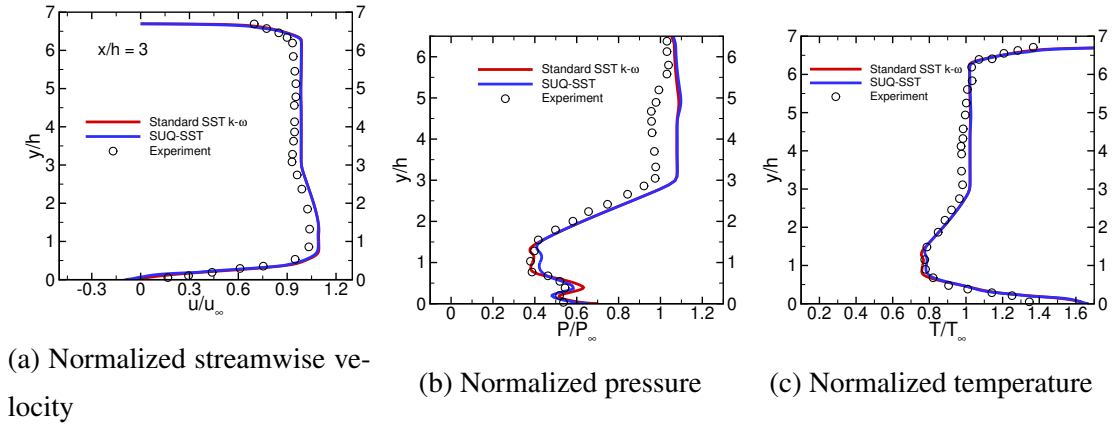


Figure 7.6: Comparison of the standard SST and SUQ-SST model predictions with the experiments at $x/h = 3$ (just after the experimentally observed reattachment).

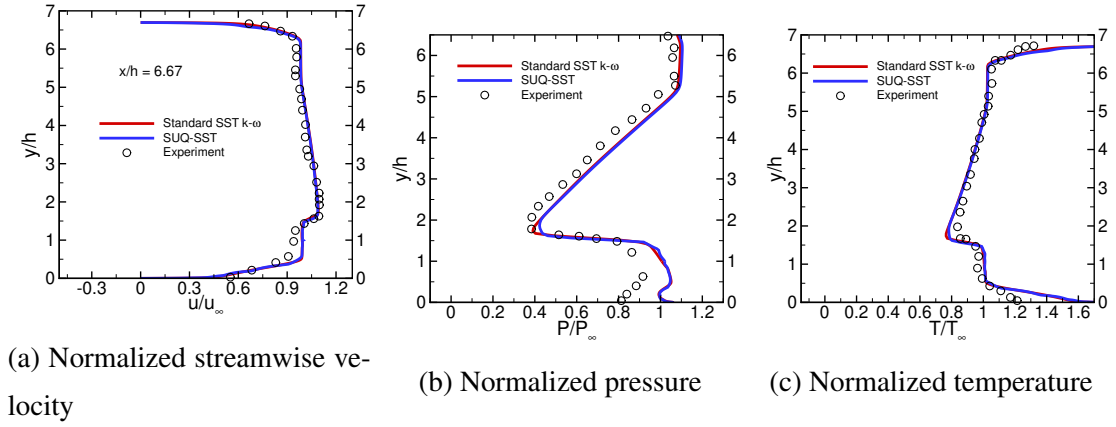


Figure 7.7: Comparison of the standard SST and SUQ-SST model predictions with the experiments at $x/h = 6.67$ (downstream of the experimentally observed reattachment).

Figure 7.5 shows the wall normal variations of mean quantities at the first station ($x/H = 1.75$). Here, the recirculation zone extends from the wall to about $y/H = 0.6$. The computed velocities and the pressure in this region show a fairly good match with the experiment. However, both the models predict slightly greater temperature values than the experiment in the recirculation zone possibly due to the multiple embedded vortices obtained in the computations. The shear layer that bridges the recirculation zone and the expansion fan is located between $y/H = 0.6$ and $y/H = 1$. The computations match the experimental data in these regions fairly well. The expansion fan, which reduces the temperature and pressure and increases the streamwise velocity, begins at about $x/H = 1.2$ and extends up to about $y/H = 2.4$ (see Fig. 7.1). Again, computations match well with the measurements in this region. The mean quantities remain constant in the inviscid

region till the top wall is reached where the streamwise velocity and temperature show distinct variations. This trend is well captured in the computations.

Figure 7.6 shows the computed and measured values at the second station ($x/H = 3.0$). The computed streamwise velocity and temperature show an excellent match with the measured values at this location. However, the computed pressure shows a significant variation near the wall and inside the recirculation region as against the near constant values observed in the experiment. Also, the computations predict greater pressure values than the ambient pressure in the inviscid region between $y/H = 3$ to about $y/H = 5.5$. This is also the case at the first station shown in Fig. 7.5. However, the measurements show variation in pressure with values lower than the ambient pressure, which is in contrast to the constant and near ambient pressure values at the first location. As per the computations, the positions of the nascent reattachment shock, which elevates the pressure, and the expansion fan, which lowers the pressure, are centered at around $y/H = 0.75$ and $y/H = 2.1$, respectively. This is in excellent agreement with the measurements.

The wall normal variation of the mean quantities obtained from the standard SST and SUQ-SST model computations and experimental measurements at the last station ($x/H = 6.7$) are shown in Fig. 7.7. The position of the reattachment shock, which is centered at around $y/H = 1.7$, is in good agreement between the computations and the experiment. This is evident from the mean velocity, pressure, and temperature plots. The computed velocity and temperature variations from the bottom to the top wall show a fairly good match with the experiment. The temperature values predicted by the two models are slightly higher than the measurements in the near wall region. The discrepancy near the wall is greater for the pressure values, where computations give higher values than experiments. However, similar behaviour of pressure variation is observed in the computations of [104, 107]. Overall, the agreement between measurements and computations is fairly good at the three stations in terms of the location and extent of separation bubble, expansion fan, and the reattachment shock.

7.2 Supersonic base flow

The simplest axisymmetric base flow geometry that retains all of the primary features of a typical missile or projectile flowfield is the circular cylinder with a blunt base without a propulsive jet or control fins. A turbulence model should be able to accurately predict the flowfield in the base region of a solid-walled cylindrical afterbody. This is a primary requirement before applying the model to practical applications, such as launch vehicles involving propulsive jets during their ascent. For the present study, the axisymmetric base

flow configuration of Herrin and Dutton [109], discussed in Section 3.7 of Chapter 3, is used for the model validation. Figure 7.8 shows the Mach number distribution for a supersonic Mach 2.46 flow over the blunt cylindrical afterbody configuration obtained using the three-dimensional RANS simulations with the SUQ-SST model. Mach contours below the value of 1 are not shown in Fig. 7.8. The flowfield is similar to the BFS configuration, discussed in the previous section.

Figure 7.8 shows the axisymmetric base flowfield obtained using the SUQ-SST model. The mean flowfield downstream of the blunt cylindrical afterbody contains many fluid dynamic features that interact to form the base flow solution. The supersonic Mach 2.46 flow over the cylindrical body expands at the base corner with the formation of Prandtl-Meyer expansion fan. A compressible shear layer, formed by the geometric separation of the afterbody boundary layer, separates the outer inviscid flow from the recirculation region behind the base. The recirculation region consists of two symmetric vortices separated by the geometric centerline. Approximately two base radii downstream, the shear layer enters the recompression and realignment region and eventually reattaches along the centerline of the wake at around $z/R = 2.9$.

The reattachment point (or rear stagnation point) separates the downstream wake flow from the fluid that is returned to the base by the adverse pressure gradient associated with the shear layer realignment. A reattachment shock wave is formed in this region and it interacts with the realigning shear layers. The shock-shear layer interaction (SSLI) influences the recirculating flow in the base region and, therefore, the base

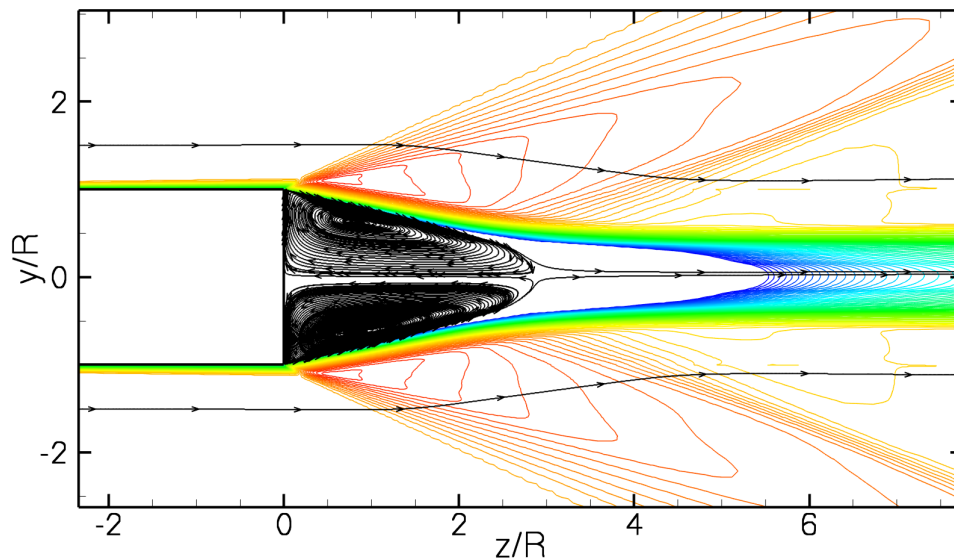


Figure 7.8: Mach contours showing the flowfield behind a cylindrical blunt base afterbody obtained using the SUQ-SST model.

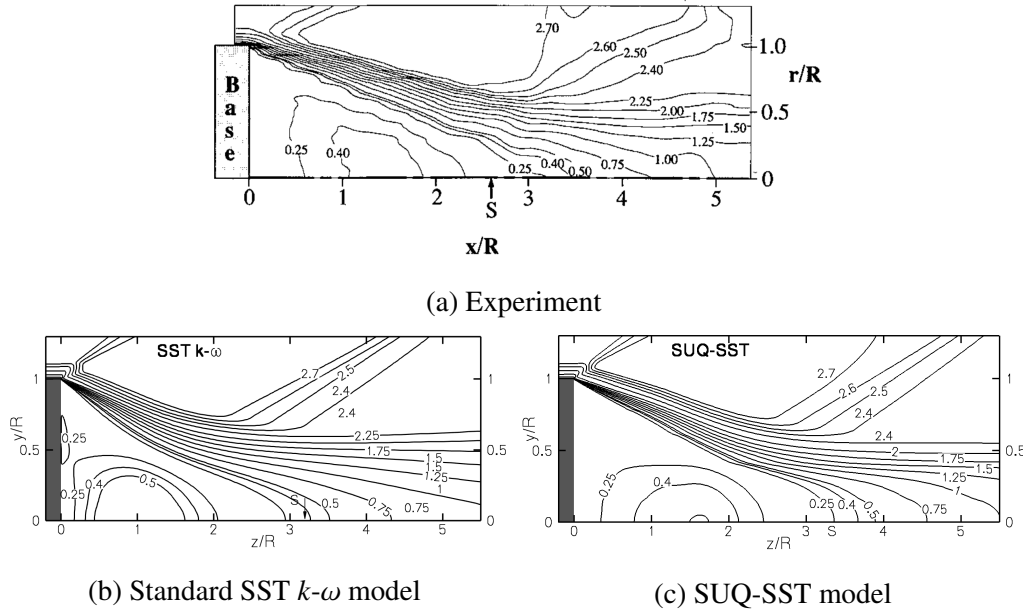


Figure 7.9: Comparison of Mach number distribution in the axisymmetric base flowfield obtained using the standard SST, SUQ-SST turbulence models, and the experimental measurements.

pressure. Overall, the SUQ-SST model is able to capture all the important flow features in the axisymmetric base flowfield, such as expansion at the base corner, separation and recirculation regions, and reattachment shock wave. Similar flowfield is obtained with the standard SST model, but the results show differences in the distribution of mean quantities and the locations of flow reattachment and the reattachment shock formation behind the base, as discussed in the subsequent sections.

7.2.1 Comparison of Mach number distribution

A comparison between the standard SST, SUQ-SST models and the experiment for the Mach number distribution near the base region is shown in Fig. 7.9. The steep Mach number gradients through the initial portion of the shear layer are clearly evident in the turbulence model results consistent with the measurement. The spreading of the contour lines further downstream is indicative of the growth of the shear layer prior to reattachment and, also, the wake development downstream. Both the turbulence models show this trend and match the experimental data well. The experiment shows that the flow along the axis reaccelerates to sonic velocity at approximately five base radii downstream. However, both the models predict this distance downstream of the measurement.

The Mach number distribution along the recompression waves starting at around $z/R = 3$ is almost similar between the two model predictions and the experiment. In

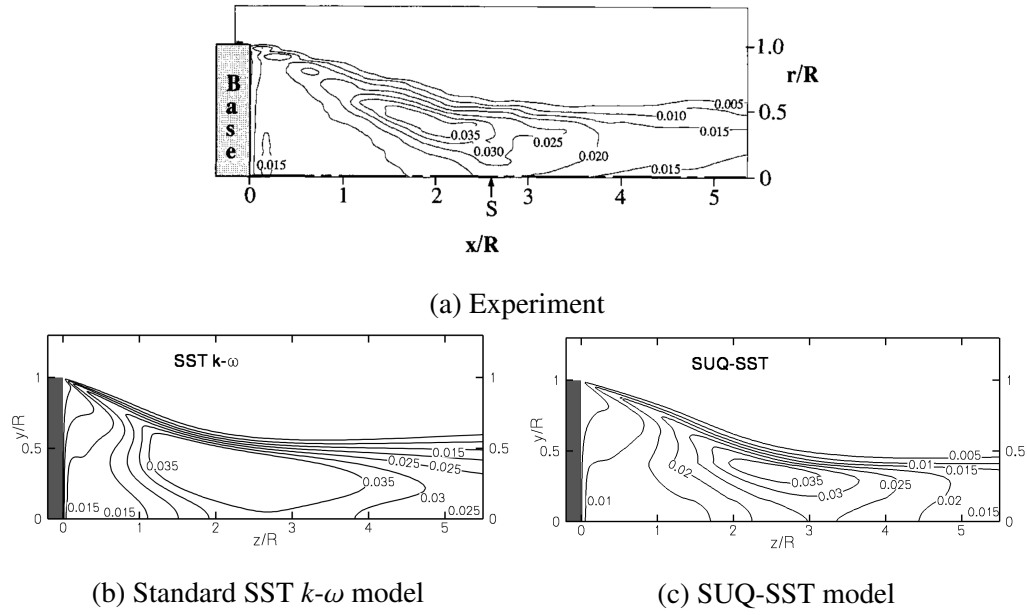


Figure 7.10: Comparison of TKE distribution in the axisymmetric base flowfield obtained using the standard SST, SUQ-SST turbulence models, and the experimental measurements.

the experimental measurements, the maximum Mach number of the reversed flow is 0.48 and is located on the centerline at approximately $z/R = 1.5$. The SUQ-SST model shows a good match with the measurement and gives almost a similar distribution of Mach number in the recirculation region. On the other hand, standard SST model predicts a slightly higher value of the reversed flow Mach number (about 0.5) than the experiment and it is located closer to the base i.e. at around $z/R = 1$. Overall, SUQ-SST model shows a better match with the experiment for the Mach number distribution, especially in the recirculation region, than the standard SST model.

7.2.2 Comparison of TKE distribution

Turbulence kinetic energy (TKE) is an important turbulence quantity often used to describe the overall level of turbulent fluctuations. Figure 7.10 shows the comparison of the TKE distribution throughout the near-wake of the cylindrical afterbody obtained using the standard SST and SUQ-SST turbulence models and the experimental measurement. In the experimental measurements, the turbulence kinetic energy increases as the shear layer develops up to approximately 2.2 base radii downstream of the base where a local maximum of about 0.35 occurs. The SUQ-SST model gives a similar TKE distribution along the shear layer development region with a similar local maximum value of 0.35. However, the location of this maximum value is predicted slightly downstream of the

experiment at around $z/R = 2.6$. On the other hand, the standard SST model gives a significantly different TKE distribution in the shear layer than the experiment and gives a considerably larger region of the higher TKE values.

Generally, the magnitude of the turbulence kinetic energy in the shear layer is significantly larger than in the approach boundary layer indicating that rapid mixing and entrainment occurs along the shear layer boundaries. Thus, SUQ-SST model gives a better representation of this phenomenon than the standard SST model. As the shear layer undergoes recompression and realignment, measurements suggest that the turbulence activity is reduced and continues to diminish into the downstream wake. Both the turbulence models are able to predict this trend, but the SUQ-SST model shows a better match with the experiment than the standard SST model. In the recirculation region near the base, the level of turbulence kinetic energy is reduced by the lack of turbulence production due to small mean velocity gradients. The two turbulence models are able to predict this trend as well. Overall, the SUQ-SST model is found to give a better resolution of the TKE distribution than the standard SST model in the axisymmetric base flowfield.

7.2.3 Variation of mean velocities at different streamwise locations

Figure 7.11 plots the normalized streamwise velocity profiles downstream of base and the comparisons between the standard SST and SUQ-SST model predictions are shown with the available experimental data. The different abscissa have been chosen for representing different parts of the flow. The first three locations i.e. $z/R = 0.079$, $z/R = 1.26$, and $z/R = 2.52$ represents the recirculation region. At $z/R = 0.079$, which is close to the base, the negative reverse velocity is predicted well by the two models and the shear layer location and extent at around $y/r = 0.9$ is also predicted well. Around half-way to the experimentally predicted reattachment location, i.e. $z/R = 1.26$, the negative reverse velocity is reasonably well predicted by the two models. However, the spread of the shear layer and the location at which the streamwise velocity crosses the zero value is better predicted by the SUQ-SST model. The standard SST model predicts a greater extent of the shear layer moving the zero streamwise velocity value slightly towards the center of the base.

Similar observations for the spread of shear layer can be made for $z/R = 2.52$, which is slightly upstream of the experimentally observed reattachment point. At this location, the standard SST model predicts positive values of the velocity near the base center suggesting that the model predicts an earlier reattachment than the experiment. At a location downstream of the reattachment ($z/R = 3.78$), which is the redevelopment region, the velocity values in the shear layer along with the strength and location of the recompression

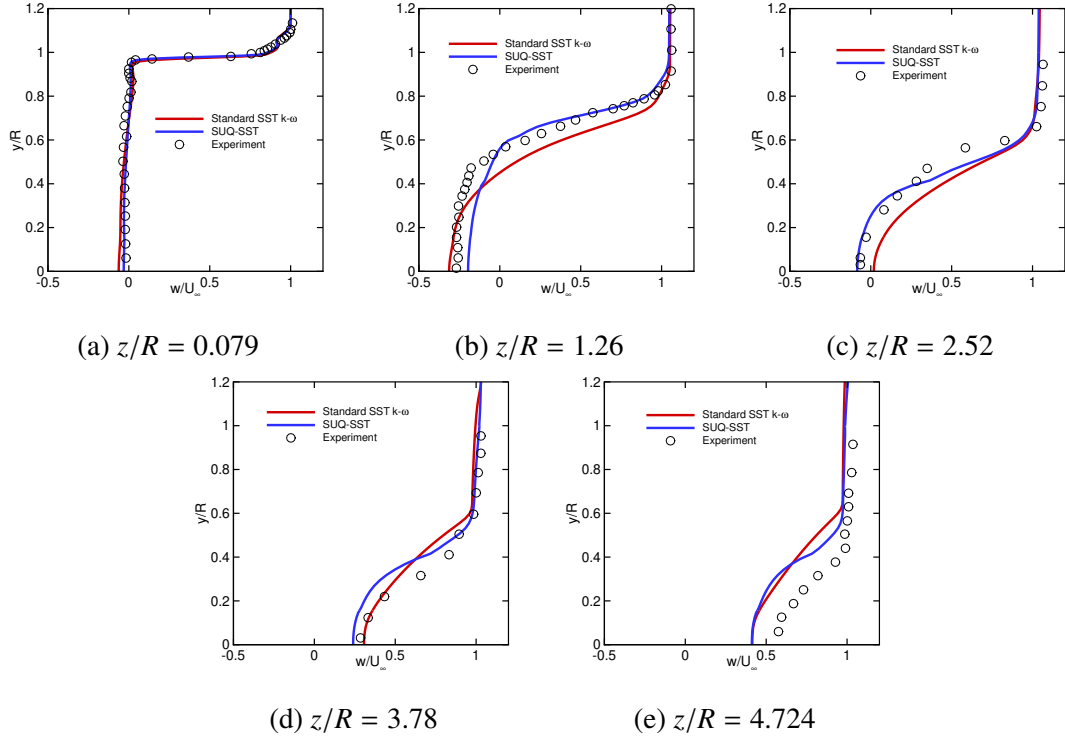


Figure 7.11: Normalized streamwise velocity variation at different axial locations in the Mach 2.46 axisymmetric base flowfield.

shock wave are predicted fairly well by the two models. Further downstream at $z/R = 4.724$, both the models predict similar values of the velocity in the redeveloping shear layer. However, the values are lower than the measured values indicating a lower rate of velocity recovery during flow redevelopment, which is a deficiency observed in the RANS simulations using the two-equation models [110, 112].

The comparisons of the radial mean velocity component at three different axial locations obtained using the standard SST, SUQ-SST models and the available experimental measurements are shown in Fig. 7.12. The first two locations i.e. $z/R = 0.079$ and $z/R = 1.26$ correspond to the recirculation region whereas the last location, i.e. $z/R = 2.67$, lies just downstream of the experimentally observed shear layer reattachment. At $z/R = 0.079$, the two models predict positive values of the radial velocity near the base center whereas experiment shows negative values. However, both the models predict a similar trend as the measurements for the variation of the radial velocity at this location and give correct extent of the separated shear layer.

At the next station, i.e. $z/R = 1.26$, the SUQ-SST model gives an excellent match with the experiment than the standard SST model, which predicts greater negative values of the radial velocity in the recirculation region than the experiment. At the location just downstream of the experimentally observed reattachment, i.e. $z/R = 2.67$, the trend of

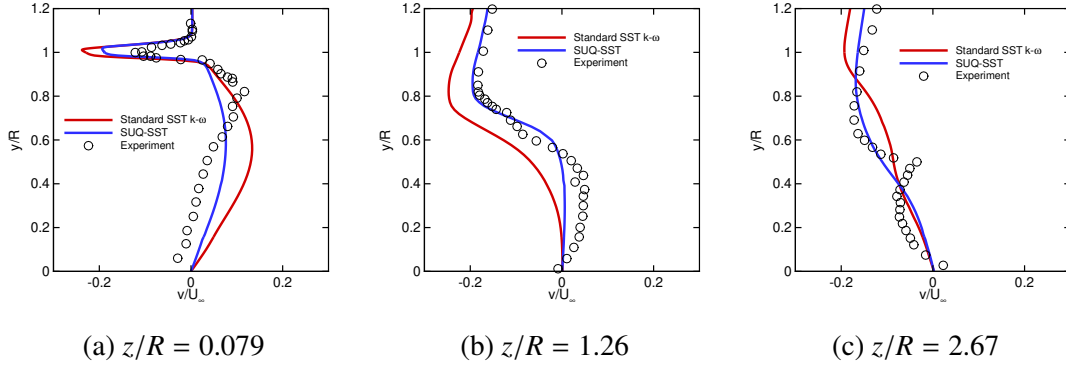


Figure 7.12: Normalized radial velocity variation at different axial locations in the Mach 2.46 axisymmetric base flowfield.

the radial velocity predicted by the two model is not in agreement with the experiment, especially in the region $y/R \sim 0.4 - 0.6$ where the recompression shock starts to appear. However, beyond $y/R = 0.6$, SUQ-SST gives excellent match with the experiment and a large error is obtained for the standard SST model. This suggests that the degree of expansion at the base corner is predicted correctly by the SUQ-SST model. The correct prediction of the strength of the expansion is extremely important in defining the shear layer characteristics throughout the near-wake [169]. Hence, the SUQ-SST model is able to accurately predict the shear layer characteristics, as pointed out earlier. Overall, the mean velocity profiles obtained using the SUQ-SST model correlate more closely with the experiment than the standard SST model and the improvements are especially seen in the recirculation region, where the velocity gradients are large and turbulence plays an influential role.

7.2.4 Normal Reynolds stress variations at different streamwise locations

Figures 7.13 and 7.14 show the profiles for the nondimensional Reynolds stress components in the streamwise and radial directions, respectively. Comparisons between the standard SST model, SUQ-SST model, and the measurements at different axial locations are shown. Experimental measurements suggest that the two Reynolds stress components have similar variations in all the considered locations; $z/R = 2.52$ corresponds to the location just upstream of the experimentally observed shear layer reattachment whereas the other four locations correspond to the redevelopment region. The measured peak values of streamwise Reynolds stress occur in the shear layer and the peak values decrease with an increase in the axial distance from the base. Similar observations can be made for the radial Reynolds stress component.

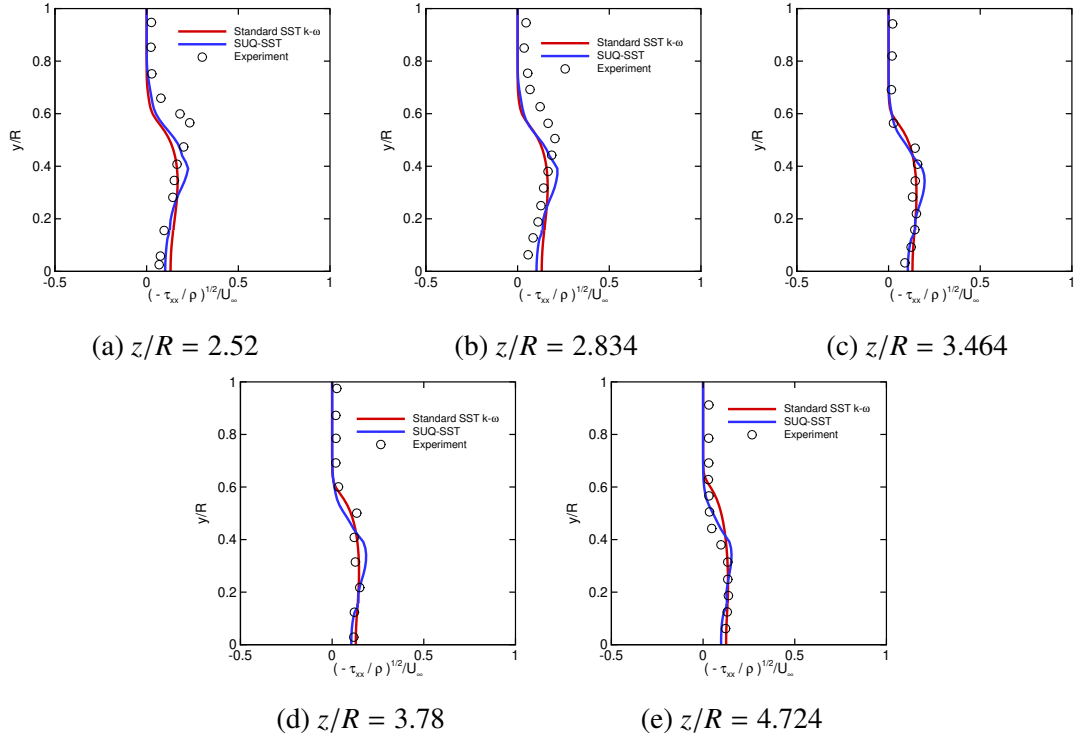


Figure 7.13: Normalized streamwise Reynolds stress τ_{xx} variation at different axial locations in the Mach 2.46 axisymmetric base flowfield.

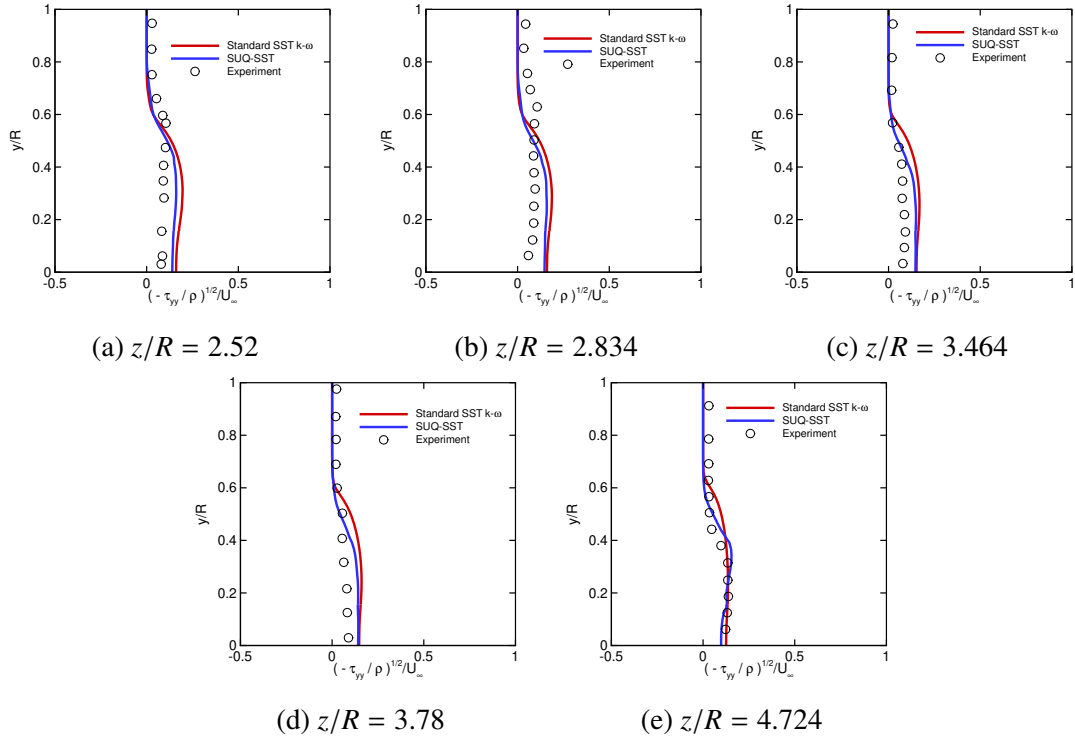


Figure 7.14: Normalized radial Reynolds stress τ_{yy} variation at different axial locations in the Mach 2.46 axisymmetric base flowfield.

The transfer of energy from the mean flow to the turbulence field via classical production mechanisms occurs primarily through the streamwise Reynolds stress component. The radial and tangential components acquire energy via more passive redistribution mechanisms such as pressure-velocity interactions and turbulent diffusion. As a result, the experimental measurements give the relative ordering of the normal Reynolds stresses at the corresponding axial locations in the near wake flowfield as $\tau_{xx} > \tau_{yy} \sim \tau_{zz}$, as seen in Figs. 7.13 and 7.14.

The standard SST and SUQ-SST models give similar profiles for the two normal Reynolds stresses at all the axial locations considered for comparisons. The results obtained using these two turbulence models show a reasonable match with the experiment at all the axial locations. However, one noticeable difference in the two model predictions is that the standard SST model predicts almost similar values of the two normal Reynolds stresses at the corresponding axial locations. This showcases the limitation of the isotropic Boussinesq hypothesis. On the other hand, SUQ-SST model gives higher streamwise Reynolds stresses than the radial components at the corresponding axial locations which is consistent with the experimental observations. Consequently, SUQ-SST model gives a slightly better match for the two normal Reynolds stress components with the experiment than the standard SST model.

7.2.5 Base pressure and centerline velocity comparisons

The accurate prediction of base pressure and, hence, base drag is of particular interest from an engineering perspective. Figure 7.15(a) shows the radial variation of the pressure coefficient C_p over the base, with $y/R = 0$ denoting the base center and $y/R = 1$ indicating the base edge. The base pressure coefficients predicted by the standard SST and SUQ-SST models are compared with the experimental data. The base pressure coefficient C_p in Fig. 7.15(a) is calculated as,

$$C_{p_{base}} = \frac{2[(P_{base}/P_{\infty}) - 1]}{\gamma M_{\infty}^2} \quad (7.1)$$

where P_{base} is the base pressure. The experimental data shows an essentially constant pressure acting over the entire base surface with only a slight increase (about 3%) in pressure near the outer edge of the base. This results an essentially constant C_p acting over the entire base surface. The two turbulence models also predict a similar trend of near constant C_p distribution over the base. However, the standard SST model underpredicts C_p compared to the experiment and the difference is as high as 30% towards the outer edge and about 15% near the base center. On the other hand, SUQ-SST model shows a significant improvement over the standard SST model and gives an excellent match

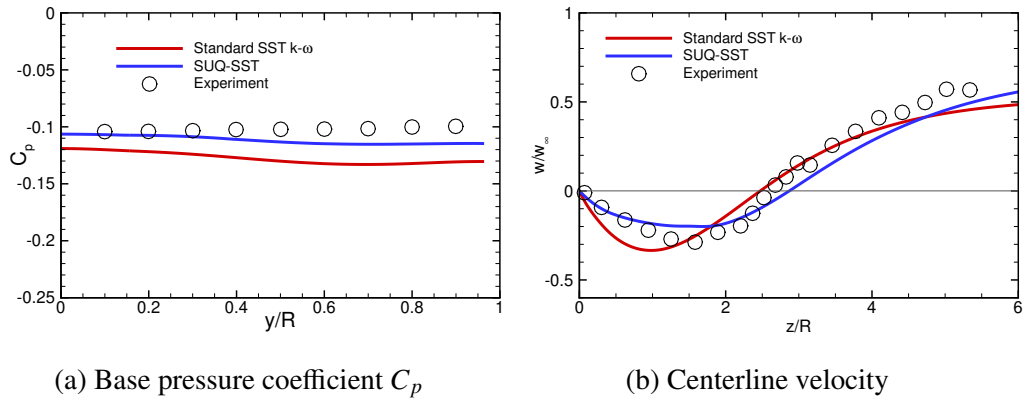


Figure 7.15: Comparison of the standard SST and SUQ-SST model predictions for the base pressure coefficient and centerline velocity with the experiments.

with the experiment. The SUQ-SST model predicts almost identical values of C_p as the experiment near the base center and the maximum error near the outer base edge is less than 10%.

Figure 7.15 shows the evolution of the centerline streamwise velocity normalized with the free stream velocity. Here, $z/R = 0$ represents the center of the base with the axial distances positive downstream. The axial location where the data crosses the $= 0$ line clearly defines the rear stagnation point since the other two measured velocity components are negligible along the centerline. This occurs at $x/R = 2.65$ in case of the experiment. The standard SST model predicts the reattachment point slightly upstream of the experiment while SUQ-SST model gives the location slightly downstream of the measurement. However, both the models predict the location within 5% of the measurement.

The experiment gives a maximum reverse velocity at $z/R = 1.5$ with a value of approximately 27% of the approach freestream velocity. The standard SST model gives similar reverse velocities as the experiment in the recirculation region, however, the location of the maximum reverse velocity is predicted upstream of the measured value at around $z/R = 1$. This trend is also evident in the Mach number distribution discussed earlier. On the other hand, SUQ-SST model predicts a slightly lower value of the maximum reverse velocity than the experiment, although the location is similar. Downstream of the reattachment location, both the models fit reasonably well with the experiment.

7.3 Summary

The two base flow configurations consisting of a Mach 2 Supersonic flow over a back-facing step geometry and a Mach 2.46 axisymmetric base flow are computed using the

SUQ-SST and standard SST $k-\omega$ models. A detailed comparison, including mean and turbulent quantities, between the model predictions and the available experimental data is shown for the two flow configurations. For the BFS geometry, both the models correctly capture all the flowfield details and give similar predictions for the mean quantities which match well with the experimental data. The results suggest the suitability of both the models in predicting the selected BFS flowfield configuration. On the other hand, for the supersonic flow over a blunt cylindrical body SUQ-SST model gives better predictions for the mean flow quantities, especially in the recirculation region. The SUQ-SST model gives improved predictions of the shear layer development with correct estimation of the TKE levels in the near wake. This is because of the SUQ-SST model's better treatment of the SSLI phenomenon. The standard SST model shows significant discrepancies in predicting the shear layer characteristics and the distribution of mean quantities in the recirculation region. The base pressure, which is one of the most important engineering quantities of interest for such flow configurations, is considerably underpredicted by the standard SST model with an error of 30%. The SUQ-SST model, on the other hand, gives an excellent match for the base pressure distribution with the experiment. Overall, the SUQ-SST model clearly shows a better performance compared to the standard SST model in predicting the axisymmetric base flow case.

Chapter 8

Application of SUQ-SST model to multi-jet interaction

During the ascent phase of a launch vehicle, the high-speed exhaust jets from the engines are in an underexpanded state. The structure of an underexpanded jet is complex and consists of jet boundary mixing layers, shock waves, and their interaction with each other. The clustering of engines makes the flowfield behind the launch vehicle base significantly complex as it leads to the mutual interaction of neighbouring underexpanded jets. The multi-jet interaction flowfield involves complex inviscid-viscous interactions including shock-boundary layer and shock-free shear layer interactions. The newly developed SUQ-SST model is first applied to predict the structure of a supersonic axisymmetric underexpanded jet. Comparisons with available experimental data of Seiner and Norum [155, 156] and the standard SST model with and without compressibility corrections (CC) are shown. Next, the SUQ-SST model is applied to a four-nozzle clustered rocket engine configuration typical of launch vehicles. A detailed CFD analysis of the multi-jet interaction flowfield for two different altitude conditions is shown and the predicted base flow characteristics are compared with the available wind tunnel data of Musial and Ward [131].

8.1 Supersonic underexpanded jet

The supersonic underexpanded jet of Seiner and Norum, discussed in Section 3.8 of Chapter 3, is used for the model validation purposes. Three-dimensional RANS simulations for the Mach 2 underexpanded jet have been performed and the flowfield is assumed to be axisymmetric, as per the previous works [36]. The standard SST and SUQ-SST models are used for the computations. The standard SST model is expected to operate in a

k - ϵ model framework for this flow configuration because of the absence of walls. Previous studies [36] have shown that the standard k - ϵ model is not able to correctly predict the jet structure for such flow configurations. Compressibility corrections in the form of compressible dissipation rate and pressure-dilatation are generally used to improve the flow predictions in such cases [127]. In this work, compressibility correction proposed by Wilcox [57] is used. The correction modifies the model coefficients associated with the dissipation terms in both the k and ω transport equations. The Wilcox compressibility correction model is given by,

$$\begin{aligned} C'_\mu &= 0.09(1 + 1.5f_{M_t}) \\ \beta' &= \beta - 0.09 \times 1.5 \times f_{M_t} \end{aligned} \quad (8.1)$$

Here,

$$\begin{aligned} f_{M_t} &= |M_t^2 - M_{t_0}^2| \text{ if } M_t^2 > M_{t_0}^2, \text{ and} \\ f_{M_t} &= 0 \text{ otherwise} \end{aligned} \quad (8.2)$$

and the turbulent Mach number $M_t^2 = (2k/a^2)$, $M_{t_0} = 1/4$, and a is the mean speed of sound. The coefficients C'_μ and β' are used instead of the original coefficients C_μ and β in the k and ω transport equations of the standard SST model.

8.1.1 Flowfield comparisons

Figure 8.1 shows the comparison of the underexpanded jet flowfield obtained using the standard SST model with and without the Wilcox compressibility correction (CC) and

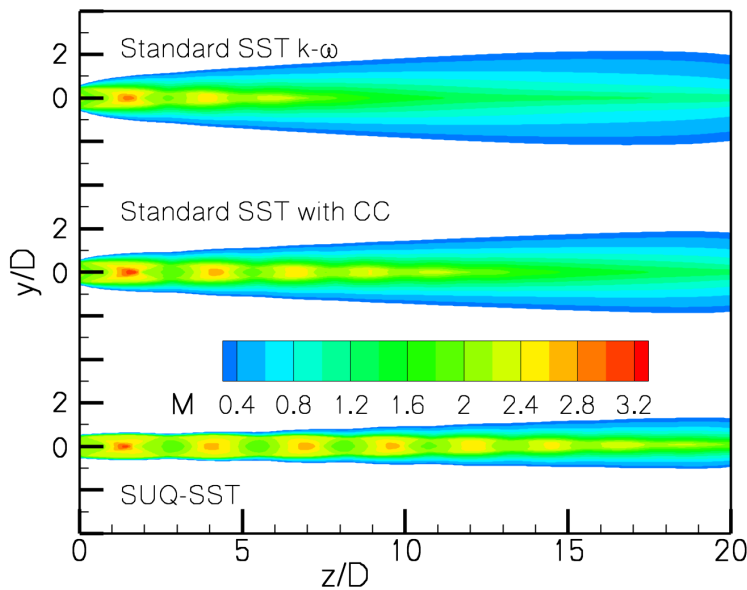


Figure 8.1: Comparison of the flowfield shown in terms of Mach number distribution obtained using different turbulence models for the Mach 2 underexpanded jet.

the SUQ-SST model. The flowfield is presented using the Mach contours, and Mach number values below 0.3 are not shown. The plot shows that the solution obtained using the standard SST model is more diffusive resulting in a rapid growth of shear layers and the predicted jet has a bigger size compared to the other two model solutions. The standard SST model predicts a rapid decay of shock-cells. Introduction of the Wilcox compressibility correction (CC) to the SST model results in a less diffusive solution and the jet size becomes slightly smaller with an increase in number of shock-cells. On the other hand, SUQ-SST model shows the least diffusive solution with a narrower jet that preserves the shock-cell structure further downstream.

8.1.2 Shock structures captured by the SUQ-SST model

Figure 8.2 shows the shock structures using the negative dilatation contours and the function F'_2 as obtained using the SUQ-SST model. The distribution of negative dilatation in Fig. 8.2(a) captures a number of pairs of oblique shock waves separated from each other by similar distances. The extent of each pair of the oblique shock waves in the axial direction depicts a single shock-cell structure. The position at which each shock wave pair terminates is the region of shock waves interacting with the jet boundary shear layers (SSLIs), after which the shock waves reflect from the jet boundaries as expansion waves. These expansion waves interact at the jet centerline and then reflect from the jet boundaries as oblique shock waves. The negative dilatation capture these set of interacting shock waves near the jet centerline and a regular reflection of the shock waves is evident without the formation of Mach stems. The function F'_2 , which is crucial for the correct implementation of the SUQ-SST model, captures all the shock structures in the flowfield, similar to the dilatation (see Fig. 8.2(b)).

8.1.3 Comparison of TKE distribution in the flowfield

Figure 8.3 plots the distribution of TKE as obtained using the three models. Here, TKE values are normalized using the nozzle exit jet velocity U_j . The standard SST model predicts more turbulence production in the shear layers compared to the other two models resulting in higher TKE values. The model predicts a rapid shear layer growth rate and the jet boundaries meet earlier than the other models. This results in faster shock-cell decay, as seen in Fig. 8.1. The compressibility correction reduces the TKE production and the shear layers meet at a later position than the standard SST model. On the other hand, SUQ-SST model further reduces the TKE production owing to the incorporation of shock/turbulence interaction physics, and the shear layers start to meet near the domain end.

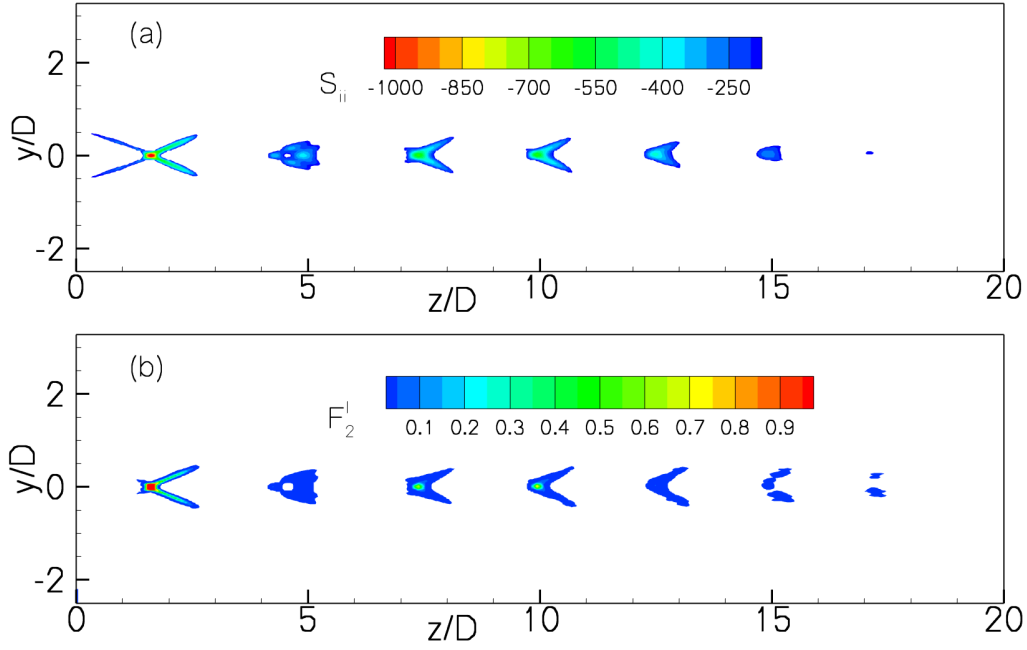


Figure 8.2: Distribution of the (a) mean dilatation and (b) boundary layer and shock detecting function F_2' in the supersonic underexpanded jet flowfield.

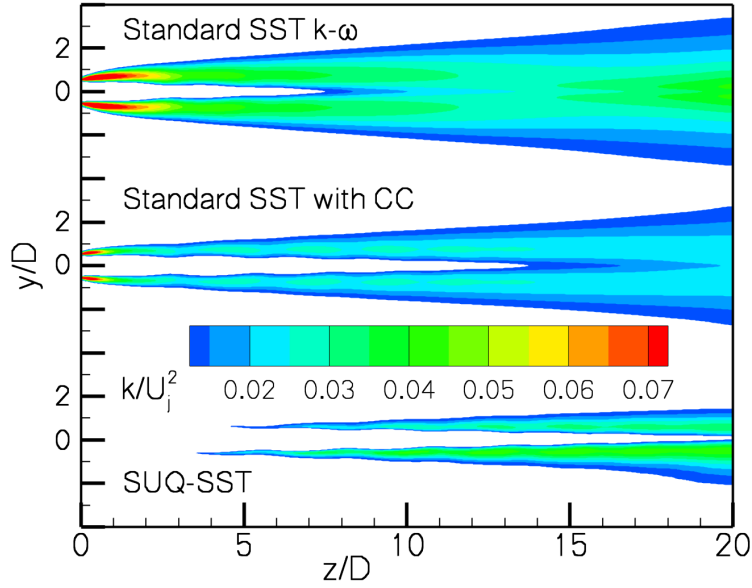


Figure 8.3: Comparison of TKE distribution obtained using different turbulence models in the Mach 2 underexpanded jet flowfield.

The variation of TKE, again normalized using U_j , along the jet centerline is shown in Fig. 8.4. The locations at which the TKE values start to rise can be considered as the place where the jet boundaries meet. As discussed earlier, the standard SST model significantly overpredicts TKE in the jet boundaries and hence shows a premature end of the shock-cell structure. The compressibility correction predicts the jet boundary meeting

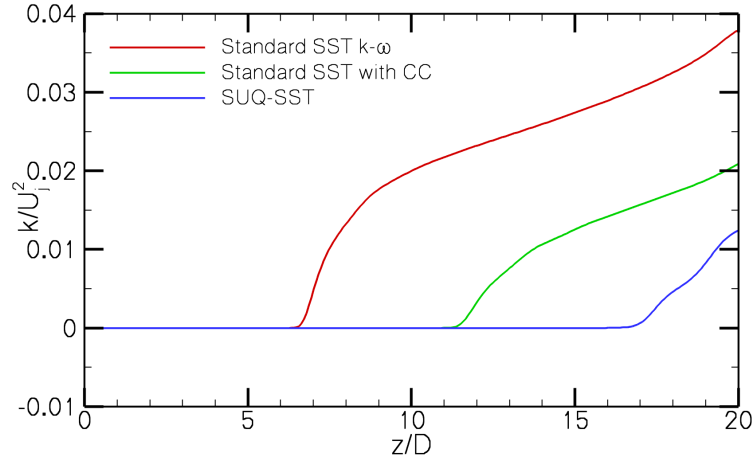


Figure 8.4: Comparison of TKE variation along the jet centerline obtained using different turbulence models for the supersonic underexpanded jet.

location further downstream because of the lower TKE values in the jet boundaries. SUQ-SST model shifts the location of the rise in TKE values along the jet centerline further downstream thereby sustaining the shock-cell structures to a longer distance than the other models.

8.1.4 Comparison of mean quantities along the jet centerline

Comparisons of the streamwise mean static pressure and Mach number variations along the jet centerline obtained using the three turbulence models and the measurements of Seiner and Norum [155, 156] are illustrated in Fig. 8.5. The mean static pressure values are normalized by the ambient value. The wave amplitudes for the pressure and Mach number are expected to decrease monotonically with distance from the nozzle exit, as observed from the experimental data. This is because of the dissipation that results from the turbulent mixing of the jet boundaries. All the model predictions exhibit the decaying oscillatory static pressure and Mach number variations caused by the interaction between the shocks and the mixing layers near the jet boundaries.

As discussed earlier, the standard SST model predicts a significantly rapid rate of mixing and, hence, foreshortens the cell lengths and dampens the wave strengths prematurely, as seen in Fig. 8.5(a). The model predicts only the first three shock-cells. Simulations with the compressibility correction indicate that the near-field region of the jet is influenced by compressibility, which reduces turbulent mixing rate, and improved results for the static pressure variation are obtained. The compressibility corrected model predicts the first six shock-cells, however, there exists a significant discrepancy in the amplitude and wavelengths of shock-cells compared to the measurements. SUQ-SST model

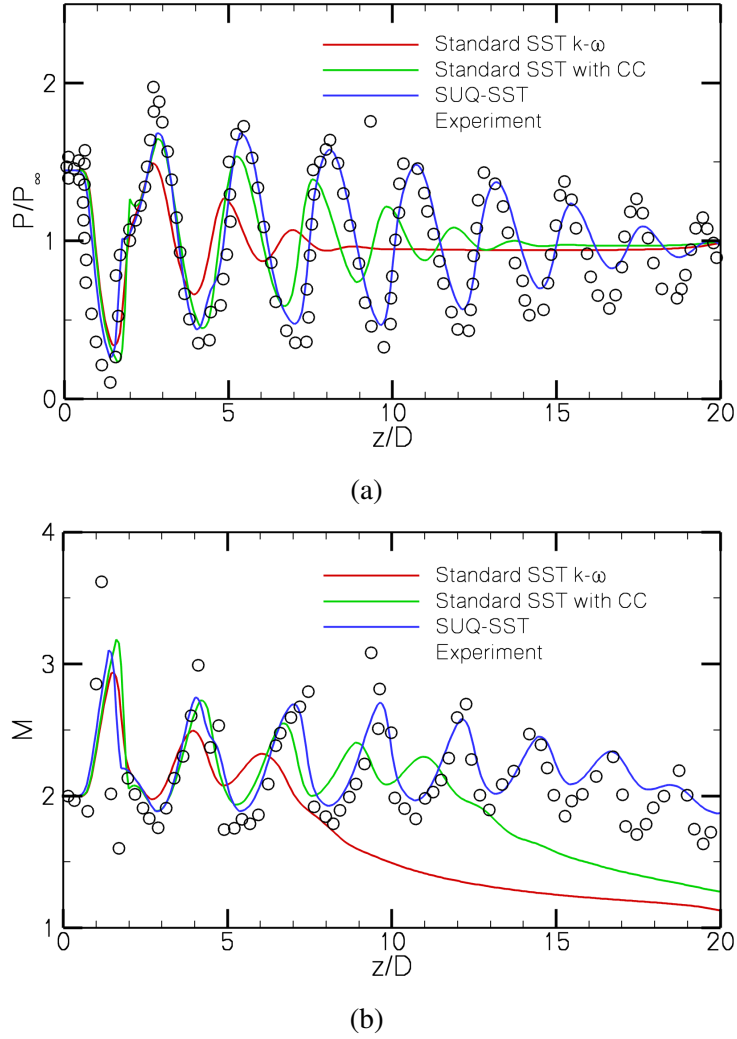


Figure 8.5: Variation of (a) normalized pressure and (b) Mach number along the jet centerline as obtained using different turbulence models and the experimental measurements of Seiner and Norum [155, 156] for the Mach 2 underexpanded jet.

considerably improves the results for pressure variation and gives a better match with the experiment. The model is able to correctly predict the shock strengths along with the wavelengths of the shock-cells. The present SUQ-SST model results are similar to the results of shock-unsteadiness $k-\epsilon$ model of Sinha et al.[34] presented in [36]. This is expected because both the models share similar shock/turbulence interaction physics.

The axial variation of Mach number along the jet centerline, as presented in Fig. 8.5(b), shows a similar trend for the model predictions as seen for static pressure. The standard SST model gives Mach numbers that decay too rapidly, and the use of the compressibility correction shows slight improvement, but the solution tends to be slightly out of phase with the measurements. SUQ-SST model predictions agree quite well with the experiment, but the last two shock-cell amplitudes are still underpredicted compared

to the experiment. However, the overall agreement of the SUQ-SST model predictions with the measurements is reasonably fair and much improved over the results obtained using the standard and compressibility corrected SST model.

8.2 Multi-jet interaction

The high-speed flow configurations computed in the earlier validation studies, like SBLIs, flow over BFS and blunt cylindrical body, and underexpanded jet, can be considered as unit or canonical flows for the more realistic multi-jet interaction flowfield. The earlier validation studies established the superior performance of the SUQ-SST model over the standard SST model and the new model is able to consistently give good results for the entire range of flow configurations considered. So, the performance of the newly developed SUQ-SST model is tested in the more complex multi-jet interaction flowfield for a four nozzle clustered rocket engine configuration typical of launch vehicles. The validation test case considered is discussed in Section 3.9 of Chapter 3 and two different NPR conditions of 990 and 3760 are considered for the model validation. Three-dimensional RANS simulations are performed and the flowfield is assumed to be axisymmetric, similar to works of Nallasamy et al. [142]. The two NPR cases represent two different altitude conditions; 990 NPR representing a low altitude of about 20 km and NPR of 3760 characterizing a higher altitude of about 30 km. Comparisons between SUQ-SST model predictions and experimental measurements are shown for the base pressure distribution, and the experimental data is taken from [131]. Simulations with standard SST model are not performed because of the constraints on the available computational resources.

8.2.1 Flowfield at two different altitudes

The multi-jet interaction flowfield at simulated high and low altitudes is shown using Mach contours in a plane that passes through the centers of the diametrically opposite nozzles in Fig. 8.6. Each of the jets in the two altitude conditions are in an underexpanded state. Interaction between the freestream flow and jets and between neighbouring jets is a crucial feature of this flowfield. The flow structure inside each of the underexpanded jets for the two altitude cases is similar in the axial direction, however the jet sizes are different. The interacting jets have smaller axial lengths at simulated low altitude and the supersonic core decays faster compared to the high altitude case. This is because of the comparatively higher ambient pressure at low altitude that causes larger shear and normal forces which pinches the jet sooner.

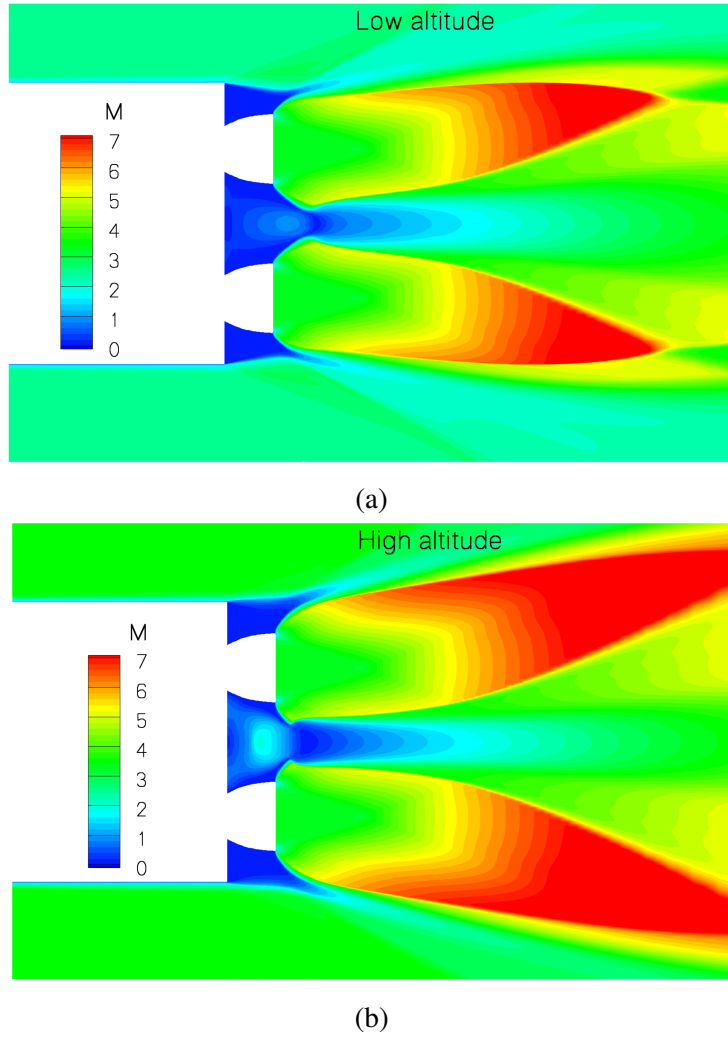


Figure 8.6: Mach number contours showing the multi-jet interaction flowfield at two different altitudes : (a) low altitude case with NPR = 990 and (b) high altitude case with NPR = 3760. The flowfield is shown in a plane passes through the centers of the diametrically opposite nozzles.

The radial expansion of each of the jets is different for the two altitude cases; it is more for the high altitude case. The difference in jet expansions causes a difference in interaction pattern between the jets and the freestream. At simulated high altitude, the radial expansion of the jets is such that their interaction with the separated shear layer from the rocket forebody takes place at a plane which is almost parallel to the rocket forebody. In fact, the flow separates slightly ahead of the base corner and a weak oblique shock wave is formed. On the contrary, flow expands around the base corner for the low altitude case.

The interaction between adjacent jets and the resulting reverse flow or the updraft plume pattern is also considerably different between high and low altitudes. It can be

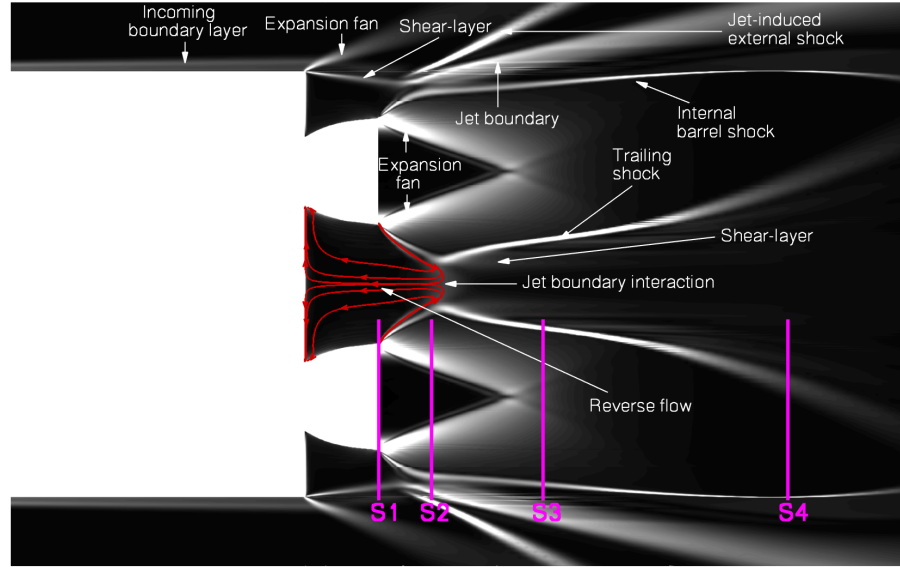


Figure 8.7: Density gradient magnitude showing different flow-features in the multi-jet interaction flowfield captured by the simulations using SUQ-SST model.

seen from Fig. 8.6 that the interaction between the jets at high altitude takes place closer to the base compared to the low altitude case. Also, a well developed supersonic updraft plume can be seen for the high altitude case whereas a near sonic plume is developed at the low altitude. However, despite the difference in speeds of the updraft plume for the two altitude cases, a common flow pattern exists. The updraft plume first accelerates till some distance away from the base and then it decelerates near the base.

8.2.2 Flow evolution in the axial direction

Figure 8.7 shows the multi-jet interaction flowfield using the density gradient magnitude contours in the yz symmetry plane, i.e. the plane passing through the centers of the diametrically opposite nozzles. The flowfield is for the low altitude case and similar flow structures in terms of the shear layers, shock waves, expansion fans, and their interactions are observed for the high altitude case. The flowfield and shock structure as obtained using the CFD simulations (Fig. 8.7) employing SUQ-SST model are similar to those described in the schematic shown in Fig. 2.10 in Chapter 2. The important flow-features such as the shear layer and expansion fan emanating from the base corner, interaction of the jets and the inflection point, trailing edge shocks formed due to the jet interaction, and jet boundaries are successfully captured in the simulations. The different viscous and inviscid interactions, as seen in Fig. 8.7 and described in Table 1 in Chapter 2, give rise to the complexity of the multi-jet interaction flowfield.

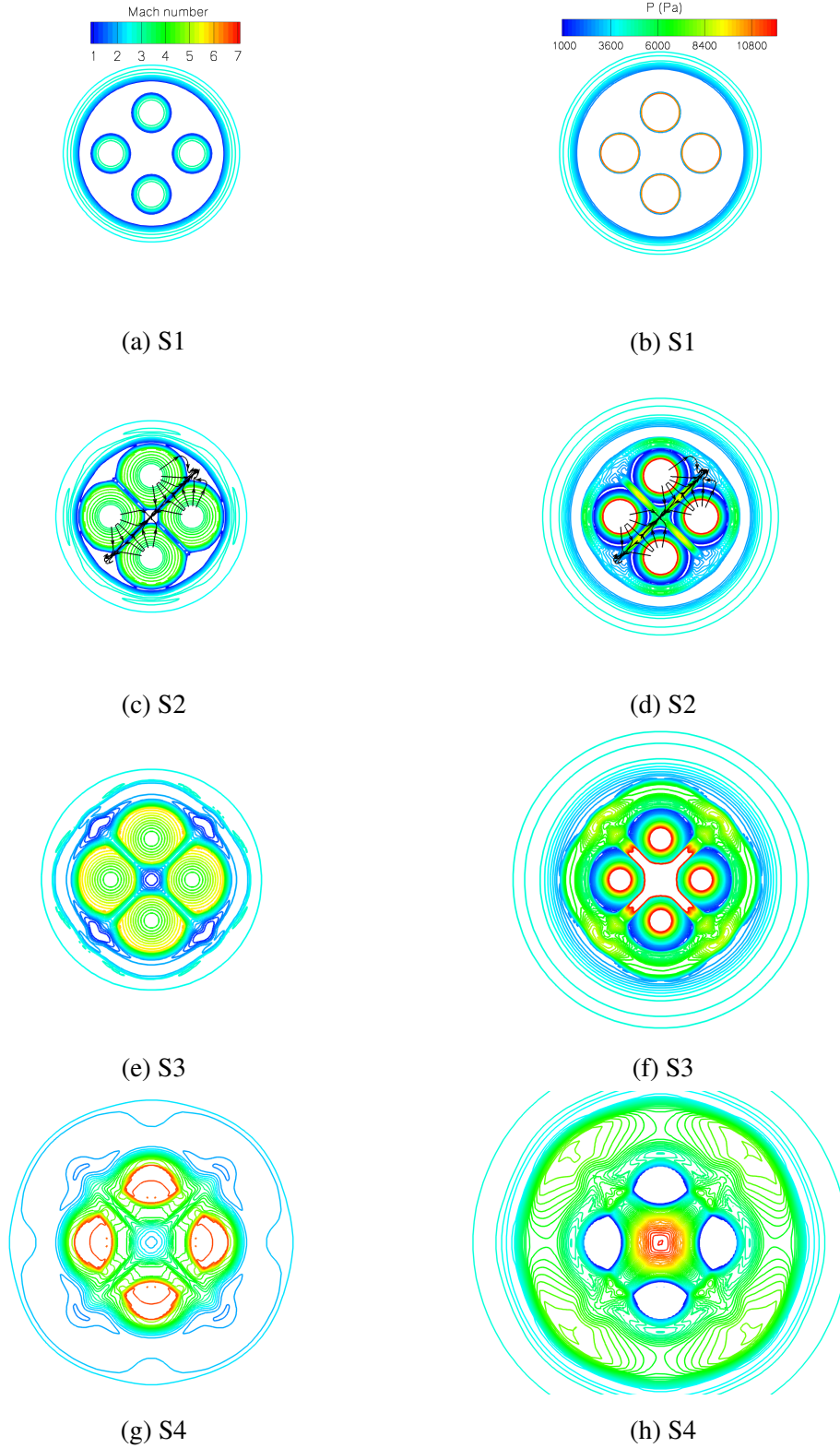


Figure 8.8: (a), (c), (e), and (g) Mach number contours and (b), (d), (f), and (h) pressure contours at different xy cross-sections along different axial locations in the multi-jet interaction flowfield (Refer Fig. 8.7 for their exact locations).

Figure 8.8 shows Mach number and pressure contours in the xy cross-sections, i.e. planes parallel to the base, at different axial locations; refer Fig. 8.7 for their exact locations. The evolution of the four jets just after nozzle exit plane till some distance away from the inflection point is presented. Figures 8.8(a) and 8.8 (b) show the flow structure in a plane just after the nozzle exit. The jets are fairly circular and they assume the nozzle exit geometry shape. Expansion fan generated at the base shoulder can be seen surrounding the external shock, free-shear layer from the base corner, and the four jets. Subsonic flow is seen in between the nozzles with fairly uniform pressure distribution. Supersonic pockets are seen in the symmetry planes in between the two adjacent jets.

The radial expansion of the jets results in their mutual interaction downstream of the nozzle exit plane and the interaction location is called as the inflection point. For a symmetric geometry and flow configuration, such as the one considered in this work, the interaction occurs in the symmetry planes (see Figs. 8.8 (c) and 8.8 (d)). The jet shear layers get flattened due to the interaction while the outer boundaries remain circular. A saddle point is observed in the interaction plane with part of the jet flow turned towards the center of the four nozzles. Another part of the jet flow moves radially outward towards the jet boundary and results in a slight protrusion in the external shock wave in the jet interaction plane.

Jet boundary distortion from circular to a pear shape increases downstream, as seen in Figs. 8.8 (e) and 8.8 (f). The flow in the jet interaction plane that turns radially inwards toward the model centerline results in a pressure increase and the maximum pressure is observed along the model centerline as seen in Figs. 8.8 (f) and 8.8 (h). A high pressure region along the symmetry planes between the two adjacent jets is also observed. By comparison, a low pressure and high Mach number region is observed in the core of each jet. It is influenced by the trailing shock and the internal barrel-shaped shock. The shear layer enclosing all four jets take a square cross-section farther away, as seen in Figs. 8.8 (g) and 8.8 (h). Prominent shear layers also develop along the symmetry planes between the adjacent jets.

8.2.3 Flow near the base shoulder

At simulated low altitude, the supersonic freestream Mach 2.5 flow over the cylindrical body expands to about Mach 2.8 around the base shoulder and separates as a free-shear layer (see Fig. 8.9(a)). The freestream pressure reduces by a factor of about 1.6 after this expansion and the separated free-shear layer interacts with the jet boundary away from the nozzle exit plane. The separated free-shear layer and the nozzle wall enclose a subsonic region. The pressure across the free-shear layer and in the enclosed subsonic region is

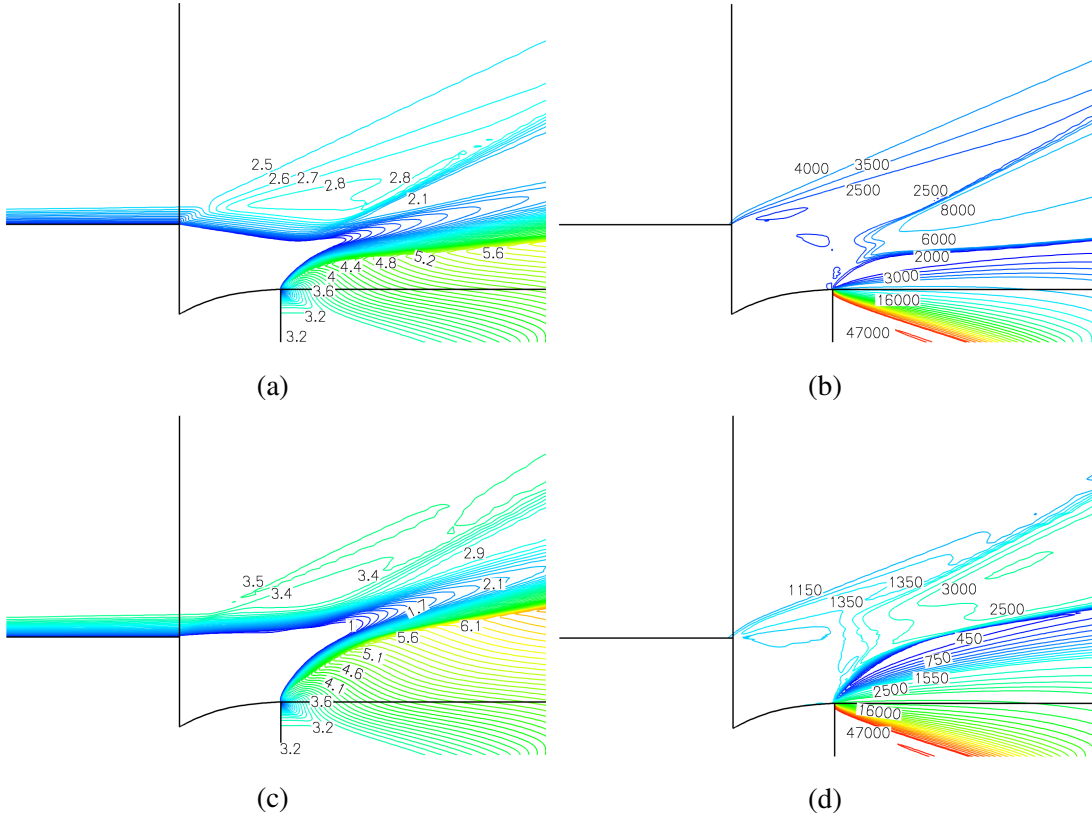


Figure 8.9: (a), (c) Mach number and (b), (c) Pressure (Pa) contours showing the expansion at the base shoulder in the yz symmetry plane. Here, (a) and (b) correspond to the low altitude case, and (c) and (d) show the multi-jet interaction flowfield at the simulated high altitude.

nearly constant and is similar to the pressure after the expansion around the base shoulder. On the other hand, a weak shock wave is seen and the flow is compressed instead of expanding around the base corner at high altitude. The Mach number reduces to about 3.3 from its freestream value of 3.5 across this shock wave and pressure increases by a factor of about 1.2. In the high altitude case, the pressure across the free-shear layer and in the enclosed subsonic region is also nearly constant and is similar to the pressure after the weak oblique shock around the base shoulder.

The interaction between separated free-shear layer and the jet boundary results in the formation of a jet-induced external shock outside the jets and a barrel-shaped shock inside the jets for both the low and high altitude cases. The pressure values after the external shock are higher than the ambient pressure and hence the effective NPR as experienced by the jets is smaller than the corresponding NPRs of 990 and 3760 for these cases. At simulated low altitude, the external shock wave interacts with the expansion fan emanated from the base shoulder and increases the pressure by a factor of about 2.4. On the other

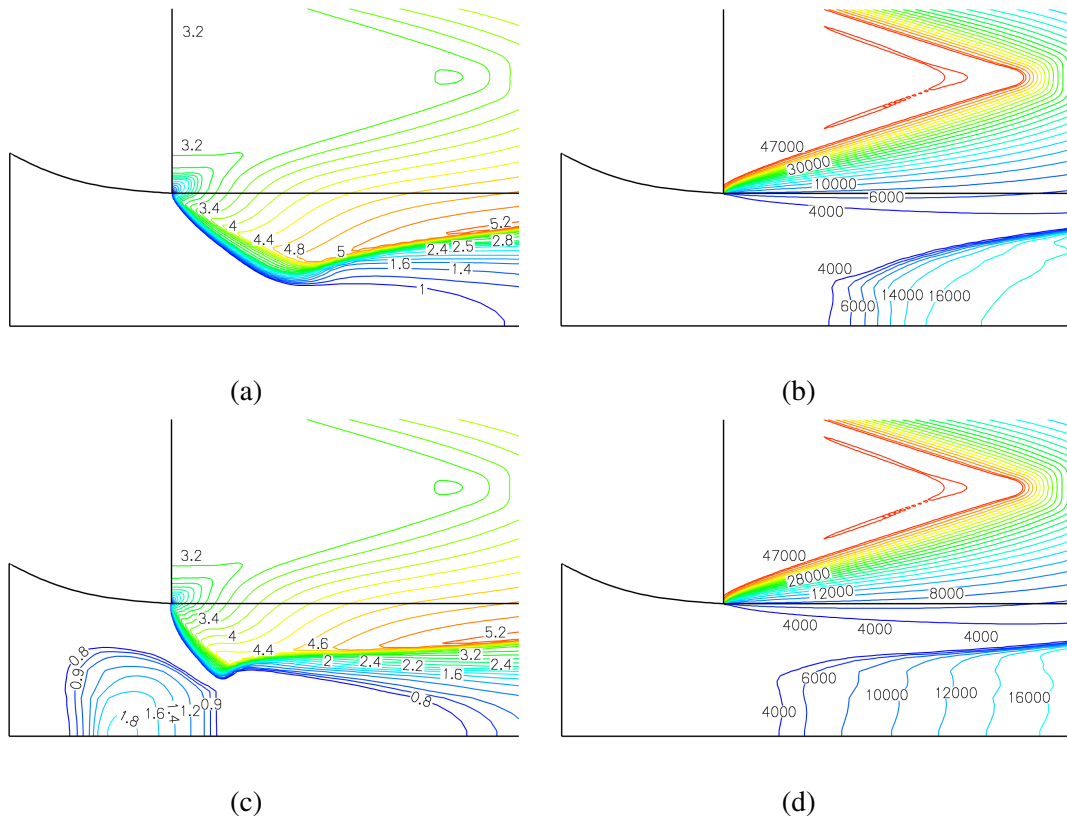


Figure 8.10: (a), (c) Mach number and (b), (d) Pressure (Pa) contours showing the interacting jet-boundary expansion in the yz symmetry plane for the multi-jet interaction cases. Here, (a), (b) represent the low altitude case, and (c), (d) correspond to the high altitude case.

hand, for the high altitude case, the external shock wave interacts with the oblique shock wave emanated near the base shoulder and it further increases the pressure by a factor of about 2. For both the altitude conditions, the strength of the external shock wave reduces as we go away from the nozzle exit. The interactions between the freestream flow and the jet boundary cause the jet to experience variable ambient pressure conditions. This results in a pressure distribution that varies along the jet boundaries.

8.2.4 Flow near interacting jets

Figure 8.10 shows the expansion of the lower part of the jet flow in terms of the Mach number and pressure contours in the yz symmetry plane for the two altitude cases. The lower part of jet flow undergoes greater expansion for the high altitude case than at low altitude, and it is evident from the jet boundary placement in Figs. 8.10(a) and (c). As a result, the position of the trailing shock wave is also different between the two cases and it is closer to the nozzle exit plane for the high altitude case. A supersonic region

with Mach numbers up to 1.9 is observed in the reverse flow area enclosed between the jet boundaries, nozzle walls, and the base for the high altitude case. Also, a standing normal shock wave is observed in this region before the base. Consequently, there is a considerable variation in pressure in this reverse flow region for the high altitude case. On the other hand, the reverse flow never becomes supersonic for the low altitude case, and the pressure in the reverse flow area remains almost constant with values less than the corresponding ambient conditions.

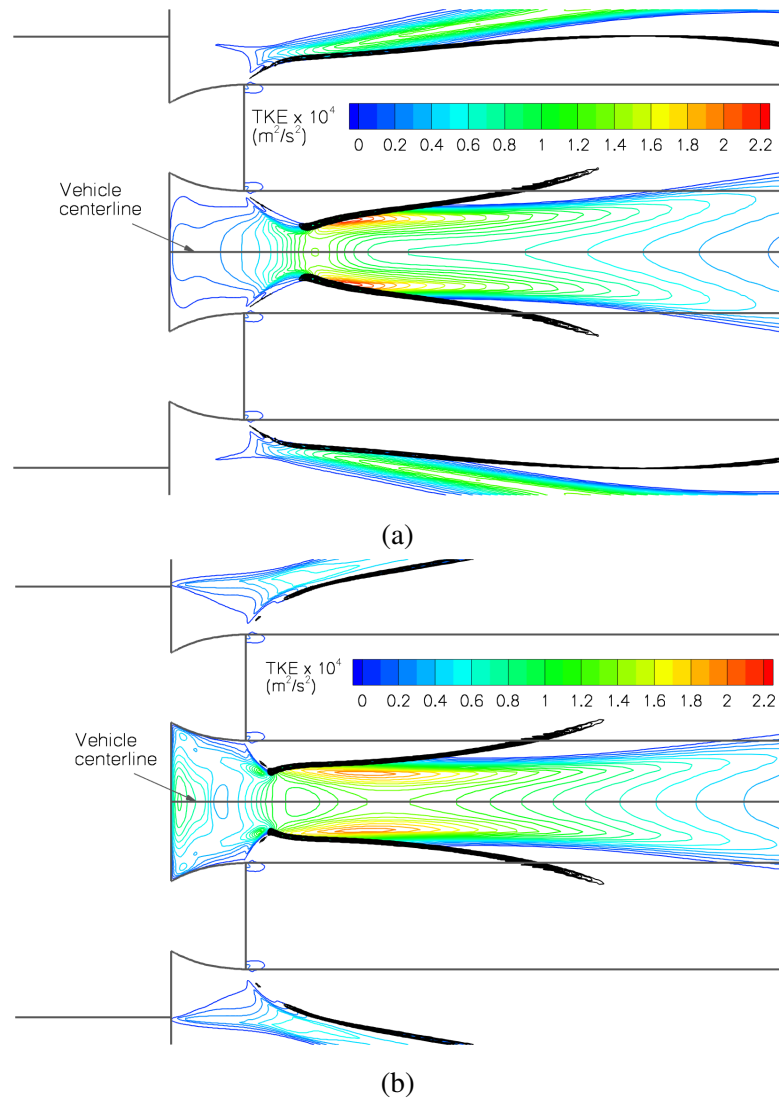


Figure 8.11: Comparison of the distribution of TKE in the multi-jet interaction flowfields as obtained using the SUQ-SST model at (a) low altitude and (b) high altitude. Shocks are identified using the negative dilatation contours shown in black color.

8.2.5 Distribution of turbulence kinetic energy in the flowfield

Figure 8.11 shows the TKE distribution in the multi-jet interaction flowfield for the two altitude conditions obtained using the SUQ-SST model. The dilatation contours, shown using black color, identify the high compression regions of internal and trailing shock waves. The trailing shock waves interact with the turbulent jet boundaries and amplify the turbulence levels in these shear layers in each of the jets at low and high altitudes. TKE distribution across the trailing shocks is almost similar between the two altitude cases, however a significant difference is observed in the region of reverse flow enclosed between the jet boundaries, nozzle walls, and the base. Turbulence levels near the base are significantly higher for the high altitude case which leads to larger variations in base pressure distribution than the low altitude case.

8.2.6 Centerline variations of mean velocity and pressure

The variation of streamwise velocity and static pressure along the vehicle centerline at the two altitude conditions obtained using the SUQ-SST model is shown in Fig. 8.12. The mean velocity and pressure are normalized using their respective freestream values. The location at which the velocity crosses the zero mark can be considered as the inflection point where the jets interact. It can be seen from Fig. 8.12(a) that the inflection point shifts slightly towards the base at high altitude. Also, the maximum velocity in the recirculation region is significantly higher for the high altitude case than the low altitude. However, in both the altitude conditions, the velocity recovers to almost similar values in the near wake. Pressure variation in Fig. 8.12(b) shows that the pressure at the inflection point is significantly higher at the high altitude than the low altitude case. The normalized base pressure (at $z/R = 0$) is also considerably higher for the high altitude case than the low altitude. Post the multi-jet interaction location and away from the base, the pressure increases and then decreases in the redeveloping wake for both the cases.

8.2.7 Base pressure comparisons

Figure 8.13 shows the distribution of pressure on the base of the rocket engine configuration with four nozzles obtained using the SUQ-SST model for the low and high altitude cases. The pressure is normalized using the corresponding freestream values at these simulated altitudes. For both the altitude cases, the pressure is significantly higher at the center of the base than in the other regions. This is because of the impingement of the high velocity reverse flow or the updraft plume, which is caused by the multi-jet interaction, onto the base center. A nice circular region of high pressure is observed at the

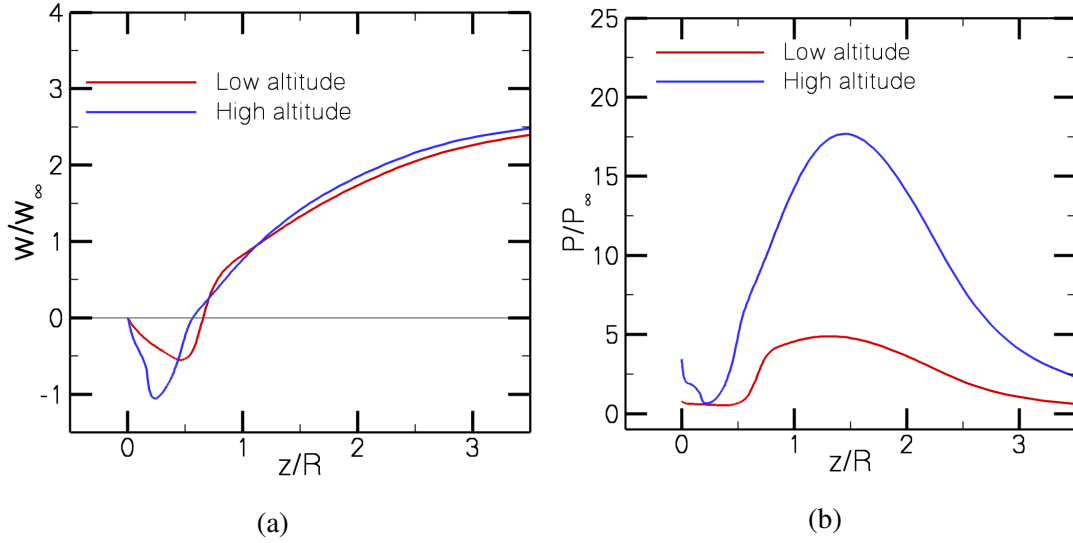


Figure 8.12: Comparison of normalized (a) streamwise velocity and (b) static pressure along the vehicle centerline obtained using SUQ-SST model for two different altitude conditions.

base center for the low altitude case, whereas a non-circular pressure hot spot is observed at high altitude. Pressure values at the base center are about $(3/4)^{th}$ of the ambient value for the low altitude case. On the other hand, pressure on the center of the base reaches as high as 3 times the ambient value at the simulated high altitude. The updraft plume at low

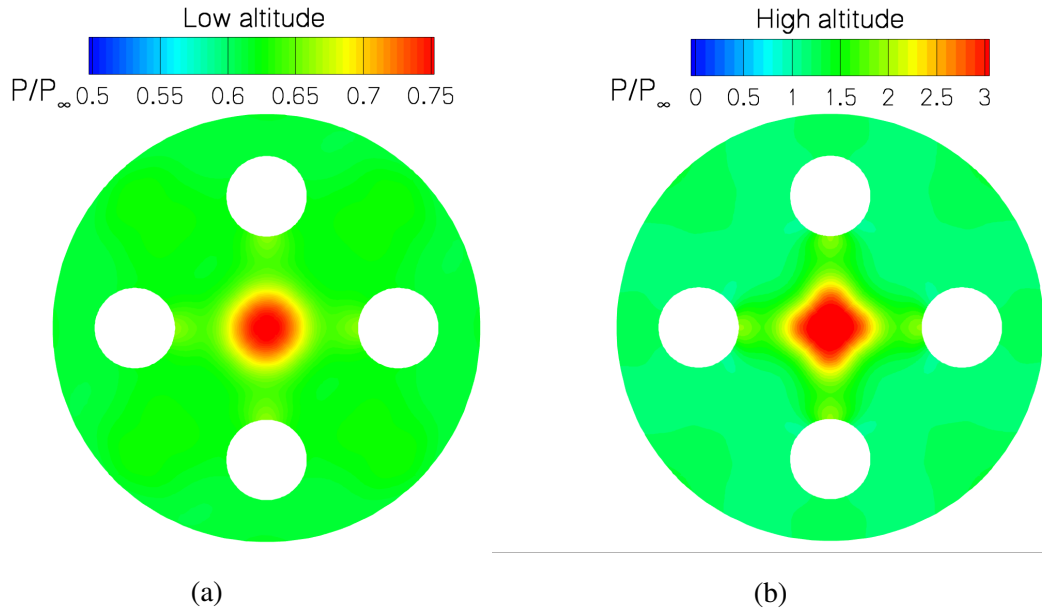


Figure 8.13: Base pressure distribution obtained using SUQ-SST model at two different altitudes : (a) low altitude case with NPR = 990 and (b) high altitude case with NPR = 3760.

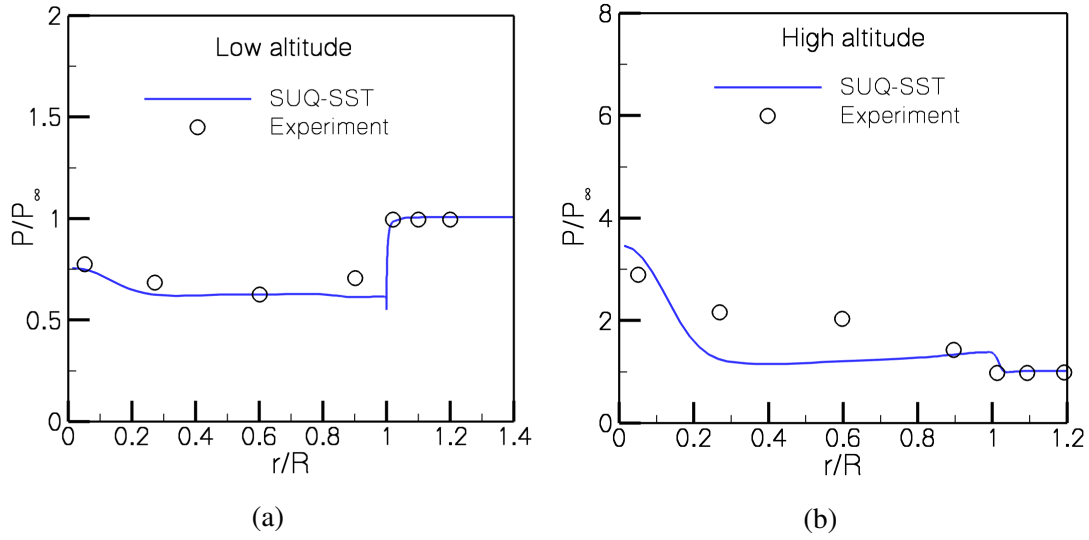


Figure 8.14: Comparison of radial variation of normalized base pressure between the SUQ-SST model and measurements for the two altitude cases. The results are shown along a symmetry plane passing midway between the centers of two adjacent nozzles.

altitude achieves near sonic values before impinging on the base center, while it reaches to about Mach 2 values resulting in such extreme base pressure environments for the high altitude case.

The radial variation of the normalized base pressure along a symmetry plane that runs midway between the centers of two adjacent nozzles is shown in Fig. 8.14. Results obtained using SUQ-SST model are compared with the wind tunnel data of Musial and Ward [131]. Here, $r/R = 0$ represents the base center and $r/R = 1$ represents the edge of the base. The experiments show smaller pressure values at the base center than the freestream for the low altitude case, whereas the trend is reversed at the simulated high altitude. The SUQ-SST model gives an excellent match of the radial pressure variation with the experimental data for the low altitude case. Model prediction gives an error of less than 2.5% for the pressure values at the center of the base, and the overall trend is captured correctly for the low altitude case. On the other hand, for the high altitude case, SUQ-SST model prediction shows some discrepancies in matching the experimental values, however the model correctly captures the trend of the base pressure variation. At the simulated high altitude, the SUQ-SST model overpredicts the pressure value at the base center by about 15% compared to the experiment, which is reasonably acceptable for this case. Overall, the SUQ-SST model gives fairly good results for the two altitude cases and matches the trends observed in experimental measurements for the base pressure distribution.

8.3 Summary

A detailed CFD analysis of the supersonic underexpanded jet and multi-jet interaction flowfield at two different altitude conditions obtained using the SUQ-SST model is presented. The flowfield of a Mach 2 underexpanded cold jet exhausted into still air is computed using the standard SST and SUQ-SST models. Wilcox compressibility correction is also used in the standard SST model to test its performance. Comparisons of the model predictions with the experimental data of Seiner and Norum [155, 156] are shown for the axial variation of Mach number and pressure. The SUQ-SST model shows a superior performance in predicting the jet structure compared to the other models. Next, the SUQ-SST model is applied to a realistic geometry of four nozzle clustered rocket engine configuration with multi-jet interaction. Two different altitude conditions are considered and the experimental data of Musial and Ward [131] is used to validate the results. The SUQ-SST model is shown to predict correct flow topologies and gives reasonably good results for base pressure distribution for both the altitude cases.

Chapter 9

Summary and Future work

9.1 Summary

Turbulence modelling is a crucially important aspect of the RANS-based CFD techniques routinely used for the engineering predictions of shock-dominated flows found in many of the technological applications, e.g. aerospace-related high-speed vehicles. However, conventional turbulence models do not take into account the physics of shock/turbulence interaction, an important phenomenon observed in the high-speed flows. This dissertation presents the development and extensive CFD validation of two advanced turbulence models for shock-dominated realistic flows. The model development strategies followed here build upon the already existing two-equation eddy viscosity models and modelling improvements are proposed considering the flow physics. The newly developed advanced models can be easily implemented in existing CFD codes. Validation studies involve aerospace-related applications, although the models can also be used for other technological applications like the inertial confinement fusion. Major focus of the turbulence model predictions in this work include position of the shock waves in the flowfield, extent of the separation and recirculation regions, and surface properties like surface pressure, skin-friction, and base pressure.

At first, development of a physically sound and numerically robust two-equation turbulence model for high-speed flows with shock waves is presented. The model development is shown in the k - ϵ modelling framework, but it can be easily extended to the k - ω framework. A new physically-consistent source term is proposed to model the shock-unsteadiness damping of turbulence kinetic energy. The turbulence dissipation rate equation is based on the physical effects of mean compression and enhancement of fluid viscosity across a shock wave. Unlike the previous physics-based turbulence models, the current model parameters are independent of the shock strength and hence do not require

prior knowledge of the upstream mean flow Mach number. This can be of significant advantage while applying the model equations to realistic flows with multiple shock waves and their interaction with boundary layers. The model equations are also cast in a conservative form so as to eliminate numerical error at flow discontinuities. The model is tested extensively in canonical shock-turbulence interaction and found to match the available DNS data well over a range of Mach numbers. The superior numerical characteristics, in terms of grid convergence of the solution, is also demonstrated.

The model is then systematically extended in the $k-\epsilon$ and $k-\omega$ frameworks for applications involving shock-turbulent boundary layer interactions. Extensive model validation is carried out by computing six canonical SBLI test cases, with varying shock strengths. Comparisons of wall pressure and skin-friction coefficient with the available experimental data show that the new model gives significant improvements over the corresponding standard $k-\epsilon$ and $k-\omega$ models. The new model matches the predictive accuracy of the shock-unsteadiness model of Sinha et al. [30], but with a significantly reduced computational effort. There is a large saving in human effort and more than 50% reduction in computing time, as compared to the shock-unsteadiness model of Sinha et al. [30]. The physical accuracy and numerical robustness of the new model thus make it a better choice over existing turbulence models for SBLI flows.

Next, development of an advanced form of SST $k-\omega$ model is presented by considering the physics of shock/turbulence interaction. The new model, called as SUQ-SST, is developed by introducing features of EARSM in the SST framework. A quadratic relation between the Reynolds stresses and strain and vorticity tensors by Rung et al. [73] is used for this purpose. The structure parameter is modified and it depends on the magnitude of local mean velocity gradients and the anisotropy parameter. The modified structure parameter increases in response to a shock-induced adverse pressure gradient, as observed in SBLI experiments. However, at a shock wave, the magnitude of local mean velocity gradients can become significantly high that may lead to unphysical values of the structure parameter. To prevent this, the value of the structure parameter at a shock wave is limited using the shock-unsteadiness (SU) limiter. The limiting value is obtained from a Menter-like analysis of the shock-unsteadiness model of Sinha et al. [30] at a shock wave. The SU limiter is similar, but more formal and based on physical analysis, than the use of ad-hoc eddy-viscosity limiters found in the literature.

The new advanced form of SST model retains the capabilities of the standard SST model in resolving a zero-pressure-gradient turbulent boundary layer. The model is tested against a wide range of high-speed flows including two-dimensional canonical SBLIs consisting of UFAST Mach 2.3 experiment, Mach 5 impinging shock cases, and Mach

8 compression corner flows, and base flows consisting of Mach 2 flow over a backward facing step and Mach 2.4 flow past a cylinder with a blunt base. A Mach 2 underexpanded axisymmetric jet exhausted into still air is also considered as a test case. Finally, a more realistic application in the form of a clustered four-nozzle rocket engine configuration with multi-jet interaction typical of launch vehicles is used to validate the performance of the model. The selected test cases cover a wide range of flow regimes from supersonic to hypersonic speeds, and near incipient to fully separated flows involving SBLIs and SSLIs.

The SUQ-SST model is able to accurately capture the behaviour of the structure parameter in the SBLI regions in accordance with the UFAST experimental data. The added physics leads to significantly improved predictions of the mean flow properties along with turbulent quantities like TKE, Reynolds stresses and their anisotropy compared to the standard SST model for the considered test cases. The SUQ-SST model is able to consistently predict the separation shock location and separation bubble size in case of SBLI flows. The model gives accurate predictions of the mean flowfield in case of the underexpanded jet. It is able to correctly resolve the separation and recirculation regions along with base pressure distributions in case of base flows and multi-jet interaction cases at two different atmospheric conditions. It thus clearly outperforms the standard SST model in predicting a wide range of high-speed shock-dominated flows of practical relevance. The SUQ-SST model requires minimal extra computational cost and time than the original SST model and provides a significant step forward in the Reynolds-averaged simulation of shock-dominated flows in practical applications.

9.2 Contributions

The dissertation discusses in detail the systematic development and extensive CFD validation of two advanced turbulence models for high-speed shock-dominated flows. The first advanced model is a physically consistent and numerically robust version of the physics-based shock-unsteadiness model of Sinha et al. [30], while the second is an advanced and improved version of the widely popular SST $k-\omega$ model. Extensive CFD validation of the newly developed models is carried out using a wide range of supersonic and hypersonic flows encountered in the flight of high-speed aerospace vehicles. The major contributions of this work are :

- Development of a **numerically robust local form of shock-unsteadiness model** with extensive validation in the $k-\epsilon$ and $k-\omega$ model frameworks for canonical STI and two-dimensional SBLI flows.

- Development of an **advanced SST k - ω model** for shock-dominated flows and the modelling strategy involves
 - Introduction of *EARSM features in SST model framework* for compressible flows.
 - Formulation of a *new relation to compute structure parameter* based on local mean flow properties.
 - Prescription of *limiting values of structure parameter at a shock wave* based on a physical analysis of the SU model.
- Formulation of a **new algebraic form of shock detecting function** which works well for a wide range of high-speed flow configurations.
- **Implementation and verification** of SST k - ω , EARSM, and SUQ-SST models in 2D and 3D in-house codes.
- **Extensive validation** of the advanced SST model for 2D SBLI, base flows and under-expanded jets.
- Model application to a realistic geometry of four-nozzle clustered rocket engine configuration with multi-jet interaction typical of launch vehicles is also shown for which such an advanced EARSM-based model has never been applied before in the available literature.

9.3 Future Work

The work presented in this dissertation can be extended further in a number of ways. Following are some of the ideas inspired by the present work that can be pursued in the future :

- **Concerning the local shock-unsteadiness model**

The shock-unsteadiness correction is derived in a co-ordinate frame where the shock is steady in the mean and unsteady oscillations about its mean location are included in the shock-unsteadiness term. The model is expected to work well on all reference frames where the mean flow is steady, i.e., the mean shock is stationary with respect to the reference frame. This includes different Cartesian co-ordinate systems with arbitrary orientation, as the local SU model equation is

shown to be invariant under rotation of the reference frame [61]. Extension of the model to unsteady mean flow problems, where the mean shock itself moves through the medium, requires additional development. Application of the model to Inertial confinement fusion can then be undertaken.

- **Concerning the SUQ-SST model**

A different EARSM formulation, e.g. EARSM by Wallin and Johansson [70], can be used to develop the advanced SST model by following a similar modelling strategy adopted in this work. Also, a correct prediction of the shock-induced separation bubble size generally results in lower values of post-reattachment skin-friction than expected, and this is a common problem seen in case of two-equation eddy viscosity models. One of the ways of tackling this issue can involve correct prediction of structure parameter values in the post-reattachment regions. This can be done by formulating an improved relation for the structure parameter.

- **Concerning heat transfer predictions**

The present work concerns with the accurate predictions of the separation regions, surface pressure, skin friction, and base pressure. However, no attempts are made to predict surface or base heat transfer-rate, which is another important engineering quantity of interest for the high-speed flow applications. Generally, a constant value of Prandtl number in the conventional gradient diffusion hypothesis based heat transfer modelling framework is found to be insufficient to correctly predict heat transfer rates, even if separation and surface pressure are accurately captured. One computationally cheaper way to address this problem is by using a variable Prandtl number model from the existing literature, e.g. the algebraic variable Prandtl number model of Roy and Sinha [94].

- **Model application to a practical geometry**

The validation test cases involved in the present work majorly include canonical geometries involving two-dimensional or axisymmetric mean flows. Application of the models to practical complex geometries like scramjet engine and launch vehicle configurations, which involve three-dimensional SBLIs or SSLIs, can be carried out.

- **Model application to reacting flows**

All the simulations carried out in this work are based on the calorically perfect gas assumption. However, in practical scenarios, chemical reactions may occur in the flowfield which can influence the mean flow and turbulence statistics. Examples can include shock waves interacting with a chemically reacting turbulent boundary layer or high temperature plume gases with alumina particles interacting with each other. Extension of the newly developed models to such scenarios can also be considered.

Appendix A

Inflow conditions for canonical shock/turbulence interaction computations

For steady one-dimensional uniform mean flow upstream of the shock wave, the variations in k and ϵ are governed by,

$$u_1 \frac{\partial k}{\partial x} = -\epsilon \quad (\text{A.1})$$

$$u_1 \frac{\partial \epsilon}{\partial x} = -c_{\epsilon 2} \frac{\epsilon^2}{k} \quad (\text{A.2})$$

The above equations are in non-dimensional form and the respective characteristic quantities used for non-dimensionalization are mentioned in Section 3.1 of Chapter 3. These equations are obtained by neglecting the production terms from Eqs. (4.8) and (4.15), as there is no gradient in the mean velocity in this region. Also, the velocity is constant in this region and is equal to u_1 , where $u_1 = M_1$ because of the non-dimensionalization considered. The tilde over u_1 is neglected and $\epsilon_s \simeq \epsilon$ is taken. Eqs. (A.1) and (A.2) can be written as,

$$\frac{\partial \epsilon}{\epsilon} = c_{\epsilon 2} \frac{\partial k}{k} \quad (\text{A.3})$$

On integration, we get

$$\epsilon = k^{c_{\epsilon 2}} C \quad (\text{A.4})$$

where C is the integration constant. Substituting in Eq. (A.1) and integrating along x , we get

$$k = \left[\frac{(Cx + \hat{C})}{u_1} (c_{\epsilon 2} - 1) \right]^{-\frac{1}{c_{\epsilon 2}-1}} \quad (\text{A.5})$$

where $\hat{C}$ is another integration constant.

Using the boundary condition $k = k_0$ and $\epsilon = \epsilon_0$ at $x = 0$, we get

$$\hat{C} = \frac{u_1 k_0^{c_{\epsilon 2}-1}}{c_{\epsilon 2} - 1} \quad \text{and} \quad C = \epsilon_0 k_0^{-c_{\epsilon 2}} \quad (\text{A.6})$$

and thus

$$k(x) = k_0 \left[1 + \frac{\epsilon_0 x (c_{\epsilon 2} - 1)}{k_0 u_1} \right]^{-\frac{1}{c_{\epsilon 2}-1}} \quad (\text{A.7})$$

The expression for the turbulent dissipation rate can be obtained using Eq. (A.1) as,

$$\epsilon(x) = \epsilon_0 \left[1 + \frac{\epsilon_0 x (c_{\epsilon 2} - 1)}{k_0 u_1} \right]^{-\frac{c_{\epsilon 2}}{c_{\epsilon 2}-1}} \quad (\text{A.8})$$

The above equations give the variations in k and ϵ from the inlet station to the point just upstream of the shock. Now, if x_{sh} corresponds to the shock location, then at $x = x_{sh}$, $k = k_1$ and $\epsilon = \epsilon_1$. Hence, Eqs. (A.7) and (A.8) can be written in terms of the inlet values as,

$$k_0 = k_1 \left[1 + \frac{\epsilon_0 x_{sh} (c_{\epsilon 2} - 1)}{k_0 u_1} \right]^{-\frac{1}{c_{\epsilon 2}-1}} \quad (\text{A.9})$$

and

$$\epsilon_0 = \epsilon_1 \left[1 + \frac{\epsilon_0 x_{sh} (c_{\epsilon 2} - 1)}{k_0 u_1} \right]^{-\frac{c_{\epsilon 2}}{c_{\epsilon 2}-1}} \quad (\text{A.10})$$

The same expressions hold if the shock is at $x = 0$ and x_{shk} is the distance of the shock from the inlet station. In the present work, $x_{shk} = 6.83$ for all the computations performed. Now, given the values of k_1 and ϵ_1 calculated using Eqs. (3.3) and (3.4), the inlet values of k and ϵ can be calculated from Eqs. (A.9) and (A.10) as,

$$\epsilon_0 = \epsilon_1 \left[1 + \frac{1}{\left(\frac{k_1}{\epsilon_1} - x_{sh} \frac{(c_{\epsilon 2}-1)}{u_1} \right)} x_{sh} \frac{(c_{\epsilon 2} - 1)}{u_1} \right]^{-\frac{c_{\epsilon 2}}{c_{\epsilon 2}-1}} \quad (\text{A.11})$$

and

$$k_0 = \epsilon_0 \left(\frac{k_1}{\epsilon_1} - x_{sh} \frac{(c_{\epsilon 2} - 1)}{u_1} \right) \quad (\text{A.12})$$

It is to note that, in the above expressions, all the quantities are normalized and non-dimensional, with the quantities used for normalization given in Section 3.1 of Chapter 3.

Appendix B

Rotational invariance of local SU model

Rotational invariance states that if the co-ordinate system is rotated by some angle w.r.t. to the original reference frame, then the form of equations in both the co-ordinate systems remains the same.

Generalized form of local SU model in tensor notation

$$b'_1 \frac{\partial u_i}{\partial x_i} = -c \frac{u_i}{p} \frac{\partial p}{\partial x_i} \quad (\text{B.1})$$

In 2D, it can be written as,

$$b'_1 \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) = -c \left(\frac{u}{p} \frac{\partial p}{\partial x} + \frac{v}{p} \frac{\partial p}{\partial y} \right) \quad (\text{B.2})$$

The k and ϵ equations of the new model in Cartesian tensor form can be written as,

$$\rho u_i \frac{\partial k}{\partial x_i} = -\frac{2}{3} \rho k \frac{\partial u_i}{\partial x_i} - \frac{2}{3} c \rho k \frac{u_i}{p} \frac{\partial p}{\partial x_i} - \rho \epsilon \quad (\text{B.3})$$

$$\rho u_i \frac{\partial \epsilon}{\partial x_i} = (1 - c_w) \rho \epsilon \frac{\partial u_i}{\partial x_i} + \alpha \rho \epsilon \frac{u_i}{T} \frac{\partial T}{\partial x_i} - c_2 \rho \frac{\epsilon^2}{k} \quad (\text{B.4})$$

The transport equation for TKE in 2D in the Cartesian X-Y frame can be written as,

$$\rho u \frac{\partial k}{\partial x} + \rho v \frac{\partial k}{\partial y} = -\frac{2}{3} \rho k \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) - \frac{2}{3} c \rho k \left(\frac{u}{p} \frac{\partial p}{\partial x} + \frac{v}{p} \frac{\partial p}{\partial y} \right) - \rho \epsilon \quad (\text{B.5})$$

Let us assume that the shock wave is at an angle θ to the Y-axis of this XY frame. Suppose now the reference frame is rotated anti-clockwise by θ so that the shock-wave is parallel to the new rotated Y' - axis in the new rotated $X'Y'$ co-ordinate system.

Consider a fluid particle $\mathbf{P}$ with velocity components u and v along the X and Y axes respectively, in the original XY frame.

Let its velocity in the $X'Y'$ reference frame has components u_n and u_t parallel to the X' and Y' respectively.

The transformation of the velocity and partial derivatives from the original XY frame to the rotated $X'Y'$ reference frame is given by,

$$\begin{aligned}
 u &= u_n \cos \theta - u_t \sin \theta \\
 v &= u_n \sin \theta + u_t \cos \theta \\
 \frac{\partial}{\partial x} &= \cos \theta \frac{\partial}{\partial x'} - \sin \theta \frac{\partial}{\partial y'} \\
 \frac{\partial}{\partial y} &= \sin \theta \frac{\partial}{\partial x'} + \cos \theta \frac{\partial}{\partial y'}
 \end{aligned} \tag{B.6}$$

Substituting Eqs. (B.6) in Eq. (B.5), we can find the form of the equation in the rotated reference frame.

Term by Term analysis

$$\begin{aligned}
 L.H.S. &= \rho u \frac{\partial k}{\partial x} + \rho v \frac{\partial k}{\partial y} \\
 &= \rho (u_n \cos \theta - u_t \sin \theta) \left(\cos \theta \frac{\partial k}{\partial x'} - \sin \theta \frac{\partial k}{\partial y'} \right) \\
 &\quad + \rho (u_n \sin \theta + u_t \cos \theta) \left(\sin \theta \frac{\partial k}{\partial x'} + \cos \theta \frac{\partial k}{\partial y'} \right) \\
 &= \rho u_n \cos^2 \theta \frac{\partial k}{\partial x'} - \rho u_n \cos \theta \sin \theta \frac{\partial k}{\partial y'} - \rho u_t \sin \theta \cos \theta \frac{\partial k}{\partial x'} \\
 &\quad + \rho u_t \sin^2 \theta \frac{\partial k}{\partial y'} + \rho u_n \sin^2 \theta \frac{\partial k}{\partial x'} + \rho u_n \sin \theta \cos \theta \frac{\partial k}{\partial y'} \\
 &\quad + \rho u_t \cos \theta \sin \theta \frac{\partial k}{\partial x'} + \rho u_t \cos^2 \theta \frac{\partial k}{\partial y'} \\
 &= \rho u_n \frac{\partial k}{\partial x'} + \rho u_t \frac{\partial k}{\partial y'}
 \end{aligned}$$

R.H.S. 1st term

$$= -\frac{2}{3} \rho k \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right)$$

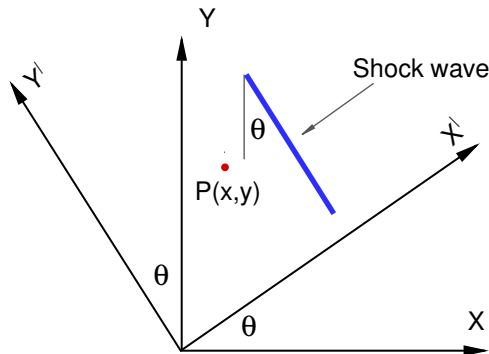


Figure B.1: Original XY and rotated $X'Y'$ co-ordinate system with shock orientation.

$$\begin{aligned}
&= -\frac{2}{3}\rho k \left[\left(\cos \theta \frac{\partial}{\partial x'} - \sin \theta \frac{\partial}{\partial y'} \right) (u_n \cos \theta - u_t \sin \theta) \right. \\
&\quad \left. + \left(\sin \theta \frac{\partial}{\partial x'} + \cos \theta \frac{\partial}{\partial y'} \right) (u_n \sin \theta + u_t \cos \theta) \right] \\
&= -\frac{2}{3}\rho k \left[\cos^2 \theta \frac{\partial u_n}{\partial x'} - \sin \theta \cos \theta \frac{\partial u_n}{\partial y'} - \sin \theta \cos \theta \frac{\partial u_t}{\partial x'} + \sin^2 \theta \frac{\partial u_t}{\partial y'} \right. \\
&\quad \left. + \sin^2 \theta \frac{\partial u_n}{\partial x'} + \sin \theta \cos \theta \frac{\partial u_n}{\partial y'} + \sin \theta \cos \theta \frac{\partial u_t}{\partial x'} + \cos^2 \theta \frac{\partial u_t}{\partial y'} \right] \\
&= -\frac{2}{3}\rho k \left(\frac{\partial u_n}{\partial x'} + \frac{\partial u_t}{\partial y'} \right)
\end{aligned}$$

R.H.S. 2nd term

$$\begin{aligned}
&= -\frac{2}{3}c \frac{\rho k}{p} \left(u \frac{\partial p}{\partial x} + v \frac{\partial p}{\partial y} \right) \\
&= -\frac{2}{3}c \frac{\rho k}{p} \left[(u_n \cos \theta - u_t \sin \theta) \left(\cos \theta \frac{\partial p}{\partial x'} - \sin \theta \frac{\partial p}{\partial y'} \right) \right. \\
&\quad \left. + (u_n \sin \theta + u_t \cos \theta) \left(\sin \theta \frac{\partial p}{\partial x'} + \cos \theta \frac{\partial p}{\partial y'} \right) \right] \\
&= -\frac{2}{3}c \frac{\rho k}{p} \left[u_n \cos^2 \theta \frac{\partial p}{\partial x'} - \rho u_n \cos \theta \sin \theta \frac{\partial p}{\partial y'} - u_t \sin \theta \cos \theta \frac{\partial p}{\partial x'} \right. \\
&\quad \left. + u_t \sin^2 \theta \frac{\partial p}{\partial y'} + u_n \sin^2 \theta \frac{\partial p}{\partial x'} + u_n \sin \theta \cos \theta \frac{\partial p}{\partial y'} \right. \\
&\quad \left. + u_t \cos \theta \sin \theta \frac{\partial p}{\partial x'} + u_t \cos^2 \theta \frac{\partial p}{\partial y'} \right] \\
&= -\frac{2}{3}c \frac{\rho k}{p} \left(u_n \frac{\partial p}{\partial x'} + u_t \frac{\partial p}{\partial y'} \right)
\end{aligned}$$

The 3rd term i.e. $-\rho\epsilon$ is a scalar and remains the same on rotation of the axis.

Appendix C

Standard SST model : k and ω transport equations

The standard SST k - ω model combines the desired features of the two most widely used two-equation turbulence models, i.e. k - ϵ and k - ω , in a k - ω modelling framework. It uses high-Re k - ϵ model in free-shear layers and standard k - ω model in the boundary layers near the wall. Following steps are followed to arrive at the model equations.

Step 1 : Transform ϵ equation to ω equation using the transformation : $\epsilon = \beta^* k \omega$ with $\beta^* = 0.09$.

Original ϵ -equation :

$$\frac{\partial \bar{\rho} \epsilon}{\partial t} + \frac{\partial (\bar{\rho} \epsilon \tilde{u}_j)}{\partial x_j} = c_{\epsilon 1} \frac{\epsilon}{k} \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] - c_{\epsilon 2} \bar{\rho} \frac{\epsilon^2}{k} \quad (\text{C.1})$$

Transformed ϵ -equation :

$$\frac{\partial \bar{\rho} \omega}{\partial t} + \frac{\partial (\bar{\rho} \tilde{u}_j \omega)}{\partial x_j} = \frac{\gamma_2}{\nu_T} \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} - \beta_2 \bar{\rho} \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \mu_T \sigma_{\omega 2}) \frac{\partial \omega}{\partial x_j} \right] + 2 \frac{\bar{\rho} \sigma_{\omega 2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (\text{C.2})$$

The two models, i.e., original k - ω model and the transformed k - ϵ model are given as,

Original k - ω model :

$$\begin{aligned} \frac{\partial \bar{\rho} k}{\partial t} + \frac{\partial (\bar{\rho} \tilde{u}_j k)}{\partial x_j} &= \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} - \beta^* \bar{\rho} k \omega + \frac{\partial}{\partial x_j} \left[(\mu + \mu_T \sigma_{k1}) \frac{\partial k}{\partial x_j} \right] \\ \frac{\partial \bar{\rho} \omega}{\partial t} + \frac{\partial (\bar{\rho} \tilde{u}_j \omega)}{\partial x_j} &= \frac{\gamma_1}{\nu_T} \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} - \beta_1 \bar{\rho} \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \mu_T \sigma_{\omega 1}) \frac{\partial \omega}{\partial x_j} \right] \end{aligned} \quad (\text{C.3})$$

Transformed k - ϵ model :

$$\begin{aligned} \frac{\partial \bar{\rho} k}{\partial t} + \frac{\partial (\bar{\rho} \tilde{u}_j k)}{\partial x_j} &= \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} - \beta^* \bar{\rho} k \omega + \frac{\partial}{\partial x_j} \left[(\mu + \mu_T \sigma_{k2}) \frac{\partial k}{\partial x_j} \right] \\ \frac{\partial \bar{\rho} \omega}{\partial t} + \frac{\partial (\bar{\rho} \tilde{u}_j \omega)}{\partial x_j} &= \frac{\gamma_2}{\nu_T} \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} - \beta_2 \bar{\rho} \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \mu_T \sigma_{\omega 2}) \frac{\partial \omega}{\partial x_j} \right] + 2 \frac{\bar{\rho} \sigma_{\omega 2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \end{aligned} \quad (\text{C.4})$$

Step 2 : The transformed k - ϵ model is then multiplied by $(1 - F_1)$ and added to the original k - ω model times F_1 , where F_1 is a blending function; $F_1 = 1$ in the sublayer and logarithmic region of the boundary layer and gradually switches to zero in the wake region. This hybrid model is called as the Baseline model (BSL). It is based on a k - ω formulation, with the standard k - ω model activated in the near wall region and the high-Re k - ϵ model activated in the outer wake region and in free shear layers. Following are the model equations :

Final form of k and ω transport equations

$$\bar{\rho} \frac{\partial k}{\partial t} + \bar{\rho} \tilde{u}_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} - \beta^* \bar{\rho} k \omega + \frac{\partial}{\partial x_j} \left[(\mu + \mu_T \sigma_k) \frac{\partial k}{\partial x_j} \right] \quad (C.5)$$

$$\bar{\rho} \frac{\partial \omega}{\partial t} + \bar{\rho} \tilde{u}_j \frac{\partial \omega}{\partial x_j} = \frac{\gamma}{\nu_T} \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} - \beta \bar{\rho} \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \mu_T \sigma_\omega) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \frac{\bar{\rho} \sigma_{\omega_2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (C.6)$$

Appendix D

Performance of linear eddy viscosity based SST k - ω model variants

The standard SST k - ω model has a number of variants with each variant aimed at removing certain deficiencies of the original model. Here, we show the performance of two of its major variants that are widely used in practice. First one is the SST-V model [171] which uses the exact same standard SST model equations except that the production term in both k and ω model equations is replaced with vorticity-based production of the form

$$P = \mu_T \Omega^2 - \frac{2}{3} \rho k \delta_{ij} \frac{\partial u_i}{\partial x_j} \quad (\text{D.1})$$

This version is sometimes favored for hypersonic flow applications, particularly when strong shocks are present to avoid numerical difficulties associated with the use of exact production term. The other variant is the SST 2003 version [171] wherein eddy viscosity definition is modified and strain-rate magnitude S is used instead of the vorticity magnitude Ω in Eq. (5.1). Also, the production of k and ω is replaced by $\min(P, 10\beta^* \rho \omega k)$, with P being the exact production term in the k and ω equations and $\beta^* = 0.09$. This form was proposed to remove the stagnation point anomaly of the standard SST model. These two variants either try to limit the k and ω production or eddy viscosity or both.

The performance of these model variants is tested using Schulein's 14° impinging shock and Holden's 36° compression ramp cases. These two cases have different length scales of interaction with Holden's case having about 3 times stronger shock strength and about 1.5 times bigger separation than the Schulein's case. As discussed earlier, the standard SST significantly underpredicts the separation for the Schulein's case while giving considerable overprediction for the Holden's case. As the two variants either limit the production of k and ω and/or eddy viscosity, they are expected to give bigger separation than the standard SST model for flows with strong interactions. This can be seen in Fig. D.1

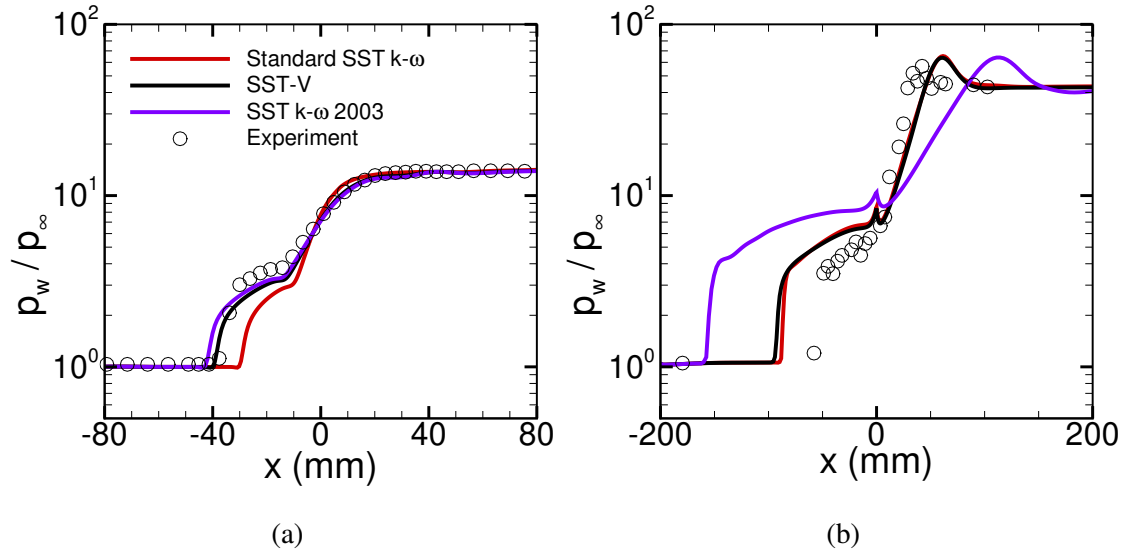


Figure D.1: Comparison of wall pressure obtained using different variants of SST $k-\omega$ model with the experimental data for (a) Schulein's 14° impinging shock and (b) Holden's 36° compression corner cases.

which plots the surface pressure distribution obtained using the different SST variants and experimental data for the Schulein's 14° and Holden's 36° SBLI cases. For the Schulein's 14° case, both variants predict significantly bigger separation than the standard SST and in fact improve the results while giving slight overpredictions. On the other hand, the predictions become worse for the Holden's 36° case, with SST-2003 version giving about 10 times bigger separation than the experiment. Thus, the different variants of SST model should be cautiously used for high-speed flow applications with shock-induced flow separation.

Appendix E

SUQ-SST model for two- and three-dimensional mean flows

E.1 Two-dimensional mean flows

Lets say $B_{ij} = S_{ik}\Omega_{kj} - \Omega_{ik}S_{kj}$ and $C_{ij} = S_{ik}S_{kj} - \frac{1}{3}S_{kl}S_{kl}\delta_{ij}$

B_{xx} :

$$B_{xx} = S_{xx}\cancel{\Omega_{xx}}^0 + S_{xy}\Omega_{yx} - \cancel{\Omega_{xx}}^0 S_{xx} - \Omega_{xy}S_{yx} \quad (\text{E.1})$$

$$B_{xx} = S_{xy}\Omega_{yx} - \Omega_{xy}S_{yx} \quad (\text{E.2})$$

$$B_{xx} = -2S_{xy}\Omega_{xy} \quad (\text{E.3})$$

B_{xy} :

$$B_{xy} = S_{xx}\Omega_{xy} + S_{xy}\cancel{\Omega_{yy}}^0 - \cancel{\Omega_{xx}}^0 S_{xy} - \Omega_{xy}S_{yy} \quad (\text{E.4})$$

$$B_{xy} = \Omega_{xy}(S_{xx} - S_{yy}) \quad (\text{E.5})$$

B_{yy} :

$$B_{yy} = S_{yx}\Omega_{xy} + S_{yy}\cancel{\Omega_{yy}}^0 - \Omega_{yx}S_{xy} - \cancel{\Omega_{yy}}^0 S_{yy} \quad (\text{E.6})$$

$$B_{yy} = 2S_{xy}\Omega_{xy} \quad (\text{E.7})$$

C_{xx} :

$$C_{xx} = S_{xx}S_{xx} + S_{xy}S_{yx} - \frac{1}{3}II_S \quad (\text{E.8})$$

$$C_{xx} = S_{xx}^2 + S_{xy}^2 - \frac{1}{3}II_S \quad (\text{E.9})$$

C_{xy} :

$$C_{xy} = S_{xx}S_{xy} + S_{xy}S_{yy} \quad (\text{E.10})$$

$$C_{xy} = S_{xy}(S_{xx} + S_{yy}) \quad (E.11)$$

C_{yy} :

$$C_{yy} = S_{yx}S_{xy} + S_{yy}S_{yy} - \frac{1}{3}II_S \quad (E.12)$$

$$C_{yy} = S_{yy}^2 + S_{xy}^2 - \frac{1}{3}II_S \quad (E.13)$$

where

$$II_S = S_{xx}^2 + S_{yy}^2 + 2S_{xy}^2 \quad (E.14)$$

Thus, the components of the extra anisotropy tensor becomes,

$$a_{xx}^{(ex)} = -2C_\mu^* \tau^2 [\beta_2 B_{xx} - \beta_3 C_{xx}] \quad (E.15)$$

$$a_{xy}^{(ex)} = -2C_\mu^* \tau^2 [\beta_2 B_{xy} - \beta_3 C_{xy}] \quad (E.16)$$

$$a_{yy}^{(ex)} = -2C_\mu^* \tau^2 [\beta_2 B_{yy} - \beta_3 C_{yy}] \quad (E.17)$$

$$a_{xx}^{(ex)} = -2C_\mu^* \tau^2 \left[\beta_2 (-2S_{xy}\Omega_{xy}) - \beta_3 \{ (S_{xx}^2 + S_{xy}^2) - \frac{1}{3}(S_{xx}^2 + S_{yy}^2 + 2S_{xy}^2) \} \right] \quad (E.18)$$

$$a_{xy}^{(ex)} = -2C_\mu^* \tau^2 \left[\beta_2 \{ \Omega_{xy}(S_{xx} - S_{yy}) \} - \beta_3 \{ S_{xy}(S_{xx} + S_{yy}) \} \right] \quad (E.19)$$

$$a_{yy}^{(ex)} = -2C_\mu^* \tau^2 \left[\beta_2 (2S_{xy}\Omega_{xy}) - \beta_3 \{ (S_{yy}^2 + S_{xy}^2) - \frac{1}{3}(S_{xx}^2 + S_{yy}^2 + 2S_{xy}^2) \} \right] \quad (E.20)$$

Reynolds stress tensor

$$-\overline{\rho u_i'' u_j''} = \tau_{ij} = 2\mu_T S'_{ij} - \frac{2}{3}\rho k \delta_{ij} - \rho k a_{ij}^{(ex)} \quad (E.21)$$

where $S'_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) = S_{ij} - \frac{1}{3} S_{kk} \delta_{ij}$ Thus,

$$\tau_{ij} = 2\mu_T S_{ij} - \frac{2}{3}\mu_T S_{kk} \delta_{ij} - \frac{2}{3}\rho k \delta_{ij} - \rho k a_{ij}^{(ex)} \quad (E.22)$$

The individual components are given by :

τ_{xx} :

$$\begin{aligned} \tau_{xx} &= 2\mu_T S_{xx} - \frac{2}{3}\mu_T S_{kk} - \frac{2}{3}\rho k - \rho k a_{xx}^{(ex)} \\ \tau_{xx} &= 2\mu_T S_{xx} - \frac{2}{3}\mu_T S_{kk} - \frac{2}{3}\rho k \\ &\quad + 2C_\mu^* \tau^2 \rho k \left[\beta_2 (-2S_{xy}\Omega_{xy}) - \beta_3 \{ (S_{xx}^2 + S_{xy}^2) - \frac{1}{3}(S_{xx}^2 + S_{yy}^2 + 2S_{xy}^2) \} \right] \end{aligned} \quad (E.23)$$

τ_{yy} :

$$\begin{aligned} \tau_{yy} &= 2\mu_T S_{yy} - \frac{2}{3}\mu_T S_{kk} - \frac{2}{3}\rho k - \rho k a_{yy}^{(ex)} \\ \tau_{yy} &= 2\mu_T S_{yy} - \frac{2}{3}\mu_T S_{kk} - \frac{2}{3}\rho k \\ &\quad + 2C_\mu^* \tau^2 \rho k \left[\beta_2 (2S_{xy}\Omega_{xy}) - \beta_3 \{ (S_{yy}^2 + S_{xy}^2) - \frac{1}{3}(S_{xx}^2 + S_{yy}^2 + 2S_{xy}^2) \} \right] \end{aligned} \quad (E.24)$$

τ_{xy} :

$$\begin{aligned}\tau_{xy} &= 2\mu_T S_{xy} - \rho k a_{xy}^{(ex)} \\ \tau_{xy} &= 2\mu_T S_{xy} \\ &\quad + 2C_\mu^* \tau^2 \rho k \left[\beta_2 \{ \Omega_{xy} (S_{xx} - S_{yy}) \} - \beta_3 \{ S_{xy} (S_{xx} + S_{yy}) \} \right]\end{aligned}\tag{E.25}$$

E.2 Three-dimensional mean flows

$$-\overline{\rho u_i'' u_i''} = \tau_{ij} = 2\mu_T S_{ij} - \frac{2}{3}\mu_T S_{kk} \delta_{ij} - \frac{2}{3}\rho k \delta_{ij} - \rho k a_{ij}^{(ex)}\tag{E.26}$$

$$\tau_{ij} = \tau_{ij}^B - \rho k a_{ij}^{(ex)}\tag{E.27}$$

where, τ_{ij}^B is the part of the Reynolds stress as given by Bousinesq hypothesis and $\rho k a_{ij}^{(ex)}$ is the extra anisotropic part of the Reynolds stress.

Different components

$$\tau_{11} = \tau_{11}^B - \rho k a_{11}^{(ex)} = \tau_{11}^B + \tau_{11}^R\tag{E.28}$$

$$\tau_{22} = \tau_{22}^B - \rho k a_{22}^{(ex)} = \tau_{22}^B + \tau_{22}^R\tag{E.29}$$

$$\tau_{33} = \tau_{33}^B - \rho k a_{33}^{(ex)} = \tau_{33}^B + \tau_{33}^R\tag{E.30}$$

$$\tau_{12} = \tau_{21} = \tau_{12}^B - \rho k a_{12}^{(ex)} = \tau_{12}^B + \tau_{12}^R\tag{E.31}$$

$$\tau_{13} = \tau_{31} = \tau_{13}^B - \rho k a_{13}^{(ex)} = \tau_{13}^B + \tau_{13}^R\tag{E.32}$$

$$\tau_{23} = \tau_{32} = \tau_{23}^B - \rho k a_{23}^{(ex)} = \tau_{23}^B + \tau_{23}^R\tag{E.33}$$

where, τ_{ij}^R is the additional stress component due to Rung's EARSMS and $\tau_{ij}^R = -\rho k a_{ij}^{(ex)}$

Invariants

$$II_S = \tau^2 S_{ij} S_{ji}\tag{E.34}$$

$$\begin{aligned}&= \tau^2 (S_{xx} S_{xx} + S_{xy} S_{yx} + S_{xz} S_{zx} + S_{yx} S_{xy} + S_{yy} S_{yy} + S_{yz} S_{zy} + S_{zx} S_{xz} + S_{zy} S_{yz} + S_{zz} S_{zz}) \\ &\tag{E.35}\end{aligned}$$

$$II_S = \tau^2 (S_{xx}^2 + S_{yy}^2 + S_{zz}^2 + 2S_{xy}^2 + 2S_{xz}^2 + 2S_{yz}^2)\tag{E.36}$$

$$II_\Omega = \tau^2 \Omega_{ij} \Omega_{ji}\tag{E.37}$$

$$\begin{aligned}&= \tau^2 (\Omega_{xx} \Omega_{xx} + \Omega_{xy} \Omega_{yx} + \Omega_{xz} \Omega_{zx} + \Omega_{yx} \Omega_{xy} + \Omega_{yy} \Omega_{yy} + \Omega_{yz} \Omega_{zy} + \Omega_{zx} \Omega_{xz} + \Omega_{zy} \Omega_{yz} + \Omega_{zz} \Omega_{zz}) \\ &\tag{E.38}\end{aligned}$$

$$\begin{aligned}&= \tau^2 (\Omega_{xx} \Omega_{xx} - \Omega_{xy} \Omega_{xy} - \Omega_{xz} \Omega_{xz} - \Omega_{yx} \Omega_{yx} + \Omega_{yy} \Omega_{yy} - \Omega_{yz} \Omega_{yz} - \Omega_{zx} \Omega_{zx} - \Omega_{zy} \Omega_{zy} + \Omega_{zz} \Omega_{zz}) \\ &\tag{E.39}\end{aligned}$$

$$II_S = \tau^2 \left(-2\Omega_{xy}^2 - 2\Omega_{xz}^2 - 2\Omega_{yz}^2 \right) \quad (E.40)$$

Strain-rate and Vorticity tensor components

$$S'_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} = S_{ij} - \frac{1}{3} S_{kk} \delta_{ij} \quad (E.41)$$

Strain-rate tensor components

$$S_{xx} = \frac{\partial u}{\partial x} - \frac{1}{3} \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) \quad (E.42)$$

$$S_{xy} = S_{yx} = \frac{1}{2} \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \quad (E.43)$$

$$S_{xz} = S_{zx} = \frac{1}{2} \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) \quad (E.44)$$

$$S_{yy} = \frac{\partial v}{\partial y} - \frac{1}{3} \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) \quad (E.45)$$

$$S_{yz} = S_{zy} = \frac{1}{2} \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) \quad (E.46)$$

$$S_{zz} = \frac{\partial w}{\partial z} - \frac{1}{3} \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) \quad (E.47)$$

Vorticity tensor components

$$\Omega_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \quad (E.48)$$

$$\Omega_{xx} = 0 \quad (E.49)$$

$$\Omega_{xy} = -\Omega_{yx} = \frac{1}{2} \left(\frac{\partial u}{\partial y} - \frac{\partial v}{\partial x} \right) \quad (E.50)$$

$$\Omega_{xz} = -\Omega_{zx} = \frac{1}{2} \left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} \right) \quad (E.51)$$

$$\Omega_{yy} = 0 \quad (E.52)$$

$$\Omega_{yz} = -\Omega_{zy} = \frac{1}{2} \left(\frac{\partial v}{\partial z} - \frac{\partial w}{\partial y} \right) \quad (E.53)$$

$$\Omega_{zz} = 0 \quad (E.54)$$

Extra anisotropy $a_{ij}^{(ex)}$

Lets say $B_{ij} = S_{ik}\Omega_{kj} - \Omega_{ik}S_{kj}$ and $C_{ij} = S_{ik}S_{kj} - \frac{1}{3}S_{kl}S_{kl}\delta_{ij}$

B_{xx} :

$$B_{xx} = S_{xx}\Omega_{xx} + S_{xy}\Omega_{yx} + S_{xz}\Omega_{zx} - \Omega_{xx}S_{xx} - \Omega_{xy}S_{yx} - \Omega_{xz}S_{zx} \quad (E.55)$$

$$B_{xx} = -S_{xy}\Omega_{xy} - S_{xz}\Omega_{xz} - \Omega_{xy}S_{xy} - \Omega_{xz}S_{xz} \quad (E.56)$$

$$B_{xx} = -2S_{xy}\Omega_{xy} - 2S_{xz}\Omega_{xz} \quad (\text{E.57})$$

B_{xy} :

$$B_{xy} = S_{xx}\Omega_{xy} + S_{xy}\Omega_{yy} + S_{xz}\Omega_{zy} - \Omega_{xx}S_{xy} - \Omega_{xy}S_{yy} - \Omega_{xz}S_{zy} \quad (\text{E.58})$$

$$B_{xy} = S_{xx}\Omega_{xy} - S_{xz}\Omega_{yz} - \Omega_{xy}S_{yy} - \Omega_{xz}S_{yz} \quad (\text{E.59})$$

B_{xz} :

$$B_{xz} = S_{xx}\Omega_{xz} + S_{xy}\Omega_{yz} + S_{xz}\Omega_{zz} - \Omega_{xx}S_{xz} - \Omega_{xy}S_{yz} - \Omega_{xz}S_{zz} \quad (\text{E.60})$$

$$B_{xz} = S_{xx}\Omega_{xz} + S_{xy}\Omega_{yz} - \Omega_{xy}S_{yz} - \Omega_{xz}S_{zz} \quad (\text{E.61})$$

B_{yy} :

$$B_{yy} = S_{yx}\Omega_{xy} + S_{yy}\Omega_{yy} + S_{yz}\Omega_{zy} - \Omega_{yx}S_{xy} - \Omega_{yy}S_{yy} - \Omega_{yz}S_{zy} \quad (\text{E.62})$$

$$B_{yy} = S_{xy}\Omega_{xy} - S_{yz}\Omega_{yz} + \Omega_{xy}S_{xy} - \Omega_{yz}S_{yz} \quad (\text{E.63})$$

$$B_{yy} = 2S_{xy}\Omega_{xy} - 2S_{yz}\Omega_{yz} \quad (\text{E.64})$$

B_{yz} :

$$B_{yz} = S_{yx}\Omega_{xz} + S_{yy}\Omega_{yz} + S_{yz}\Omega_{zz} - \Omega_{yx}S_{xz} - \Omega_{yy}S_{yz} - \Omega_{yz}S_{zz} \quad (\text{E.65})$$

$$B_{yz} = S_{xy}\Omega_{xz} + S_{yy}\Omega_{yz} + \Omega_{xy}S_{xz} - \Omega_{yz}S_{zz} \quad (\text{E.66})$$

B_{zz} :

$$B_{zz} = S_{zx}\Omega_{xz} + S_{zy}\Omega_{yz} + S_{zz}\Omega_{zz} - \Omega_{zx}S_{xz} - \Omega_{zy}S_{yz} - \Omega_{zz}S_{zz} \quad (\text{E.67})$$

$$B_{zz} = S_{xz}\Omega_{xz} + S_{yz}\Omega_{yz} + \Omega_{xz}S_{xz} + \Omega_{yz}S_{yz} \quad (\text{E.68})$$

$$B_{zz} = 2S_{xz}\Omega_{xz} + 2S_{yz}\Omega_{yz} \quad (\text{E.69})$$

C_{xx} :

$$C_{xx} = S_{xx}S_{xx} + S_{xy}S_{yx} + S_{xz}S_{zx} - \frac{1}{3}\widetilde{II}_S \quad (\text{E.70})$$

$$C_{xx} = S_{xx}^2 + S_{xy}^2 + S_{xz}^2 - \frac{1}{3}\widetilde{II}_S \quad (\text{E.71})$$

C_{xy} :

$$C_{xy} = S_{xx}S_{xy} + S_{xy}S_{yy} + S_{xz}S_{zy} \quad (\text{E.72})$$

$$C_{xy} = S_{xx}S_{xy} + S_{xy}S_{yy} + S_{xz}S_{yz} \quad (\text{E.73})$$

C_{xz} :

$$C_{xz} = S_{xx}S_{xz} + S_{xy}S_{yz} + S_{xz}S_{zz} \quad (\text{E.74})$$

C_{yy} :

$$C_{yy} = S_{yx}S_{xy} + S_{yy}S_{yy} + S_{yz}S_{zy} - \frac{1}{3}\widetilde{\Pi}_S \quad (\text{E.75})$$

$$C_{yy} = S_{xy}^2 + S_{yy}^2 + S_{yz}^2 - \frac{1}{3}\widetilde{\Pi}_S \quad (\text{E.76})$$

C_{yz} :

$$C_{yz} = S_{yx}S_{xz} + S_{yy}S_{yz} + S_{yz}S_{zz} \quad (\text{E.77})$$

$$C_{yz} = S_{xy}S_{xz} + S_{yy}S_{yz} + S_{yz}S_{zz} \quad (\text{E.78})$$

C_{zz} :

$$C_{zz} = S_{zx}S_{xz} + S_{zy}S_{yz} + S_{zz}S_{zz} - \frac{1}{3}\widetilde{\Pi}_S \quad (\text{E.79})$$

$$C_{zz} = S_{xz}^2 + S_{yz}^2 + S_{zz}^2 - \frac{1}{3}\widetilde{\Pi}_S \quad (\text{E.80})$$

where

$$\widetilde{\Pi}_S = S_{kl}S_{kl} \quad (\text{E.81})$$

$$= S_{xx}S_{xx} + S_{xy}S_{xy} + S_{xz}S_{xz} + S_{yx}S_{yx} + S_{yy}S_{yy} + S_{yz}S_{yz} + S_{zx}S_{zx} + S_{zy}S_{zy} + S_{zz}S_{zz} \quad (\text{E.82})$$

$$\widetilde{\Pi}_S = S_{xx}^2 + S_{yy}^2 + S_{zz}^2 + 2S_{xy}^2 + 2S_{xz}^2 + 2S_{yz}^2 \quad (\text{E.83})$$

Thus, the components of the extra anisotropy tensor becomes,

$$a_{xx}^{(ex)} = -2C_\mu^* \tau^2 [\beta_2 B_{xx} - \beta_3 C_{xx}] \quad (\text{E.84})$$

$$a_{xy}^{(ex)} = -2C_\mu^* \tau^2 [\beta_2 B_{xy} - \beta_3 C_{xy}] \quad (\text{E.85})$$

$$a_{xz}^{(ex)} = -2C_\mu^* \tau^2 [\beta_2 B_{xz} - \beta_3 C_{xz}] \quad (\text{E.86})$$

$$a_{yy}^{(ex)} = -2C_\mu^* \tau^2 [\beta_2 B_{yy} - \beta_3 C_{yy}] \quad (\text{E.87})$$

$$a_{yz}^{(ex)} = -2C_\mu^* \tau^2 [\beta_2 B_{yz} - \beta_3 C_{yz}] \quad (\text{E.88})$$

$$a_{zz}^{(ex)} = -2C_\mu^* \tau^2 [\beta_2 B_{zz} - \beta_3 C_{zz}] \quad (\text{E.89})$$

$$a_{xx}^{(ex)} = -2C_\mu^* \tau^2 \left[\beta_2 (-2S_{xy}\Omega_{xy} - 2S_{xz}\Omega_{xz}) - \beta_3 \{S_{xx}^2 + S_{xy}^2 + S_{xz}^2 - \frac{1}{3}\widetilde{\Pi}_S\} \right] \quad (\text{E.90})$$

$$a_{xy}^{(ex)} = -2C_\mu^* \tau^2 \left[\beta_2 \{S_{xx}\Omega_{xy} - S_{xz}\Omega_{yz} - \Omega_{xy}S_{yy} - \Omega_{xz}S_{yz}\} - \beta_3 \{S_{xx}S_{xy} + S_{xy}S_{yy} + S_{xz}S_{yz}\} \right] \quad (\text{E.91})$$

$$a_{xz}^{(ex)} = -2C_\mu^* \tau^2 \left[\beta_2 \{S_{xx}\Omega_{xz} + S_{xy}\Omega_{yz} - \Omega_{xy}S_{yz} - \Omega_{xz}S_{zz}\} - \beta_3 \{S_{xx}S_{xz} + S_{xy}S_{yz} + S_{xz}S_{zz}\} \right] \quad (\text{E.92})$$

$$a_{yy}^{(ex)} = -2C_\mu^* \tau^2 \left[\beta_2 (2S_{xy} \Omega_{xy} - 2S_{yz} \Omega_{yz}) - \beta_3 \{S_{xy}^2 + S_{yy}^2 + S_{yz}^2 - \frac{1}{3} \widetilde{II}_S\} \right] \quad (\text{E.93})$$

$$a_{yz}^{(ex)} = -2C_\mu^* \tau^2 \left[\beta_2 (S_{xy} \Omega_{xz} + S_{yy} \Omega_{yz} + \Omega_{xy} S_{xz} - \Omega_{yz} S_{zz}) - \beta_3 \{S_{xy} S_{xz} + S_{yy} S_{yz} + S_{yz} S_{zz}\} \right] \quad (\text{E.94})$$

$$a_{zz}^{(ex)} = -2C_\mu^* \tau^2 \left[\beta_2 (2S_{xz} \Omega_{xz} + 2S_{yz} \Omega_{yz}) - \beta_3 \{S_{xz}^2 + S_{yz}^2 + S_{zz}^2 - \frac{1}{3} \widetilde{II}_S\} \right] \quad (\text{E.95})$$

Appendix F

Grid convergence studies

F.1 Canonical STI case

The non-conservative and the conservative forms of the new k - ϵ model are subjected to a grid refinement study. A test case with upstream Mach number of 2.5 is chosen and the results are computed using three grids : baseline grid of 1600 points, a finer grid of 3200 points and a coarser grid with 800 points. Grid points are uniformly distributed between the inlet and exit locations, such that the shock normal grid spacing for the three grids are 4.65×10^{-2} , 2.325×10^{-2} and 1.1625×10^{-2} respectively. Note that the x -coordinate is normalized by the most energetic wave number in the upstream turbulence field.

Fig. F.1 shows the variation of k and ϵ along the streamwise direction, where the turbulence quantities are normalized by their values immediately upstream of the shock

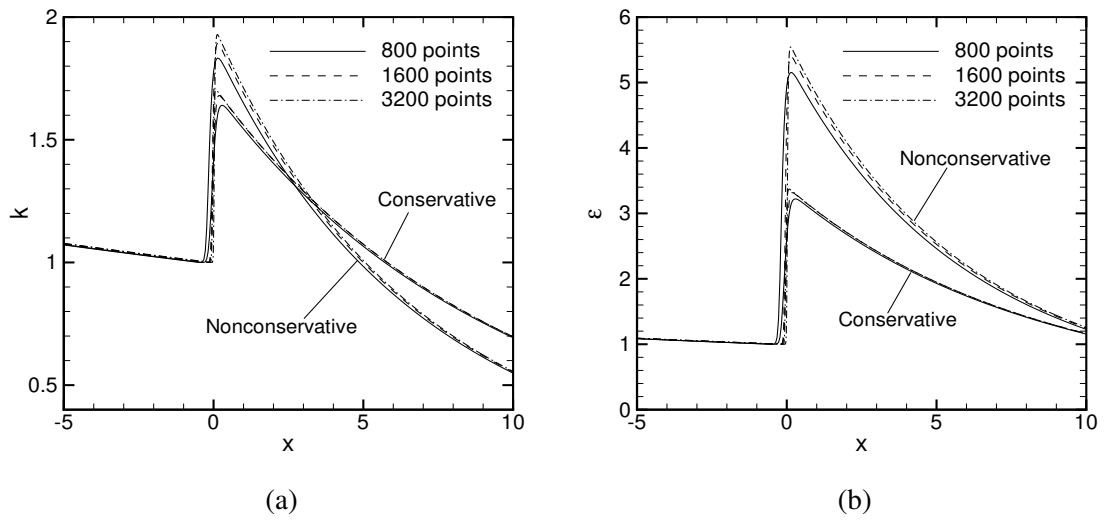


Figure F.1: Effect of grid refinement on (a) TKE and (b) turbulent dissipation rate evolution for upstream Mach number of 2.5.

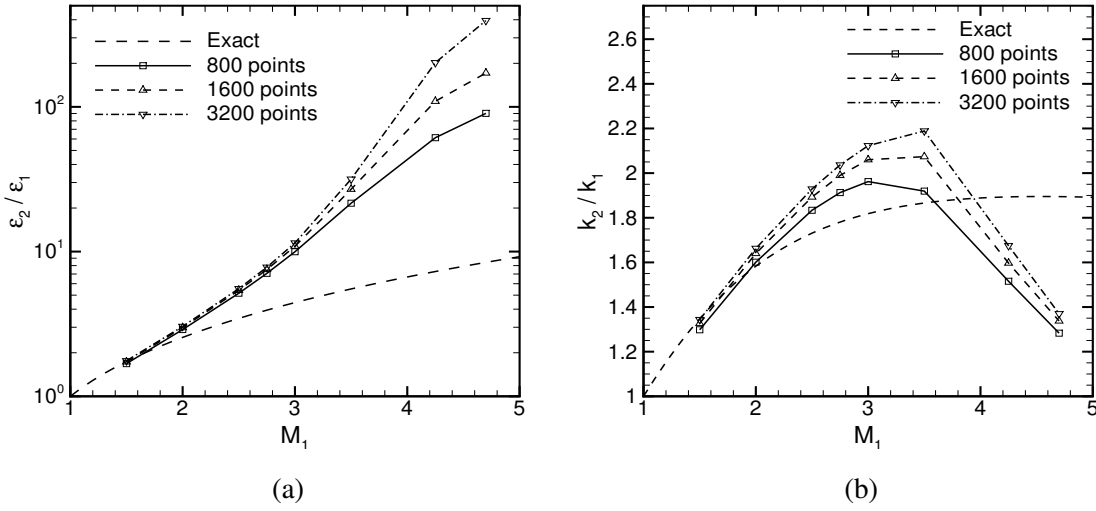


Figure F.2: Amplification in (a) ϵ and (b) k computed using the non-conservative k - ϵ equations on successively refined grids for a range of upstream Mach number.

wave. It is found that the variation in grid-point density has noticeable effect on the k and ϵ solutions in the shock region. The thickness of the shock decreases on successive grid refinement, and we see an increase in the k and ϵ amplification. The pre and post-shock decays are relatively insensitive to grid refinement for both the conservative and non-conservative formulations of the proposed k - ϵ model. Overall, the baseline grid of 1600 points is found to be adequate for the Mach 2.5 case considered here, and is used in the majority of comparisons presented in this section. The non-conservative k - ϵ model solution is more sensitive to grid-point density at higher Mach numbers, as discussed next.

The amplification in turbulent dissipation rate as computed using the non-conservative Eq. (4.15) is plotted as a function of upstream Mach number in Fig. F.2a. The exact solution (4.16) obtained by analytical integration of the equation is also shown for comparison. The numerical solution is much higher than the exact integration for Mach numbers beyond 2; note the log scale used for the vertical axis. Further, the amplification increases with grid refinement and does not show a converging trend. This is due to the increase in the truncation error in the ϵ -equation with grid refinement. The leading velocity and temperature derivatives in the error term have $1/\delta^3$ variation, as discussed earlier. This scales as $1/\Delta x^3$, since δ is proportional to the grid size Δx . Thus, for a fixed Mach number, the amplification in turbulent dissipation rate increases with grid refinement.

Similarly, owing to the large discretization errors due to the non-conservative production terms in the k -equation (4.8), the numerical results over-predict the exact inviscid integration (4.10). For a fixed Mach number, the TKE amplification also increases on

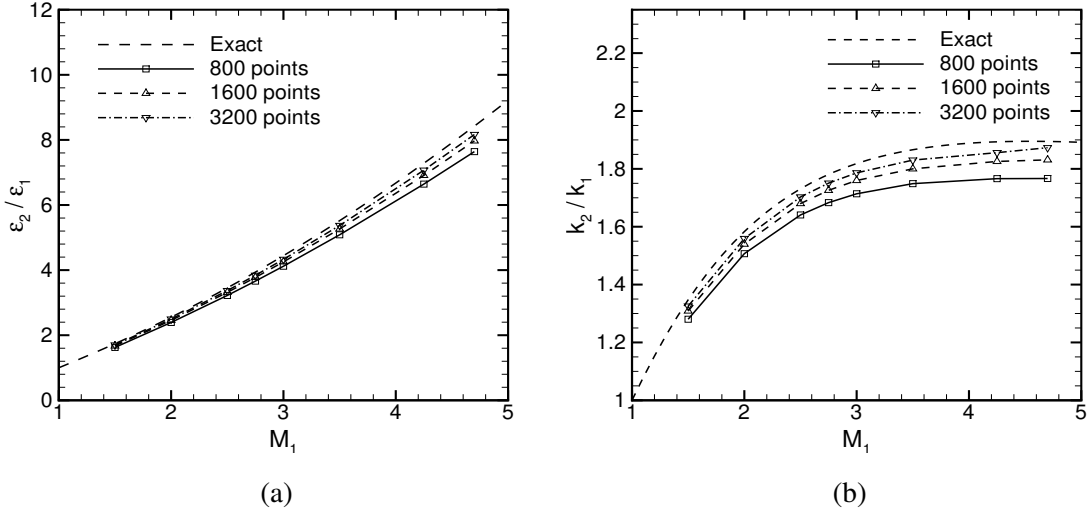


Figure F.3: The conservative k - ϵ equations show a systematic grid convergence in (a) ϵ and (b) k amplifications for all Mach numbers.

successive grid refinement for Mach numbers up to about 3. At higher Mach numbers, the numerical prediction is lower than the exact solution. Further, the TKE amplification decreases with increasing shock strength, which is opposite to what is observed for the turbulent dissipation rate and is physically unrealistic. This behaviour is similar to that observed in [75] and is due to very high amplification of the turbulent dissipation rate for high Mach numbers.

On the other hand, the TKE and dissipation rate amplifications obtained using the conservative equations (Fig. F.3) show a converging behaviour as the grid is successively refined. The solutions are also consistent with the exact integration results. This improvement in the results over the non-conservative case is due to the reason that the conservative formulation removes the non-conservative source terms, which causes large discretisation errors. The truncation error in the convective terms scales as $(\Delta x)^2$, and hence decreases as the grid spacing is reduced. The 1600 and 3200 grids yield close results, and it can be seen that, in the limit of $\Delta x \rightarrow 0$, the numerical results will approach the exact inviscid integration results.

F.2 Two-dimensional SBLI cases

Extensive grid convergence studies have been carried out for the SBLI test cases using SUQ-SST model by systematically varying the number of grid points in both x and y directions as well as refining the cell size in critical regions. The methodology adopted in performing the grid refinement studies is explained below for the UFAST Mach 2.3

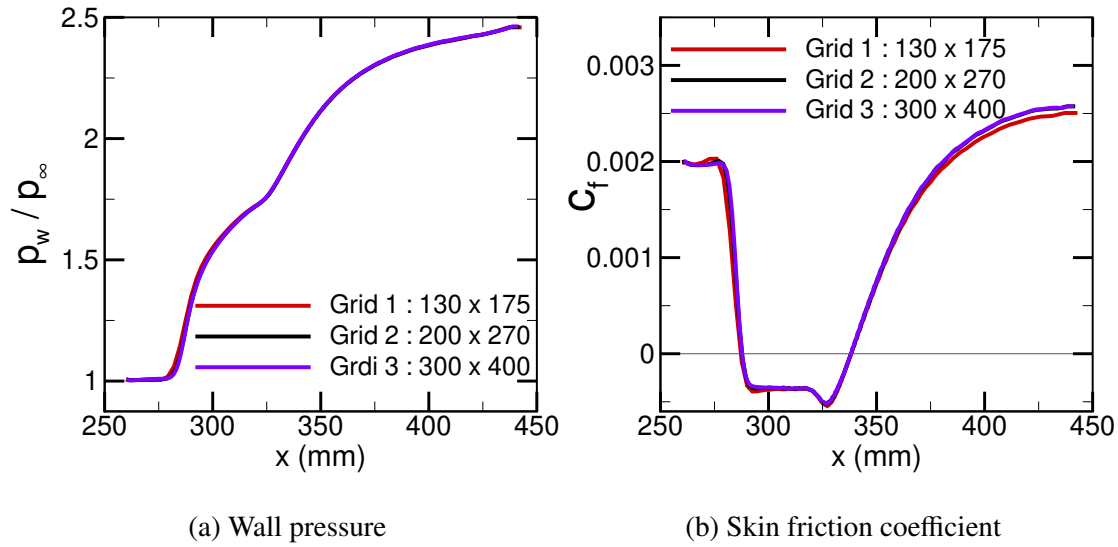


Figure F.4: Results obtained on three different grids using SUQ-SST model for the UFAST SBLI case.

impinging shock case. Grid convergence studies for the SBLI cases of Schulein and Holden follow a similar methodology and are not presented here for the sake of brevity.

For the UFAST case, three successively refined grids with 130×175 , 200×270 , and 300×400 finite volume cells are used for the grid convergence study. The grids are successively refined by a factor of about 1.5 in each direction. The near wall spacing and the cell size at the interaction location are both reduced by a factor of 1.5 for each successively refined grid. Figure F.4 shows the skin friction and wall pressure distribution for the three grids. Skin friction is found to be more sensitive to grid refinement than the wall pressure and so it is used as an indicator to arrive at a grid converged solution. While surface pressure is almost identical for the three grids, skin friction shows noticeable difference between the 130×175 grid and the other two finer grids. The two finer grid solutions almost overlap for the skin friction with a maximum variation of less than 0.1% between them. So, 200×270 grid is considered for the computations.

F.3 Back-facing step case

A careful grid convergence study is done using the standard SST model for the BFS flow configuration. The different grids used for the study are shown in Table fig:GC-BFS. While refining the grids, block-wise cells in the i,j, and k directions are increased by a factor of about 1.5 between the different grids. However, the overall grid sizes increase by more than that factor. In fact, there's an increase by a factor of about 3.8 between Grid 1 and Grid 2 and a factor of about 3.7 between Grid 2 and Grid 3. A comparison of cell

Table F.1: Comparison of grids used for computing the supersonic flow over the BFS geometry.

Block	Grid 1 (coarse)	Grid 2 (baseline)	Grid 3 (fine)
Block 1	$50 \times 3 \times 115$	$75 \times 5 \times 175$	$115 \times 8 \times 270$
Block 2	$70 \times 3 \times 115$	$105 \times 5 \times 175$	$160 \times 8 \times 270$
Block 3	$80 \times 3 \times 70$	$120 \times 5 \times 105$	$180 \times 8 \times 160$
No. of cells in separation region	65	100	150
Total Grid size ($\times 10^3$)	58.2	220.5	824.4

Table F.2: Comparison of grid sizes in meters in the important regions in the structured grid used for BFS simulations.

Grid	dyw (BL) in m	dxw (corner) in m	dx (reattachment) in m
Grid 1 (coarse)	1.5×10^{-6}	1.5×10^{-5}	5×10^{-4}
Grid 2 (baseline)	1×10^{-6}	1×10^{-5}	3×10^{-4}
Grid 3 (fine)	6.7×10^{-7}	6.7×10^{-6}	2×10^{-4}

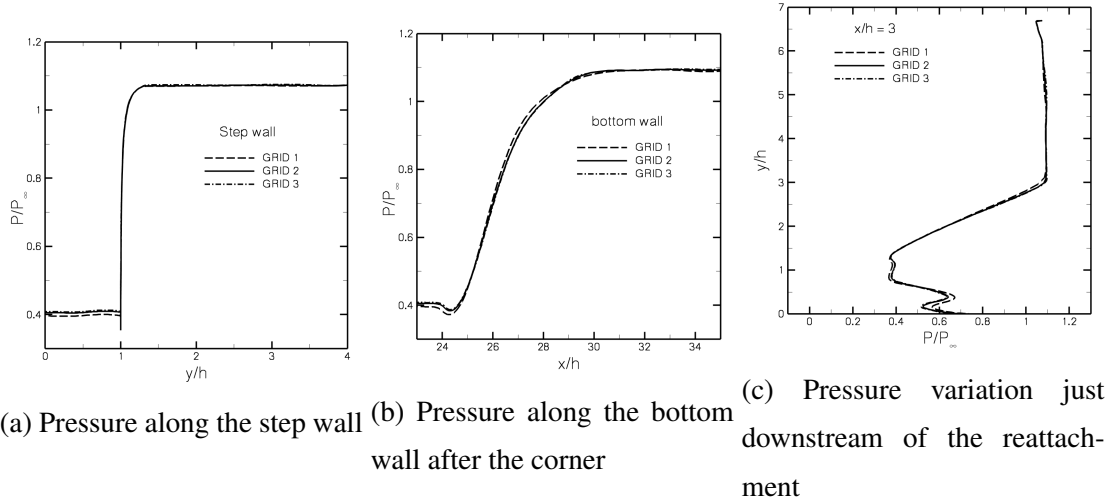


Figure F.5: Grid convergence results for back-facing step case

sizes at the important locations, namely, near the boundary layer walls, near the corner, and near the free-shear layer reattachment region is shown in Table F.2. Figure F.5 show the grid convergence study results in terms of pressure variations at different important locations for the 3 grids. The computations show that Grid 2 and Grid 3 results are almost identical and so Grid 2 is considered to give grid-converged results. Hence, Grid 2 is used for all the computations.

Table F.3: Comparison of grids for the supersonic base flow case.

Block	Grid 1 (coarse)	Grid 2 (baseline)	Grid 3 (fine)
Block 1	$10 \times 10 \times 130$	$15 \times 15 \times 200$	$25 \times 25 \times 300$
Block 2	$35 \times 40 \times 130$	$55 \times 60 \times 200$	$85 \times 100 \times 300$
Block 3	$35 \times 40 \times 130$	$55 \times 60 \times 200$	$85 \times 100 \times 300$
Block 4	$55 \times 40 \times 35$	$85 \times 60 \times 55$	$130 \times 100 \times 85$
Sep. region cells	100	150	225
Grid size ($\times 10^6$)	0.454	1.6455	6.3925

Table F.4: Comparison of cell sizes in meters in the crucial regions in the structured grid used for base flow computations.

Grid	dyw (BL)	dxw (corner)	dx (reattachment)
Grid 1 (coarse)	1.5×10^{-6}	1.5×10^{-5}	5×10^{-4}
Grid 2 (baseline)	1×10^{-6}	1×10^{-5}	3×10^{-4}
Grid 3 (fine)	6.7×10^{-7}	6.7×10^{-6}	2×10^{-4}

F.4 Axisymmetric base flow case

A strategy similar to that followed for the BFS case is adopted to perform grid convergence studies for the supersonic base flow case. The standard SST model is used and a careful grid convergence study is performed. Table F.3 shows the grid sizes in the different blocks used during the grid convergence studies. While refining the grids, block-wise cells in the i, j , and k directions are increased by a factor of about 1.5 between the different grids. However, the overall grid sizes increase by more than that factor. In fact, there's an increase by a factor of about 3.6 between Grid 1 and Grid 2 and a factor of about 3.9 between Grid 2 and Grid 3. A comparison of cell sizes at the important locations, namely, near the boundary layer walls, near the corner, and near the free-shear layer reattachment region is shown in Table F.4. Results for the base pressure and model centerline velocity (see Fig. F.6 obtained using the different grids show that Grid 2 can be considered to give grid-converged results.

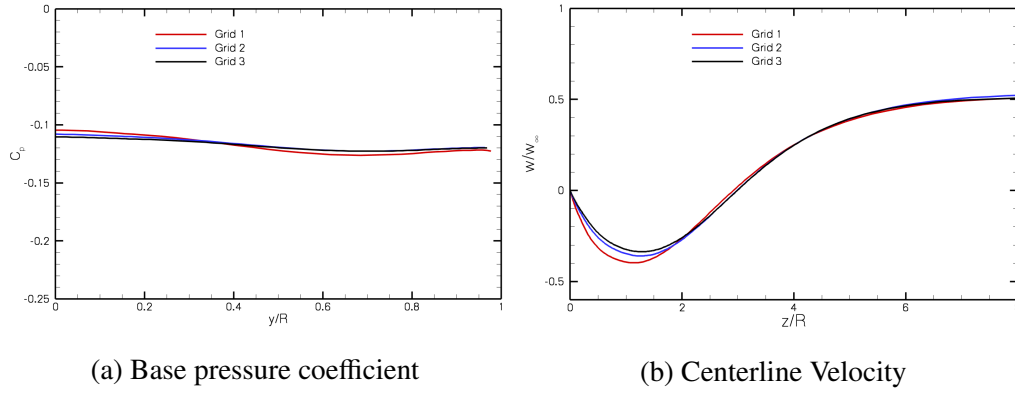


Figure F.6: Results obtained using the different grids for the axisymmetric base flow case.

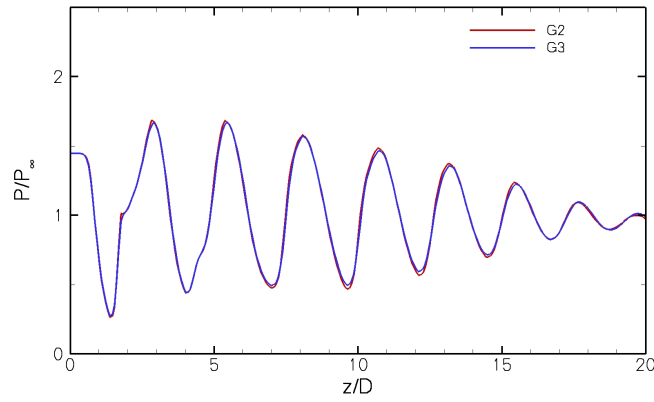


Figure F.7: Centerline pressure variation for two finer grids used in the supersonic under-expanded jet computations.

F.5 Supersonic underexpanded jet case

Grid convergence studies performed for the supersonic underexpanded jet also follow a similar strategy of grid refinement as that followed for the BFS and axisymmetric base flow cases. Grid sizes in the i , j , and k directions of each of the blocks are refined by a factor of about 1.5. Two finer grids G1 and G2 used for the computations are shown below. The two grids give almost identical results for the pressure variations along the jet centerline in the axial direction, as shown in Fig. F.7. Similar results are obtained for other mean flow quantities. For all the computations performed for this case, the grid G3 grid has been used. Note that increasing the grid size by a factor of 1.5 beyond the size of 267 in the axial direction, i.e. the k -direction, resulted in solution convergence issues. So, the finer grids utilized identical number of cells equal to 267 in the axial direction and the grid was then refined in the radial direction.

- G2: Block 1) $10 \times 10 \times 267$ & Block 2) $120 \times 30 \times 267$
- G3: Block 1) $15 \times 15 \times 267$ & Block 2) $195 \times 45 \times 267$

References

- [1] Hutton, A.G., and Ashworth, R.M., 2005, “The challenge of turbulence modelling in modern aeronautical design,” *Intl. J. for Numerical Methods in Fluids* **47**(8-9):721-37.
- [2] Liou, W.W., Huang, G., and Shih, T.H., 2000, “Turbulence model assessment for shock wave/turbulent boundary-layer interaction in transonic and supersonic flows,” *Comput Fluids* 29(3):275-299.
- [3] Balabel, A., Hegab, A.M., Nasr, M., and El-Behery, S.M., 2011, “Assessment of turbulence modeling for gas flow in two-dimensional convergent-divergent rocket nozzle,” *Applied Mathematical Modelling* **35**(7):3408-22.
- [4] Babinsky, H., and Harvey, J.K., 2011, *Shock wave-boundary-layer interactions*, Vol. 32 (Cambridge University Press).
- [5] Zheltovodov, A.A., 2006, “Some advances in research of shock wave turbulent boundary layer interactions,” *AIAA Paper* **496**:1-25.
- [6] Dolling, D, 2001, “Fifty years of shock wave/boundary layer interaction research: what next?” *AIAA J* **39**(8):1517-1531.
- [7] Mehta, M., Dufrene, A.T., Seaford, M., and Knox, K., 2016, “Space Launch System Base Heating Test: Environments and Base Flow Physics,” in 54th *AIAA Aerospace Sciences Meeting*, pp.0547.
- [8] Mehta, M., Canabal, F., Tashakkor, S.B., and Smith, S.D., 2013, “Numerical Base Heating Sensitivity Study for a Four-Rocket Engine Core Configuration,” *Journal of Spacecraft and Rockets* **50**(3):509-526.
- [9] Dimonte, G., and Tipton, R., 2006, “K-L turbulence model for the self-similar growth of the Rayleigh-Taylor and Richtmyer-Meshkov instabilities,” *Physics of Fluids* **18**(8):085101.

- [10] Morán-López, J.T., and Schilling, O., 2013, “Multicomponent Reynolds-averaged Navier-Stokes simulations of reshocked Richtmyer-Meshkov instability-induced mixing,” *High Energy Density Physics* **9**(1):112-21.
- [11] Morán-López, J.T., and Schilling, O., 2014, “Multi-component Reynolds-averaged Navier-Stokes simulations of Richtmyer-Meshkov instability and mixing induced by reshock at different times,” *Shock Waves* **24**(3):325-43.
- [12] Hillebrandt, W., and Niemeyer, J.C., 2000, “Type Ia supernova explosion models,” *Annual Review of Astronomy and Astrophysics* **38**(1):191-230.
- [13] Zhou, Y., 2017, “Rayleigh-Taylor and Richtmyer-Meshkov instability induced flow, turbulence, and mixing. I,” *Physics Reports* **720**:1-136.
- [14] Zhou, Y., 2017, “Rayleigh-Taylor and Richtmyer-Meshkov instability induced flow, turbulence, and mixing. II,” *Physics Reports* **723**:1-160.
- [15] Gusman, M., Housman, J. and Kiris, C., 2011, “Best Practices for CFD simulations of launch vehicle ascent with plumes-overflow perspective,” in *49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, pp.1054.
- [16] Vieira, R., and Azevedo, J.L., 2013, “RANS simulations of flows with shock wave-boundary layer interaction”, in *51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, 2013-0985.
- [17] Brown, J.L., 2013, “Hypersonic shock wave impingement on turbulent boundary layers: Computational analysis and uncertainty,” *Journal of Spacecraft and Rockets* **50**(1):96-123.
- [18] Manokaran, K., Ramakrishna, M., and Jayachandran, T., 2020, “Application of Flux Vector Splitting Methods with SST Turbulence Model to Wall-Bounded Flows,” *Computers & Fluids* :104611.
- [19] Yoder, D.A., and Orkwis, P.D., 2020, “On the Use of Optimization Techniques for Turbulence Model Calibration,” *Computers & Fluids* :104752.
- [20] Coakley, T., and Huang, P., 1992, “Turbulence Modeling for High Speed Flows,” in *30th Aerospace Sciences Meeting and Exhibit*, AIAA paper, 1992-0436.
- [21] Roy, C.J., and Blottner, F.G., 2006, “Review and assessment of turbulence models for hypersonic flows,” *Progress in Aerospace Sciences* **42**(7):469-530.

- [22] Ma, L., Lu, L., Fang, J., and Wang, Q., 2014, "A study on turbulence transportation and modification of Spalart-Allmaras model for shock-wave/turbulent boundary layer interaction flow," *Chinese Journal of Aeronautics* **27**(2):200-209.
- [23] Morgan, B., Duraisamy, K., Nguyen, N., Kawai, S., and Lele, S.K., 2013, "Flow physics and RANS modelling of oblique shock/turbulent boundary layer interaction," *Journal of Fluid Mechanics* **729**:231-284.
- [24] Salin, A., Yao, Y., and Zheltovodov, A.A., 2013, "Comparison and improvement of wall heat transfer prediction in crossing-shock-wave/turbulent-boundary-layer interaction conditions," in *51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, 666.
- [25] Roidl, B., Meinke, M., and Schroder, W., 2012, "A zonal RANS-LES method for compressible flows," *Comput Fluids* **67**:1-15.
- [26] Grilli, M., Adams, N.A., Willems, S., and Guelhan, A., 2012, "Experimental and numerical investigation on shockwave/turbulent boundary layer interaction," in *42nd AIAA fluid dynamics conference and exhibit, American Institute of Aeronautics and Astronautics*, New Orleans, Louisiana.
- [27] Pasha, A.A., and Sinha, K., 2012, "Simulation of hypersonic shock/turbulent boundary-layer interactions using shock-unsteadiness model," *J Prop and Power* **28**(1):46-60.
- [28] Shih, T.H., Liou, W.W., Shabbir, A., Yang, Z., and Zhu, J., 1995, "A new $k-\epsilon$ eddy viscosity model for high Reynolds number turbulent flows," *Comput Fluids* **24**(3):227-238.
- [29] Gaitonde, D.V., 2015, "Progress in shock wave/boundary layer interactions," *Prog Aerosp Sci* **72**:80-99.
- [30] Sinha, K., Mahesh, K., and Candler, G.V., 2003, "Modeling shock unsteadiness in shock/turbulence interaction," *Phys Fluids* **15**(8):2290-2297.
- [31] Zhang, Z., Gao, Z., Jiang, C., and Lee, C., 2015, "Turbulence model modification for fake amplification of Turbulence Kinetic Energy by Shock-Turbulence Interaction," in *51st AIAA/SAE/ASEE Joint Propulsion Conference*, pp.4205.
- [32] Durbin, P.A., 1996, "On the $k-3$ stagnation point anomaly," *Int J heat fluid flow* **17**(1):89-90.

- [33] Lee, S., Lele, S.K., and Moin, P., 1997, "Interaction of isotropic turbulence with shock waves: effect of shock strength," *J Fluid Mech* **340**:225-247.
- [34] Sinha, K., Mahesh, K., Candler, G.V., 2005, "Modeling the effect of shock unsteadiness in shock/turbulent boundary-layer interactions," *AIAA J.* **43**(3):586-594.
- [35] Pasha, A.A., and Sinha, K., 2008, "Shock-unsteadiness model applied to oblique shock wave/turbulent boundary-layer interaction" *Intl. J. Comput Fluid Dynamics* **22**(8):569-582.
- [36] Emami, B., Bussmann, M., and Tran, H., 2010, "Application of realizability and shock unsteadiness to k - ϵ simulations of under-expanded axisymmetric supersonic free jets," *Journal of fluids engineering* **132**(4).
- [37] Menter, F.R., 1994, "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA J.* **32**(8):1598-1605.
- [38] Menter, F.R., 1992, "Performance of popular turbulence model for attached and separated adverse pressure gradient flows." *AIAA J.* **30**(8):2066-2072.
- [39] Menter, F.R., 1996, "A comparison of some recent eddy-viscosity turbulence models. *Journal of Fluids Engineering*" **118**(3).
- [40] Steelant, J., 2002, "Effect of a compressibility correction on different turbulence models" in *Engineering Turbulence Modelling and Experiments* (Elsevier Science Ltd.), p.207-216.
- [41] Sharma, V., Eswaran, V., and Chakraborty, D., 2020, "Computational Analysis of Transverse Sonic Injection in Supersonic Crossflow Using RANS Models," *Journal of Fluids Engineering* **142**(6).
- [42] Klebanoff, P., 1955, "Characteristics of turbulence in a boundary layer with zero pressure gradient," *NACA Tech. Report*, 1247.
- [43] Bradshaw, P., Ferriss, D.H., and Atwell, N.P., 1967, "Calculation of boundary layer development using the turbulent energy equation," *J. Fluid Mech.* **28**(3):593-616.
- [44] Champagne, F.H., Harris, V.G., and Corrsin, S., 1970, "Experiments on nearly homogeneous turbulent shear flow," *Journal of Fluid Mechanics* **41**(1):81-139.
- [45] Smits, A.J., and Dussauge, J.P., 2006, *Turbulent shear layers in supersonic flow* (Springer Science & Business Media).

- [46] Duan, L., Beekmam, I., and Martin, M.P., 2011, "Direct numerical simulation of hypersonic turbulent boundary layers. Part 3. Effect of Mach number," *Journal of Fluid Mechanics* **672**:245.
- [47] Rose, W.C., and Murphy, J.D., 1973, "Ratio of Reynolds shear stress to turbulence kinetic energy in a boundary layer," *The Physics of Fluids* **16**(6):935-937.
- [48] Smits, A.J., and Muck, K.C., 1987, "Experimental Study of Three Shock Wave/Turbulent Boundary Layer Interactions," *Journal of Fluid Mechanics* **182**:291-314
- [49] Kuntz, D.W., Amatucci, V.A., and Addy, A.L., 1987, "Turbulent Boundary-Layer Properties Downstream of the Shock-Wave Boundary-Layer Interaction," *AIAA J.* **25**(5):668-675
- [50] Fernando, E.M., and Smits, A.J., 1990, "A supersonic turbulent boundary layer in an adverse pressure gradient," *Journal of Fluid Mechanics* **211**:285-307.
- [51] Doerffer, P., Hirsch, C., Dussauge, J.P., Babinsky, H., and Barakos, G., 2010, *Unsteady Effects in Shock Wave Induced Separation* (Springer).
- [52] Yaravintelimath, A., Raghunandan, B.N., and Morinigo, J.A., 2016, "Numerical prediction of nozzle flow separation: Issue of turbulence modeling," *Aerospace Science and Technology* **50**:31-43.
- [53] Georgiadis, N., and Yoder, D., 2013, "Recalibration of the shear stress transport model to improve calculation of shock separated flows", in *51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, AIAA-685.
- [54] Jie, T., and Jie, J., 2011, "Stress Limiter Consideration for $k-\omega$ Turbulence Models in Shock-Wave/Turbulent Boundary-Layer Interactions in Supersonic and Hypersonic Flows," in *20th AIAA Computational Fluid Dynamics Conference*, AIAA-3980.
- [55] Tharwat, A., El-Samanoundy, R., and El-Baz, M., 2016, "Considerations of Stress Limiter for the SST Turbulence Model in Dual Throat Nozzle Predictions," in *Ninth International Conference on Computational Fluid Dynamics (ICCFD9)*, Istanbul, Turkey, p.1-15.
- [56] Pope, S.B., 1975, "A more general effective-viscosity hypothesis," *Journal of Fluid Mechanics* **72**(2):331-340.

- [57] Wilcox, D.C. *Turbulence Modeling for CFD* 3rd ed. (DCW, Industries, Inc.).
- [58] Larsson, J., and Lele, S.K., “Direct numerical simulation of canonical shock/turbulence interaction,” *Phys Fluids* **21**(12):126101.
- [59] Jamme, S., Cazalbou, J.B., Torres, F., and Chassaing, P., 2002, “Direct numerical simulation of the interaction between a shock wave and various types of isotropic turbulence,” *Flow, Turbulence and Combustion* **68**(3):227-268.
- [60] Vemula, J.B., and Sinha, K., 2002, “Reynolds stress models applied to canonical shock-turbulence interaction,” *textJournal of Turbulence* **18**(7):653-87.
- [61] Raje, P., and Sinha, K., 2016, “A physically consistent and numerically robust k - ϵ model for computing turbulent flows with shock waves,” *Computers & Fluids* **136**:35-47.
- [62] Rodi, W., 1976, “A new algebraic relation for calculating the Reynolds stresses,” *ZAMM* **56**:219.
- [63] Craft, T.J., Launder, B.E., and Suga, K., 1996, “Development and application of a cubic eddy-viscosity model of turbulence,” *International Journal of Heat and Fluid Flow* **17**(2):108-115.
- [64] Loyau, H., Batten, P., and Leschziner, M.A., 1998, “Modelling shock/boundary-layer interaction with nonlinear eddy-viscosity closures,” *Flow, turbulence and combustion* **60**(3):257-282.
- [65] Leschziner, M.A., Loyau, H., and Apsley D.D., 2000, “Prediction of shock/boundary-layer interaction with non-linear eddy-viscosity models,” in *European Congress on Computational Methods in Applied Sciences and Engineering*, CDR Proceedings ECCOMAS.
- [66] Gatski, T.B., and Jongen, T., 2000, “Nonlinear eddy viscosity and algebraic stress models for solving complex turbulent flows,” *Progress in Aerospace Sciences* **36**(8):655-682.
- [67] Hellsten, A., and Laine, S., 2000, “Explicit algebraic Reynolds-stress modelling in decelerating and separating flows,” in *Fluids 2000 Conference and Exhibit*, p.2313.
- [68] Barakos, G. and Drikakis, D., 2000, “Investigation of nonlinear eddy-viscosity turbulence models in shock/boundary-layer interaction,” *AIAA J.* **38**(3):461-469.

- [69] Lindblad, I., Wallin, S., Johansson, A., Friedrich, R., Lechner, R., Krogmann, P., Schuelein, E., Courty J.C., Ravachol, M., and Giordano, D., 1998, "A prediction method for high speed turbulent separated flows with experimental verification," in *29th AIAA, Fluid Dynamics Conference, AIAA paper:2547*.
- [70] Wallin, S., and Johansson, A.V., 2000, "An explicit algebraic Reynolds stress model for incompressible and compressible turbulent flows," *Journal of fluid mechanics* **403**:89-132.
- [71] Vemula, J.B., and Sinha, K., 2020, "Explicit algebraic Reynolds stress model for shock-dominated flows," *International Journal of Heat and Fluid Flow* **85**:108680.
- [72] Leschziner, M.A., 2006, "Modelling turbulent separated flow in the context of aerodynamic applications," *Fluid dynamics research* **38**(2-3):174-210.
- [73] Rung, T., Lubcke, H., Franke, M., Xue, L., Thiele, F., and Fu, S., 1999, "Assessment of explicit algebraic stress models in transonic flows," in *Engineering Turbulence Modelling and Experiments : Elsevier Science Ltd.*:659-668.
- [74] Franke, M., Wallin, S., and Thiele, F., 2005, "Assessment of explicit algebraic Reynolds-stress turbulence models in aerodynamic computations," *Aerospace science and technology* **9**(7):573-581.
- [75] Sinha, K., and Balasridhar, S.J., 2013, "Conservative formulation of the k - ϵ turbulence model for shock-turbulence interaction," *AIAA J.* **51**(8):1872-1882.
- [76] Gatski, T.B., and Speziale, C.G., 1993, "On explicit algebraic stress models for complex turbulent flows," *Journal of fluid Mechanics* **254**:59-78.
- [77] Speziale, C.G., Sarkar, S., and Gatski, T.B., 1991, "Modelling the pressure-strain correlation of turbulence: an invariant dynamical systems approach," *Journal of fluid mechanics* **227**:245-272.
- [78] Launder, B.E., Reece, G.J., and Rodi, W., 1975, "Progress in the development of a Reynolds-stress turbulence closure," *Journal of fluid mechanics* **68**(3):537-566.
- [79] Launder, B. E., and Sharma, B.I., 1974, "Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc," *Letters in heat and mass transfer* **2**:131-137.
- [80] Wilcox, D.C., 1988, "Reassessment of the scale-determining equation for advanced turbulence models," *AIAA J.* **26**(11):1299-1310.

- [81] Jongen, T., and Gatski, T.B., 1998, "General explicit algebraic stress relations and best approximation for three-dimensional flows," *International Journal of Engineering Science* **36**(7-8):739-763.
- [82] Kovasznay, L.S.G., 1953, "Turbulence in supersonic flow," *Journal of the Aeronautical Sciences* **20**:657-674.
- [83] Hannappel, R., and Friedrich, R., 1995, "Direct numerical simulation of a Mach 2 shock interacting with isotropic turbulence," *Applied Scientific Research* **54**(3):205-221.
- [84] Lee, S., Lele, S.K., and Moin, P., 1993, "Direct numerical simulation of isotropic turbulence interacting with a weak shock wave," *J Fluid Mech* **251**:533-562.
- [85] Mahesh, K., Lele, S.K., and Moin, P., 1997, "The influence of entropy fluctuations on the interaction of turbulence with a shock wave," *J Fluid Mech* **334**:353-379.
- [86] Jamme, S., Cazalbou, J.B., Torres, F., and Chassaing, P., 2002, "Direct numerical simulation of the interaction between a shock wave and various types of isotropic turbulence," *Flow, Turbulence and Combustion* **68**(3):227-268.
- [87] Larsson, J., Bermejo-Moreno, I., and Lele, S.K., 2013, "Reynolds and Mach number effects in canonical shock-turbulence interaction" *J Fluid Mech* **717**:293-321.
- [88] Gatski, T.B., and Bonnet, J.P., 2013, *Compressibility, turbulence and high speed flow*, (Academic Press).
- [89] Ribner, H.S., 1953, *Convection of a pattern of vorticity through a shock wave*, NACA Report 1164 .
- [90] Veera, V.K., and Sinha, K., 2009, "Modeling the effect of upstream temperature fluctuations on shock/homogeneous turbulence interaction," *Phys Fluids* **21**(2):025101.
- [91] Sinha K., 2012, "Evolution of enstrophy in shock/homogeneous turbulence interaction," *J Fluid Mech* **707**:74-110.
- [92] Sinha, K., and Balasridhar, S.J., 2013, "A conservative turbulence model for shock dominated flows," in *The 5th European Conference for Aeronautics and Space Sciences*, Munich, Germany.
- [93] Schulein, E., 2006, "Skin-friction and heat flux measurements in shock / boundary layer interaction flows," *AIAA J.* **44**(8):1732-1741.

- [94] Roy, S., and Sinha, K., 2019, "Turbulent Heat Flux Model for Hypersonic Shock-Boundary Layer Interaction," *AIAA J.* **57**(8):3624-9.
- [95] Cecora, R.D., Radespiel, R., Eisfeld, B., and Probst, A., 2015, "Differential Reynolds-stress modeling for aeronautics," *AIAA J.* **53**(3):739-755.
- [96] Marvin, J.G., Brown, J.L., and Gnoffo, P.A., 2013, *Experimental Database with Baseline CFD Solutions: 2-D and Axisymmetric Hypersonic Shock-Wave/Turbulent-Boundary-Layer Interactions*, NASA TM-2013-216604, NASA Ames Research Center, Moffett Field, CA.
- [97] Gnoffo, P., Berry, S., and Van Norman, J., 2011, "Uncertainty Assessments of 2D and Axisymmetric Hypersonic Shock Wave-Turbulent Boundary Layer Interaction Simulations at Compression Corners," in *42nd AIAA Thermophysics Conference*, p.3142.
- [98] Holden, M., Wadhams, T., MacLean, M., and Mundy, E., 2010, "Experimental studies of shock wave/turbulent boundary layer interaction in high Reynolds number supersonic and hypersonic flows to evaluate the performance of CFD codes,' in *40th Fluid Dynamics Conference and Exhibit*, p.4468.
- [99] McDaniel, J., Fletcher, D., Hartfield, R., and Hollo, S., 1991, "Transverse injection into Mach 2 flow behind a rearward-facing step-A 3-D, compressible flow test case for hypersonic combustor CFD validation," in *3rd International Aerospace Planes Conference*, p.5071.
- [100] Hartfield Jr, R.J., Hollo, S.D., and McDaniel, J.C., 1993, "Planar measurement technique for compressible flows using laser-induced iodine fluorescence," *AIAA journal* **31**(3):483-490.
- [101] Eklund, D.R., Fletcher, D.G., Hartfield Jr, R.J., Northam, G.B., and Dancey, C.L., 1995, "A comparative computational/experimental investigation of Mach 2 flow over a rearward-facing step," **Computers & fluids** **24**(5):593-608.
- [102] Halupovich, Y., Natan, B., and Rom, J., 1999, "Numerical solution of the turbulent supersonic flow over a backward facing step," *Fluid Dynamics Research* **24**(5):251.
- [103] Khan, O.U., and Arshed, G., 2017, "High Resolution Numerical Schemes and Supersonic Flow over a Backward-Facing Step," in *55th AIAA Aerospace Sciences Meeting*, p.1434.

- [104] Karimi, A., Wijeyakulasuriya, S., and Nalim, M., 2012, "Numerical study of supersonic flow over backward-facing step for scramjet application," in *48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, p.4001.
- [105] Huang, W., Pourkashanian, M., Ma, L., Ingham, D.B., Luo, S.B., and Wang, Z.G., 2011, "Investigation on the flameholding mechanisms in supersonic flows: backward-facing step and cavity flameholder" *Journal of Visualization* **14**(1):63-74.
- [106] Liu, H., Wang, B., Guo, Y., Zhang, H., and Lin, W., 2013, "Effects of inflow Mach number and step height on supersonic flows over a backward-facing step," *Advances in Mechanical Engineering* **5**, p.147916.
- [107] Soni, R.K., Arya, N., and De, A., 2015, "March. Assessment of RANS based models in a supersonic flow," in *AIP Conference Proceedings* (AIP Publishing LLC), Vol. 1648, No.1, p.030039.
- [108] Soni, R.K., Arya, N., and De, A., 2017, "Characterization of turbulent supersonic flow over a backward-facing step," *AIAA Journal* **55**(5):1511-1529.
- [109] Herrin, J.L., and Dutton, J.C., 1994, "Supersonic base flow experiments in the near wake of a cylindrical afterbody," *AIAA journal* **32**(1):77-83.
- [110] Chuang, C., and Chieng, C., 1996, "Supersonic base-flow computation using higher-order closure turbulence models." *Journal of spacecraft and rockets* **33**(3):374-380.
- [111] Papp, J., and Ghia, K., 2001, "Application of the RNG turbulence model to the simulation of axisymmetric supersonic separated base flows," in *39th Aerospace Sciences Meeting and Exhibit*, p.727.
- [112] Sahu, J., 1994, "Numerical computations of supersonic base flow with special emphasis on turbulence modeling," *AIAA journal* **32**(7):1547-1549.
- [113] Simon, F., Deck, S., Guillen, P., and Sagaut, P., 2006, "Reynolds-averaged Navier-Stokes/large-eddy simulations of supersonic base flow," *AIAA journal* **44**(11):2578-2590.
- [114] Zhuo, C.F., Zou, Y.B., and Xu, P., 2018, "Numerical Simulation of the Supersonic Base Flows with Emphasis on Turbulence Model," *Journal of Applied Science and Engineering* **21**(2):205-212.

- [115] Brinckman, K., Rodebaugh, G., and Dash, S., 2013, "Towards a Unified $k-\epsilon$ Turbulence Model for the Practical Analysis of Aeropropulsive Flows," in *51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, p.392.
- [116] Forsythe, J.R., Hoffmann, K.A., Cummings, R.M., and Squires, K.D., 2002, "Detached-eddy simulation with compressibility corrections applied to a supersonic axisymmetric base flow" *J. Fluids Eng.* **124**(4):911-923.
- [117] Park, S.H., Acharya, S., and Kwon, J.H., 2005, "An Improved Formulation of $k\epsilon$ Turbulence Models for Supersonic Base Flow," in *17th AIAA Computational Fluid Dynamics Conference*, p.4707.
- [118] Sandberg, R.D., and Fasel, H.F., 2006, "Investigation of supersonic wakes using conventional and hybrid turbulence models," *AIAA journal* **44**(9):2071-2083.
- [119] Krishnamurty, V.S., and Shyy, W., 1997, "Study of compressibility modifications to the $k-\epsilon$ turbulence model," *Physics of Fluids* **9**(9):2769-2788.
- [120] Papp, J., Kenzakowski, D., and Dash, S., 2002, "January. Calibration and validation of EASM turbulence model for jet flowfields," in *40th AIAA Aerospace Sciences Meeting & Exhibit*, p.855.
- [121] Ayyalasomayajula, H., Kenzakowski, D., Papp, J., and Dash, S., 2005, "Assessment of $k\epsilon$ /EASM Turbulence Model Upgrades for Analyzing High Speed Aeropropulsive Flows," in *43rd AIAA Aerospace Sciences Meeting and Exhibit*, p.1101.
- [122] Desikan, S.L.N., Murugan, B., Srinivasan, K., and Sajan, S., 2015, "Twin Jet Interaction and Reverse Flow," *Journal of Spacecraft and Rockets* **52**(6):1577-1585.
- [123] Tam, C.K., 1995, "Supersonic jet noise," *Annual Review of Fluid Mechanics* **27**(1):17-43.
- [124] Pack, D.C., 1948, "On the formation of shock-waves in supersonic gas jets: two-dimensional flow," *Q J Mechanics Appl Math* **1**(1):1-17.
- [125] Cheng, T.S., and Lee, K.S., 2005, "Numerical simulations of underexpanded supersonic jet and free shear layer using WENO schemes," *International journal of heat and fluid flow* **26**(5):755-770.
- [126] Cumber, P.S., Fairweather, M., Falle, S.A.E.G., and Giddings, J.R., 1994, "Predictions of the structure of turbulent, moderately underexpanded jets".

- [127] Fairweather, M., and Ranson, K.R., 2006, "Prediction of underexpanded jets using compressibility-corrected, two-equation turbulence models," *Progress in Computational Fluid Dynamics, an International Journal* **6**(1-3):122-128.
- [128] Lakshmanan, B., and Abdol-Hamid, K.S., 1994, "Investigation of supersonic jet plumes using an improved two-equation turbulence model," *Journal of Propulsion and Power* **10**(5):736-741.
- [129] Rodionov, A.V., 2010, "On the use of Boussinesq approximation in turbulent supersonic jet modeling," *International journal of heat and mass transfer* **53**(5-6):889-901.
- [130] Molchanov, A.M., and Myakochin, A.S., 2018, "Numerical simulation of high-speed flows using the algebraic Reynolds stress model," *Russian Aeronautics* **61**(2):236-243.
- [131] Musial, N.T., and Ward, J.J., 1961, "Base flow characteristics for several four-clustered rocket configurations at mach numbers from 2.0 to 3.5," *NASA TN D-1093*, National Aeronautics and Space Administration, Washington DC.
- [132] Goethert, B.H., 1961, "Base flow characteristics of missiles with cluster-rocket exhausts," *Aerospace Engineering* **20**(3):28.
- [133] Page, R.H., and Dixon, R.J., 1963, "Base Heating on a Multiple Propulsion Nozzle Missile," *AIAA paper* **63**(179).
- [134] Lamb, J.P., Abbud, K.A., and Lenzo, C.S., 1970, "A theory for base pressures on multinozzle rocket configurations," *Journal of Spacecraft and Rockets* **7**(4):451-457.
- [135] Marion, E.D., Daniels, D.J., Herstine, G.L., and Burge, G.W., 1962, "Exhaust Reversal from Cluster Nozzles - A New Flow Model," *ARS Paper* **62**-2706.
- [136] Taylor, R.A., and Tou, P.P., 1967, "Convective Heat Transfer in the Base Region of Large Space Boosters with Clustered Engines," *Procedure Heat Transfer Fluid Mechanics Institute* (Stanford University Press).
- [137] Goethert, B.H., and Barnes, L.T., 1960, "Some studies of the flow pattern at the base of missiles with rocket exhaust jets," *AEDC-TR-58-12*, ARO Inc. Arnold AFS TN.

- [138] Charczenko, N., and Hayes, C., 1962, “Jet effects at supersonic speeds on base and afterbody pressures of a missile model having single and multiple jets,” *NASA TN D-2046*, National Aeronautics and Space Administration, Washington DC, 1963.
- [139] Cubbage, J.M., 1965, “Investigation of Exhaust Backflow from a Simulated Cluster of Three Wide-spaced Rocket Nozzles in a Near-space Environment,” *NASA TN D-3016*, National Aeronautics and Space Administration, Hampton VA Langley Research Center.
- [140] Wasko, R.A., and Cover, T.L., 1967, “Experimental investigation of base flow field at high altitudes for configurations of four and five clustered nozzles,” *NASA TM X-1371*, National Aeronautics and Space Administration, Lewis Research Center, Cleveland, Ohio.
- [141] Brewer, E.B., and Craven, C.E., 1969, “Experimental investigation of base flow field at high altitude for a four engine clustered nozzle configuration,” *NASA TN D-5164*, National Aeronautics and Space Administration, Washington DC.
- [142] Nallasamy, R., Kandula, M., Duncil, L., and Schallhorn, P., 2008, “Numerical Simulation of the Base Flow and Heat Transfer Characteristics of a Four-Nozzle Clustered Rocket Engine,” in *40th Thermophysics Conference*, pp.4128.
- [143] Ebrahimi, H., 1997, “Numerical investigation of multi-plume rocket phenomenonology,” in *33rd Joint Propulsion Conference and Exhibit*, pp.2942.
- [144] Wang, T.S., 1995, “Numerical analysis of base flowfield for a four-engine clustered nozzle configuration,” *Journal of Propulsion and Power* **11**(5):1076-1078.
- [145] Deere, K., Elmiligui, A., and Abdol-Hamid, K.S., 2012, “USM3D Simulations of Saturn V Plume-Induced Flow Separation,” *Journal of Spacecraft and Rockets* **49** (4):679-690.
- [146] Zhou, Z., Liang, X., Zhao, C., Le, G., and Ding, Y., 2020, “Investigations of Base Thermal Environment on Four-Nozzle Liquid Launch Vehicle at High Altitude,” *Journal of Spacecraft and Rockets* **57**(1):49-57.
- [147] Reddy, D.S.K., and Sinha, K., 2009, “Hypersonic turbulent flow simulation of Fire II reentry vehicle afterbody,” *Journal of Spacecraft and Rockets* **46**(4):745-757.
- [148] Reddy, D.S.K., and Sinha, K., 2011, “Effect of Chemical Reaction Rates on Aero-heating Predictions of Reentry Flows,” *Journal of Thermophysics and Heat Transfer* **2025**(1):21-33.

- [149] Reddy, D.S.K., Saikia, B., and Sinha, K., 2014, "Effect of High-Enthalpy Air Chemistry on Stagnation Point Heat Flux," *Journal Of Thermophysics And Heat Transfer* **28**(2):356-359.
- [150] Pasha, A.A., 2011, *Application of shock-unsteadiness model to hypersonic shock/turbulent boundary-layer interaction*, Ph.D. Dissertation, Dept. of Aerospace Engineering, Indian Institute of Technology Bombay, Mumbai, 400076, India.
- [151] Wright, M.J., Candler, G.V., and Bose, D., 1998, "Data-parallel line relaxation method for the Navier-Stokes equations," *AIAA J.* **36**(9):1603-1609.
- [152] MacCormack, R.W., and Candler, G.V., 1989, "The solution of the Navier-Stokes equations using Gauss-Seidel line relaxation," *Computers & fluids* **17**(1):135-150.
- [153] Dupont, P., Piponnier, S., Sidorenko, A., and Debieve, J.F., 2008, "Investigation by particle image velocimetry measurements of oblique shock reflection with separation," *AIAA J.* **46**(6):1365-1370.
- [154] Holden, M.S., Wadhams, T., MacLean, M., and Dufrene, A., 2013, "Measurements in Regions of ShockWave/Turbulent Boundary Layer Interaction from Mach 4 to 10 for Open and Blind Code Evaluation/Validation," in *21st AIAA Computational Fluid Dynamics Conference, AIAA Paper*, p.2836.
- [155] Seiner, J., and Norum, T., 1979, "Experiments of shock associated noise of supersonic jets," in *12th Fluid and Plasma Dynamics Conference*, p.1526.
- [156] Seiner, J., and Norum, T., 1980, "Aerodynamic aspects of shock containing jet plumes," in *6th Aeroacoustics Conference*, p.965.
- [157] Sarkar, S., Erlebacher, G., Hussaini, M., and Kreiss, H.O., 1991, "The analysis and modelling of dilatational terms in compressible turbulence," *J Fluid Mech* **227**:473-493.
- [158] Haidinger, F.A., and Friedrich, R., 1993, "Computation of shock wave/turbulent boundary layer interactions using a two-equation model with compressibility corrections," *Applied Scientific Research* **51**(1-2):501-505.
- [159] Hamed, A., and Vogiatzis, C., 1997, "Overexpanded Two-Dimensional-Convergent-Divergent Nozzle Flow Simulations, Assessment of Turbulence Models," *Journal of Propulsion and Power* **13**(3):444-5.

- [160] Quadros, R., Sinha, K., and Larsson, J., 2016, "Turbulent energy flux generated by shock/homogeneous-turbulence interaction," *J Fluid Mech* **796**:113-157.
- [161] Quadros, R., and Sinha, K., 2016, "Modelling of turbulent energy flux in canonical shock-turbulence interaction," *Intl. J. Heat and Fluid Flow*.
- [162] Druguet, M.C., Candler, G.V., , and Nompelis, I., 2005, "Effects of Numerics on Navier-Stokes computations of hypersonic double-cone flows." *AIAA J.* **43**(3):616-623
- [163] Sinha, K., and Singh, R.S., 2016, "Numerical Error in the k - ϵ turbulence model applied to eddy-viscous shock waves," *AIAA J.* **54**(11):3673-3678.
- [164] Settles, G. S., and Dodson, L.J., 1994, "Hypersonic shock/boundary-layer interaction database: New and corrected data," *NASA Contractor Report 177638*, NASA Ames Research Center, Moffett Field, CA.
- [165] Ardonceau, P.L., 1984, "The structure of turbulence in a supersonic shock-wave/boundary-layer interaction," *AIAA J.* **22**(9):1254-62.
- [166] Bradshaw, P., 1967, "Irrotational fluctuations near a turbulent boundary layer," *Journal of Fluid Mechanics* **27**(2):209-30.
- [167] Zhang, C., Duan, L., and Choudhari, M., 2018, "Direct Numerical Simulation Database for Supersonic and Hypersonic Turbulent Boundary Layers," *AIAA J.* **56**(11):4297-4311.
- [168] Van Driest, E.R., 1951, "Turbulent boundary layer in compressible fluids," *Journal of the Aeronautical Sciences* **18**(3):145-60.
- [169] Dutton, J., Herrin, J., Molezzi, M., Mathur, T., and Smith, K., 1995, "January. Recent progress on high-speed separated base flows," in *33rd Aerospace Sciences Meeting and Exhibit*, p.472.
- [170] Bhagatwala, A., and Lele, S.K., 2009, "A modified artificial viscosity approach for compressible turbulence simulations," *Journal of computational physics* **228**(14):4965-4969.
- [171] <https://turbmodels.larc.nasa.gov/sst.html>; Last accessed 26 May 2021.

List of Publications

Peer-reviewed Journals

1. **Raje P.** and Sinha, K. "A physically consistent and numerically robust $k-\epsilon$ model for computing turbulent flows with shock waves", Computers & Fluids, Volume 136, pages 35-47, 2016. *Submitted on 7th December, 2015, Accepted on 24th May, 2016 and Available online from 25th May, 2016.*
2. **Raje P.** and Sinha, K. "Anisotropic SST turbulence model for shock-boundary layer interaction", *Under review*, Computers & Fluids

International Conferences

1. **Raje P.** and Sinha, K. "Three-dimensional simulation of rocket nozzles with multi-jet interaction using shock-unsteadiness model" *Presented in AIAA aviation forum's 49th Fluid Dynamics Conference*", USA.
2. **Raje P.** and Sinha, K. "An anisotropic SST turbulence model for shock-boundary layer interaction" *To be presented in AIAA aviation forum's 51st Fluid Dynamics Conference*", USA.

National Conferences

1. **Raje P.**, Saikia B. and Sinha K. "Numerical investigation of axisymmetric under-expanded supersonic jets" *Presented in National Aerospace Propulsion Conference (NAPC-2017)*, March 15-17, 2017, IIT Kanpur.
2. Vemula, J. B., **Raje P.**, Singh, R., Roy, S. and Sinha K. "Parameteric study of the performance of two-dimensional Scramjet Intake", 18th Annual AeSI CFD Symposium, August 10-11, 2016, CSIR-National Aerospace Laboratories, Bangalore, India.

The author of this dissertation wrote these papers and manuscripts in part or in full, performed the required numerical simulations, and analyzed the results of the simulations. The co-authors on these works provided feedback during these studies, assisted in editing the manuscripts, and were instrumental in determining the model problem.

Acknowledgements

I would like to express my deep sense of gratitude to my advisor, Prof. Krishnendu Sinha, Dept. of Aerospace Engg., Indian Institute of Technology (IIT) Bombay, Mumbai, India, for his invaluable help and guidance. I am highly indebted to him for giving his critiques on my work and for the thought provoking discussions. His guidance during the model development and simulations was immense, and is duly appreciated.

I acknowledge SpaceTime supercomputing facility at IIT Bombay for the computing time, which was very helpful for performing the computationally intensive multi-jet interaction simulations.

I thank my fellow labmates (past and present), Drs. Russell and Bijaylakshmi, Rachit, Yogesh, Jagadish, Subhajit, Pranav, Rahul, and Pankaj for their help and support. Special thanks are due to our previous lab secretary, Mukesh, running errands for me during this long tenure of mine.

Last but not the least, I thank my mother and sisters for their continuous support and encouragement during my Ph.D. at IIT Bombay. Their contribution in every aspect of my education and life, cannot be overstated.

Pratikkumar Vikramark Raje
IIT Bombay