JSST Journal of Space Science and Technology

P-ISSN: 2008-4560
E-ISSN: 2423-4516

Journal of Space Science and Technology

Vol. 18, Special Issue, pp. 52-59, 2025
https://doi.org/10.22034/JSST.2025.1525
Journal Homepage: https://www.jsstpub.com

Original Research Paper

Boundary Layer Transition and Re-Laminarization in the Nozzle of a Shock Tunnel–a Numerical Study

Shahed Malekipour[1], Iman Bahman Jahromi[2], and Meisam Mohammadi Amin[3]*

1,2. Aerospace Research Institute, Ministry of Science, Research and Technology, Tehran, Iran
3. Technical University of Denmark

ARTICLE INFO

Article History:

Received 21 January 2025
Revised 07 February 2025
Accepted 16 February 2025
Available Online 23 February 2025

Keywords:

Boundary layer transition
Computational fluid dynamics
Hypersonic nozzle flow
Nozzle starting flow
Shock tunnel

ABSTRACT

The boundary layer behavior in hypersonic nozzle of a shock tunnel has been investigated by means of the computational fluid dynamics simulations. The state of boundary layer at the nozzle outlet can highly affect the downstream flow passing around the test model in the shock tunnel test section. Focusing on the transitional boundary layer simulation, results of the optimal nozzle's steady-state flow indicate that the boundary layer enters the throat region, undergoes a phase transition, and then returns to a laminar state due to re-acceleration. The turbulence intensity in this region, coupled with the width of the transition zone, increases with higher upstream stagnation pressure. Furthermore, simulation of the unsteady starting flow shows that the passage time of unsteady waves and the quasi-steady region is good enough to have a steady state boundary layer assumption. The formation of shock and expansion wave systems towards the downstream diaphragm and the reflection and expansion waves towards the upstream shock tube were well-simulated after the diaphragm rupture.

* Corresponding Author's E-mail: dr.mohammadiamin@gmail.com

How to Cite this Article:
S. Malekipour, I. Bahman Jahromi, and M. Mohammadi Amin, "Boundary layer transition and re-laminarization in the nozzle of a shock tunnel–a numerical study," *Journal of Space Science and Technology*, Vol. 18, Special Issue, pp. 52-59, 2025, https://doi.org/10.22034/JSST.2025.1525.

1. INTRODUCTION

The exploration of flow patterns in shock tunnels is one of the fascinating areas for the development and optimization of hypersonic test equipment [1]. In many hypersonic flows through convergent-divergent nozzles, the boundary layer is subjected to transition condition or even experiences a significant reduction in turbulence intensity [2]. The growth rate of the laminar or turbulent boundary layer has a substantial impact on the flow quality at the exit [3]. So, numerous studies have been conducted to investigate the status of the boundary layer in such applications. Gu and Herbert, have studied the capabilities and limitations of the available hypersonic equipment [4]. One of the major drawbacks of conventional hypersonic wind tunnels and shock tunnels is the presence of noise due to upstream flow oscillations, often one or two orders of magnitude higher than flight conditions. The dominant noise in these oscillations is acoustic noise emanating from the turbulent boundary layer on the diverging nozzle walls and they might have a noticeable effect on boundary layer transition and other phenomena in test section. One of the main challenges in high speed flow research is the accurate prediction of boundary layer transition and re-laminarization. In many high speed flows through convergent-divergent nozzles, the boundary layer is subjected to transition condition and also experiences a significant reduction in turbulence intensity, up to 100%. Researches have shown that pressure gradients resulting from flow curvature and geometry in hypervelocity boundary layers can significantly affect the flow turbulence level [2].

Since the early 21st century, most numerical analyses of high speed nozzles have used the SST (Shear Stress Transport) model. The SST model combines the advantages of the k-ε and k-ω models. Duan et al. used it to simulate the performance of hypersonic nozzles [5]. Ownbey used the SST model for designing a high-enthalpy hypersonic nozzle [6]. Collen et al. and Hoffman et al. also utilized this model for studying nozzle of Mach 7 in test facilities [7,8]. Shear Stress Transport model is not capable to predict the transition or re-laminarization of the boundary layer.

The topic of re-laminarization in the boundary layer of high speed nozzles emerged in the late 20th century and continues to be a significant focus in this field. Back et al. observed that the actual boundary layer thickness was lower than expected while examining heat transfer rates on the wall of a conical nozzle at high Reynolds numbers (around 105 to 106 based on throat diameter) in the JPL laboratory. Further investigation led them to conclude that the turbulent boundary layer seemed to become laminar again. They proposed a criterion based on the properties of the boundary layer edge [9]. Kemp et al. studied the boundary layers of high speed nozzles from Mach 20 to 47. Despite the desirable pressure gradient, they found that the flow becomes turbulent in the throat and becomes laminar again in that region [10]. Kreskovsky et al. investigated the effect of severe acceleration of the flow on the turbulent boundary layer by comparing analytical and experimental results. The findings showed that even in the presence of high turbulence levels upstream, the boundary layer becomes laminar in the throat, and its thickness significantly decreases [11]. Jentink investigated various definitions of flow acceleration criteria using turbulence models to identify boundary layer re-laminarization in nozzles [12]. The study of transition conditions and re-laminarization of the boundary layer continued on various geometries until around 2005 when turbulence models with the capability to detect transition zones started being used in simulations.

Abraham et al. utilized the three-equation Menter model (γ-SST) for defining a criterion to identify the re-laminarization of turbulent flow in internal flows. He defined a criterion as the ratio of turbulence production to its dissipation, stating that if it is less than one, the boundary layer tends to become laminar [13]. Bader et al. studied the capability of the four-equation model (γ-Reθ-SST) in predicting the re-laminarization of the flow over a turbine airfoil under rapid acceleration conditions. The results of these analyses were compared with DNS results, demonstrating that the four-equation model performs well [14].

Kiselv et al. investigated heat transfer in a small supersonic nozzle under conditions of re-laminarization of the boundary layer. They used both the three-equation and four-equation turbulence models. The heat transfer coefficient in the throat region was higher in the three-equation model than the actual value, while in the four-equation model, it was lower than the experimental value, indicating that four-equation model has a greater tendency toward flow re-laminarization conditions [15].

So, one of the main challenges in hypersonic flow research is the accurate prediction of boundary layer transition. The performance of the shock tunnel hypersonic flow is highly sensitive to various disturbances and pressure fluctuations, introducing complexity to the analysis. To ensure a uniform flow at the outlet, a nozzle design is considered here that was optimized previously by the authors [16] based on a fully turbulent Reynolds-Averaged Navier-Stokes (RANS) steady-state CFD model (Ansys Fluent) which similar to most numerical analyses of hypersonic nozzles, have used SST (Shear Stress Transport) turbulence model that combines the advantages of the k-ε and k-ω models [6].

Given the development of new numerical methods for studying the transition region and the lack of resources regarding the efficiency of these methods in converging-diverging hypersonic nozzle flows, this paper focuses on studying the transition region and investigates the effect of upstream pressure in the transition region. The transition and re-laminarization can be distinguished by transitional models in hypersonic nozzle instead of classic re-laminarization parameters for the first time. Then, the flow behavior and interaction of transient unsteady starting waves in nozzle with the boundary layer are examined. Comparing the simulation results with the experimental data obtained from the ARIST shock tunnel [17] shows a good agreement and the capability of CFD for understanding complex phenomena related to the formation of shock and expansion waves and their interaction with the boundary layer. This unsteady simulation shows that the duration of the steady state flow establishment is enough for such assumption in the boundary layer study.

2. METHODOLOGY

In this work, the ARIST shock tunnel nozzle is considered, designed to create test conditions with hypersonic Mach numbers (Mach 6) and high stagnation temperatures (around 600 Kelvin). An overall view of the ARIST shock tunnel facility, the nozzle, and the test section is depicted in Figure (1). The shock tunnel generally consists of a high-pressure tube (driver section), a low-pressure tube (driven section), the main diaphragm section, a secondary diaphragm, a converging-diverging nozzle, test section and a dump tank. Complete specifications and the details of this shock tunnel are presented in [17].

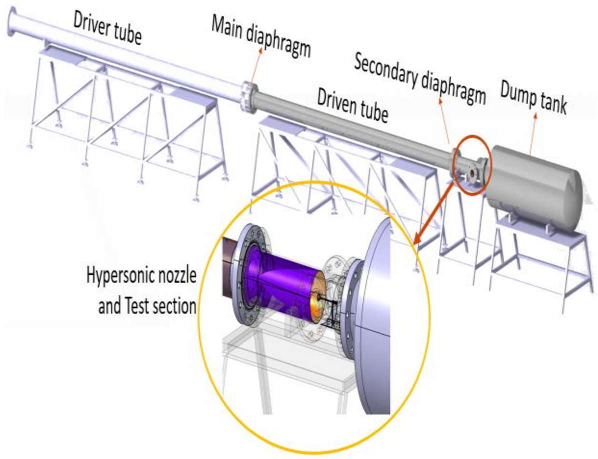


Fig. 1. ARIST reflected shock tunnel [10].

Based on Reynolds-Averaged Navier-Stokes formulation for the fluid dynamics modeling [18], the four-equation model was chosen to investigate the boundary layer in the selected nozzle. The Langtry-Menter four-equation model, also known as γ-Reθ or γ-Reθ-SST, has been developed for predicting the boundary layer transition with RANS models. For this purpose, the intermittency equation and the transition onset momentum thickness equation are added to the model. For that, the intermittency variable is multiplied in the turbulence kinetic energy production term, and its dissipation term [18]:

$$\frac{\partial \rho k}{\partial t}+\frac{\partial \rho u_j k}{\partial x_j} = \gamma_{eff}.P_k-\min\left(\max\left(\gamma_{eff},0.1\right),1.0\right).\beta^*\rho\omega k + \frac{\partial}{\partial x_j}\left[\left(\mu+\sigma_k\mu_t\right)\frac{\partial k}{\partial x_j}\right] \tag{1}$$

For γ and Re_θ:

$$\frac{\partial \rho\gamma}{\partial t}+\frac{\partial \rho u_j\gamma}{\partial x_j}=P_\gamma-E_\gamma+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_f}\right)\frac{\partial\gamma}{\partial x_j}\right]$$

$$\frac{\partial \rho\tilde{R}e_{\theta t}}{\partial t}+\frac{\partial \rho u_j\tilde{R}e_{\theta t}}{\partial x_j}=P_{\theta t}+\frac{\partial}{\partial x_j}\left[\sigma_{\theta t}(\mu+\mu_t)\frac{\partial\tilde{R}e_{\theta t}}{\partial x_j}\right]$$

The correlation equations used for transition prediction are based on the idea that the boundary layer transition initiates when reaching a critical value of the Transition onset momentum thickness Reynolds number, $Re_{\theta t}$. However, this criterion is meaningful only along the wall and in one dimension. Therefore, Langtry and Menter introduced the transport variable of $\tilde{R}e_{\theta t}$, which can be applied throughout the flow field. It is expressed as an empirical relationship as a function of the pressure gradient and the turbulence intensity of the free stream. The production term in the transport equation of $\tilde{R}e_{\theta t}$ is formulated as follows:

$$P_{\theta t} = c_{\theta t}\rho \frac{\rho U^2}{500\mu}(Re_{\theta t} - \tilde{R}e_{\theta t})(1.0 - F_{\theta t}) \qquad (2)$$

The production and dissipation terms of the γ transport equation are formulated as follows:

$$P_\gamma = F_{length} c_{a1} \rho S [\gamma F_{onset}]^{0.5}(1 - c_{e1}\gamma)$$

$$E_\gamma = c_{a2}\rho\Omega\gamma F_{turb}(c_{e2}\gamma - 1)$$

$$F_{onset} = \max\,(F_{onset2} - F_{onset3}, 0)$$

$$F_{onset3} = \max\,\left(1 - \left(\frac{R_T}{2.5}\right)^3, 0\right)$$

$$F_{onset2} = \min\,(\max(F_{onset1}, F_{onset1}{}^4), 2.0)$$

$$F_{onset1} = \frac{Re_v}{2.193 Re_{\theta c}}$$

$$Re_v = \frac{\rho S d^2}{\mu}$$

The production term in the intermittency transport equation leads to an increase in intermittency and, consequently, triggers the transition, while the dissipation term promotes the re-laminarization of the boundary layer. The F_{length} and F_{onset} terms are essential for determining the length of the transition region length and location, depending on the $\tilde{R}e_{\theta t}$. The term F_{turb} depends only on local turbulence parameters, k and ω. The term $Re_{\theta c}$ in a function of $\tilde{R}e_{\theta t}$ and d is the nearest distance of arbitrary point in flow field to the wall. S and Ω are the strain rate and the vorticity, respectively. In laminar boundary layer separation conditions, to ensure the separated flow quickly enters the transition and turbulence phase and reattaches to the surface, the parameter γ_{eff} is introduced. In fact, γ is forced to have a value higher than one in separation conditions [19].

3. RESULTS AND DISCUSSIONS

For the validation of the CFD model, the boundary layer analysis of references [20, 21] for hypersonic flow over a flat plate has been studied. The results of the three- and four-equation SST models, are compared with the experimental results from [12], and the numerical results from [13], that showed a good agreement for the heat flux on the wall Figure (2). It is observed that the three-equation model predicts the transition location upstream of experimental onset, while the four-equation model predicts it a bit downstream. However, the behavior of the experimental data in the transitional region is much closer to the four-equation model. Thus, the four-equation model was chosen to investigate the boundary layer in the selected nozzle.

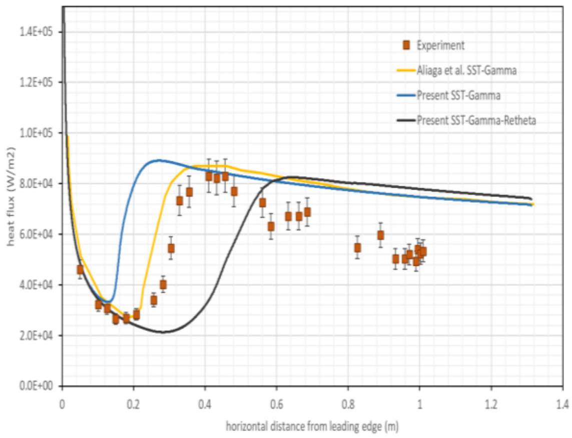


Fig. 2. Heat flux on a flat plate in Mach 11.

A no-slip adiabatic wall is considered for the nozzle and axisymmetric condition for the centerline. The simulations are performed with ANSYS Fluent 2016. This solver uses a finite volume approach with cell-centered variable arrangement. Density-based solution is used which is more advantageous for highly compressible applications. The fully coupled implicit algorithm provides 2nd order spatial discretization of the flow and turbulence equations, and it includes an AUSM scheme for the numerical convective flux. The independence of the solution from the grid is examined with four different computational grids. Additionally, the value of y^+ should be kept below one. The convergence criterion is the residual values to be less than 10^{-6}.

The computational domain, grid, and boundary conditions are illustrated in Figure (3). The comparison of Mach number distribution at the outlet of nozzle using numerical and experimental data in Figure (4) shows a good match between the experimental and simulation results (<3% error).

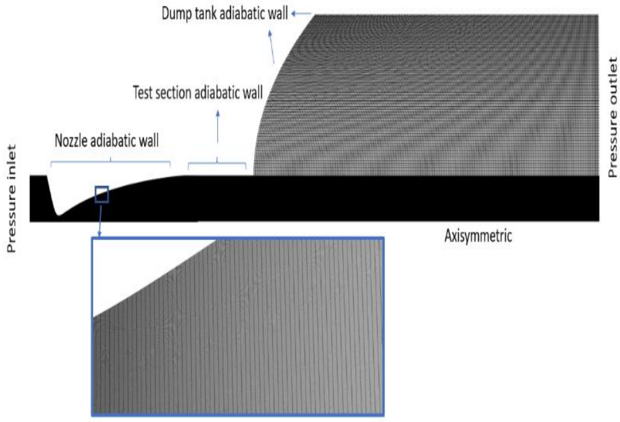


Fig. 3. Computational domain, grid, and boundary conditions.

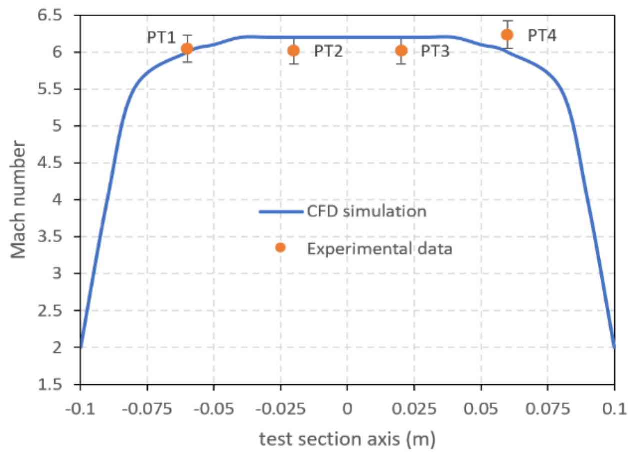


Fig. 4. Validation of Mach number at outlet.

For a better understanding of the flow nature, results of the shear stress distribution are presented under the assumptions of laminar flow and fully turbulent flow. It should be noted that here, the independence of the solution from the upstream turbulence intensity is examined and it was observed that with an increase in turbulence intensity, a negligible increase in wall shear stress occurs in the early centimeters of the convergent section. As shown in Figures (5) and (6), the flow in the region before the throat deviates from laminar state and moves towards transition and turbulence. After passing through the throat with acceleration, it returns to a laminar state again. The highlighted region in the figures indicates the transitional region. It is also can be observed that with an increase in pressure, the transitional and turbulent regions become larger and move towards the divergent section and the nozzle exit.

Additionally, the values of wall shear stress in the transitional analysis approach those obtained from the turbulent flow analysis, to the point where at 25 bars, these two curves coincide. Based on further numerical results, increasing the total pressure up to 40 bar, will result the boundary layer to be turbulent along almost the entire length of the nozzle.

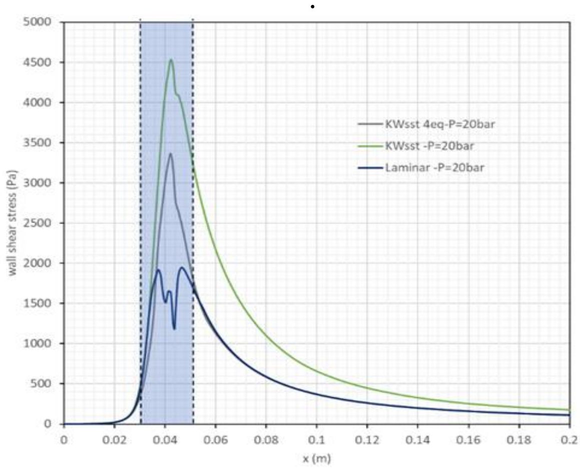


Fig. 5. Wall shear stress on the nozzle wall at total pressure of 20 bar.

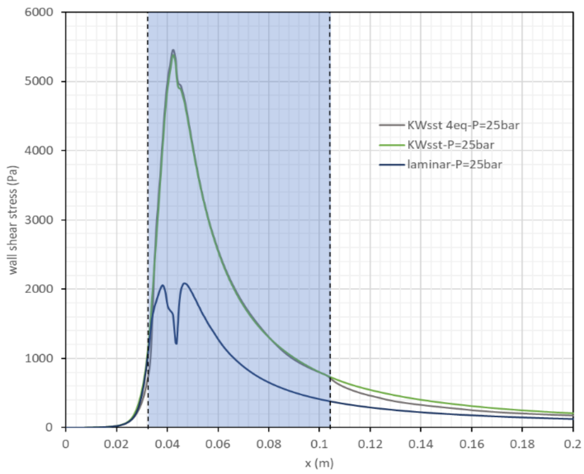


Fig. 6. Wall shear stress on the nozzle wall at total pressure of 25 bar.

In the next step, it is necessary to examine the boundary layer conditions for the optimal nozzle in the design conditions with the base nozzle. In Figure (7), the wall shear stress distribution for the base and optimal nozzles in the design conditions are compared. Due to the reduction in the length of the convergent section of the optimal nozzle, the shear stress level in the throat area of this nozzle is higher, the length of the optimal nozzle is shorter, but by controlling the curve of the divergent section and reducing the total length of the nozzle, the wall shear stress level in the divergent section and the outlet of the optimal nozzle is about 50% lower than that of the base nozzle.

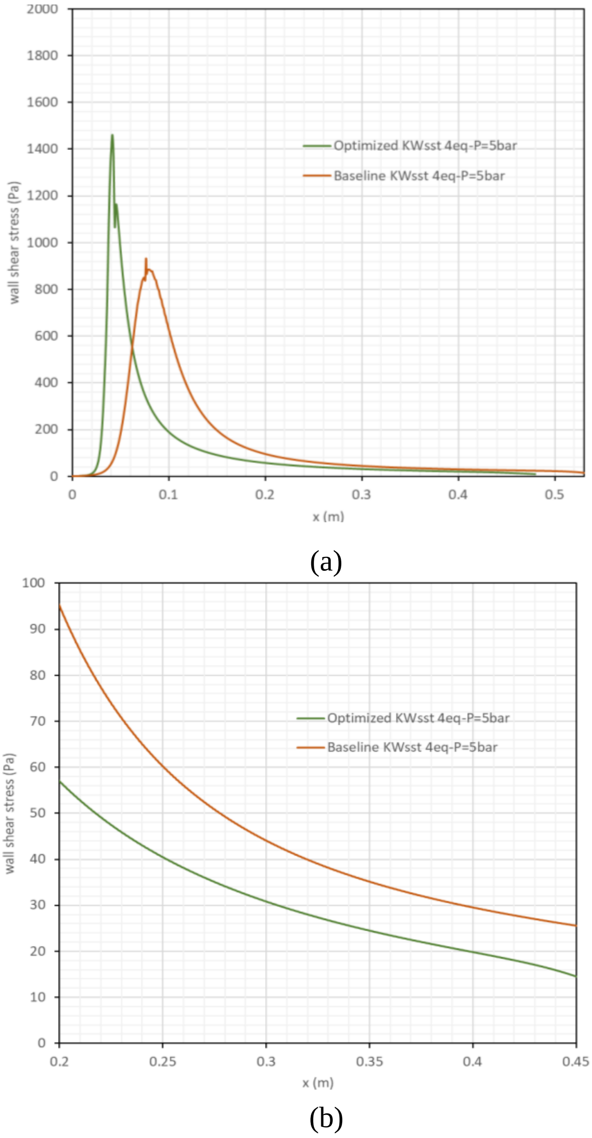


Fig. 7. Base and optimal nozzle wall shear stress distribution a) entire nozzle, b) end of divergent section.

In Figure (8), the velocity distribution in the nozzle outlet plane is presented in terms of the nozzle outlet radius. Also, in this figure, the effect of the analysis assuming laminar, turbulent and transient flow can be seen. It can be seen that the analysis with the four-equation model is consistent with the laminar analysis. Also, the boundary layer thickness in the optimal nozzle based on the 99% free stream velocity criterion has been reduced by about 40%.

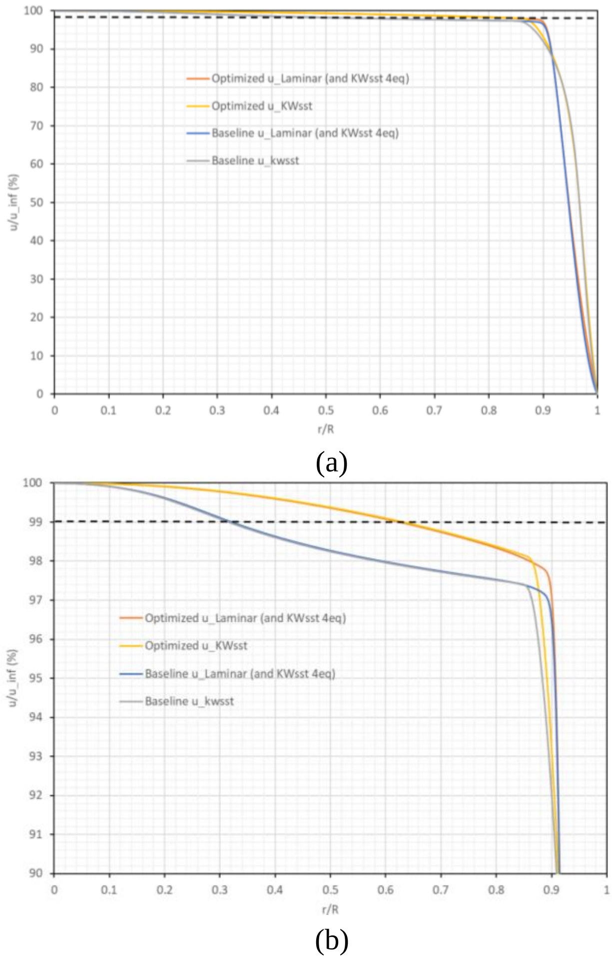
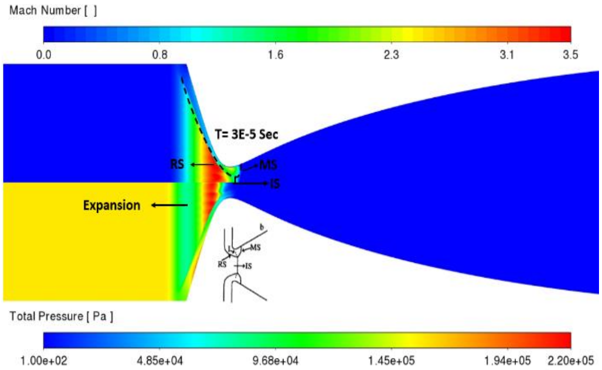
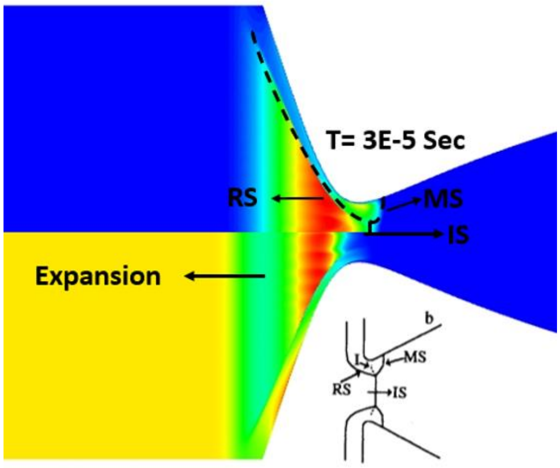


Fig. 8. Velocity distribution vs the outlet radius a) over the entire radius b) the boundary layer edge.

For the unsteady numerical simulation of hypersonic nozzle flow, the finite volume method with second-order accuracy and axisymmetric assumptions has been employed. The simulations have been conducted for both laminar and turbulent flow conditions. The working fluid, consistent with the test, is considered to be ideal air. The transient part is solved with implicit method. Also, the nozzle has a contoured convergent section with a secondary diaphragm upstream of nozzle. The diaphragm effect is simulated by initializing the upstream region of the nozzle with reservoir pressure and temperature. To observe the effect of diaphragm burst, radial oscillation is considered in the initial condition. The formation of the primary and secondary shock waves (PS & SS) moving downstream the nozzle, as well as the generation of reflective waves (RS) and expansion waves moving upstream towards the shock tube, is studied. Figures (7) and (8) show two instances of 0.03, 0.180 ms after secondary diaphragm burst, respectively. For the 0.18 ms snapshot, there are a strong secondary shock wave following an expansion region and a weaker primary shock wave heading downstream the nozzle while a reflected shock wave and expansion waves are going toward upstream of the shock tube.



(a)



(b)

Fig. 9. Mach number and total pressure contours after secondary diaphragm burst; a) entire nozzle, b) magnified throat region.

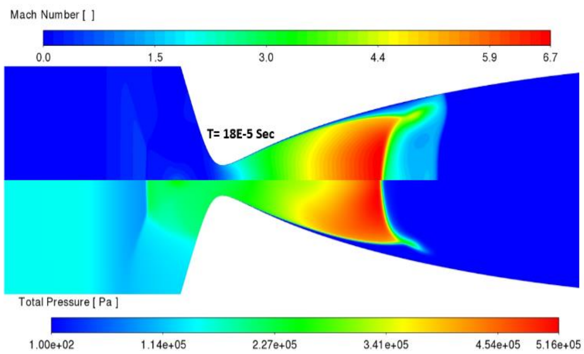


Fig. 10. Mach number and total pressure contours at 0.03 milliseconds after secondary diaphragm burst.

The starting time from numerical and experimental results is about 1.5 msec which is consistent with the calculations in design phase and it shows that the remaining time of steady state flow is an appropriate assumption for boundary layer study.

4. CONCLUSIONS

Mach 6 nozzle boundary layer was investigated via numerical simulation including a suitable four-equation model for studying the transitional flow. The results revealed the presence of a boundary layer transition in the throat region, and with the acceleration of the flow, the boundary layer reverts to a laminar state. As the upstream pressure of the nozzle increases, i.e. an increase in Reynolds number, turbulence intensity in the transitional region amplifies, causing a delay in the flow re-laminarization towards the nozzle exit. At high pressures, the entire divergent part of the nozzle can be situated within the turbulent boundary layer. Furthermore, the unsteady flow at the initiation of the nozzle was numerically examined. The formation of shock and expansion wave systems towards the downstream diaphragm and the reflection and expansion waves towards the upstream shock tube were well-simulated after the diaphragm rupture. It is observed that the passage time of unsteady waves and the quasi-steady flow establishment is good enough to have a steady state boundary layer assumption.

CONFLICT OF INTEREST

The authors declare that they have no conflict of interest.

REFERENCES

[1] I. Kim, S. Lee, G. Park, and J.K. Lee, "Overview of flow diagnosis in a shock tunnel," *International Journal of Aeronautical and Space Sciences*, vol. 18, no. 3, pp 425-435, 2017, https://doi.org/10.5139/IJASS.2017.18.3.425.

[2] N. R. Tichenor, "Characterization of the influence of a favorable pressure gradient on the basic structure of a Mach 5.0 high Reynolds number supersonic turbulent boundary layer," Ph.D. dissertation, Texas A&M University, 2010.

[3] F. Lu and D. Wilson, "Survey of short duration, hypersonic and hypervelocity facilities," in *25th Plasmadynamics and Lasers Conference*, Springs,

CO, USA, 1994, Paper 94-2491, https://doi.org/10.2514/6.1994-2491.

[4] S. Gu and O. Herbert, “Capabilities and limitations of existing hypersonic facilities,” *Progress in Aerospace Sciences*, vol. 113, 2020, Art. no. 100607, https://doi.org/10.1016/j.paerosci.2020.100607.

[5] L. Duan, G. L. Nicholson, J. Huang, K. M. Casper, R. Wagnild, and N. Bitter, “Direct numerical simulation of nozzle-wall pressure fluctuations in a mach 8 wind tunnel.” in *AIAA Scitech 2019 Forum*, San Diego, California, 2019, Paper 0874, https://doi.org/10.2514/6.2019-0874.

[6] J. C. Ownbey, “Preliminary design of a high-enthalpy hypersonic wind tunnel facility and analysis of flow interactions in a high-speed missile configuration,” M.S. thesis, School of Aeronautics and Astronautics West Lafayette, Indiana, 2021.

[7] P. Collen *et al.*, “Development and commissioning of the T6 Stalker Tunnel,” *Experiments in Fluids*, vol. 62, 2021, Art. no. 225, https://doi.org/10.1007/s00348-021-03298-1.

[8] E. N. Hoffman *et al.*, “Characterization of the UTSA mach 7 ludwieg tube,” in *AIAA SciTech 2022 Forum*, San Diego, CA & Virtual, 2022, https://doi.org/10.2514/6.2022-1600.

[9] L. H. Back, R. F. Cuffel, and P. F. Massier, “Laminarization of a turbulent boundary layer in nozzle flow—boundary layer and heat transfer measurements with wall cooling,” *ASME. Journal of Heat andMass Transfe*, vol. 92, no. 3, pp. 333-344, 1970, https://doi.org/10.1115/1.3449668.

[10] J. H. Kemp Jr and F. K. Owen, “Nozzle wall boundary layers at Mach numbers 20 to 47,” *AIAA Journal*, vol. 10, no. 7, pp. 872-879, 1972, https://doi.org/10.2514/3.50239.

[11] J. P. Kreskovsky, S. J. Shamroth, and H. Mcdonald, “Parametric study of relaminarization of turbulent boundary layers on nozzle walls,” NASA, Tech. Rep. CR-2370, 1974.

[12] T. Jentink, “An evaluation of nozzle relaminarization using low Reynolds number K-epsilon turbulence models,” in *31st Aerospace Sciences Meeting*, Reno, NV, USA, 1993, https://doi.org/10.2514/6.1993-610.

[13] J. P. Abraham, E. M. Sparrow, J. C. K. Tong, and D. W. Bettenhausen, “Internal flows which transist from turbulent through intermittent to laminar,” *International Journal of Thermal Sciences*, vol. 49, no. 2, pp. 256-263, 2010, https://doi.org/10.1016/j.ijthermalsci.2009.07.013.

[14] P. Bader, P. Pieringer, and W. Sanz, “On the capability of transition models to predict relaminarization,” in *12th European Conference on Turbomachinery Fluid dynamics & Thermodynamics*, Stockholm, Sweden, 2017, Paper ETC2017-328, https://doi.org/10.29008/ETC2017-328.

[15] N.A. Kiselev, N.S. Malastowski, Y.A. Vinogradov, and A.G. Zditovets, “Experimental and numerical study of heat transfer under laminarization condition in a small size supersonic nozzle,” *International Journal of Thermal Sciences*, vol. 187, 2023, Art. no. 108182, https://doi.org/10.1016/j.ijthermalsci.2023.108182.

[16]S. Malekipour, M. Mohammadi Amin, and I. Bahman Jahromi, “Towards a low noise shock tunnel facility via multi-objective optimization of hypersonic nozzle,” *Journal of Aerospace Engineering*, vol. 34, no. 5, 2021, https://doi.org/10.1061/(ASCE)AS.1943-5525.0001296.

[17] S. Malekipour, I. Bahman Jahromi, and M. Mohammadi Amin, “Experimental investigation of Mach 6 optimal hypersonic nozzle performance in shock tunnel,” *Journal of Aeronautical Engineering*, vol. 25, no. 2, pp. 21-33, 2023, https://doi.org/10.22034/joae.2023.366731.1139.

[18] F. R. Menter, R. B. Langtry, S. R. Likki, Y. B. Suzen, P. G. Huang, and S. Völker, “A correlation-based transition model using local variables—part I: Model formulation,” *Journal of Turbomachinery*, vol. 128, no. 3, pp. 413-422, 2006, https://doi.org/10.1115/1.2184352.

[19] S. Kang, H. C. Shin, S. H. Park, J. Park, and D. Park, “Comparative assessment of modified γ-$Re_{\theta t}$ models for scramjet intake flow analysis,” *International Journal of Aerospace Engineering*, vol. 2021, no. 1, 2021, Art. no. 9916416, https://doi.org/10.1155/2021/9916416.

[20] J. G. Marvin, J. L. Brown, and P. A. Gnoffo, “Experimental database with baseline CFD solutions: 2-D and axisymmetric hypersonic shock-wave/turbulent-boundary-layer interactions,” NASA, Tech. Rep. TM-2013-216604. 2013.

[21] C. Aliaga, K. Guan, J. Selvanayagam, J. Stokes, V. Viti, and F. Menter, “Hypersonic applications of the laminar turbulent transition SST model in ANSYS Fluent,” in *AIAA Aviation 2020 Forum*, Virtual Event, 2020, https://doi.org/10.2514/6.2020-3290.

.