SUPERSONIC TURBULENT FLOW MODELLING OVER STEPS AND CAVITIES

by

LEE KAH KHENG

Thesis submitted in fulfilment of the requirements for the

Bachelor Degree of Engineering (Honours) (Aerospace Engineering)

July 2021

ENDORSEMENT

I, Lee Kah Kheng hereby declare that all corrections and comments made by the supervisor and examiner have been taken consideration and rectified accordingly.

(Signature of Student)

Date: 11 July 2021

(Signature of Supervisor)

Name: Dr Sarjit Singh Sidhu Junior

Date: 11 July 2021

DR. NURUL MUSFIRAH MAZLAN
Pensyarah
Pusat Pengajian Kejuruteraan Aeroangkasa
Kampus Kejuruteraan, Universiti Sains Malaysia
14300 Nibong Tebal, Pulau Pinang
Tel : 04-599 5913

(Signature of Examiner)

Name: Dr Nurul Musfirah Mazian

Date: 11 July 2021

DECLARATION

This thesis is the results of my own investigation, except where otherwise stated and has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any other degree.

(Signature of Student)

Date: 11 July 2021

ACKNOWLEDGEMENT

First of all, I would like to express my sincerest gratitude to my final year project supervisor, Dr Sarjit Singh Sidhu Junior, for his effort in helping my progress throughout the project. His patience and enthusiasm in aiding me with critical knowledge is my motivation to complete the research project. I really appreciate the knowledge I learned from him that is really helpful throughout the project.

Furthermore, I would like to show my appreciation to the School of Aerospace Engineering, Universiti Sains Malaysia, who gives me the opportunity to accomplish my academic project. The facilities provided in CATIA Lab with advanced simulation software is crucial in obtaining the result of my research project.

Also, I would like to thank my final year project cohort, Mr Lim Wei Fong, who always ready in help me out in the research project. We provide mutual support to each other by discussing the topics related to our topic to gain more knowledge and experience for the project.

Last but not least, the mental support from my family and friends is priceless in motivating me throughout the project. I appreciate their encouragement during my though which helps me to stay focus in accomplish the project.

SUPERSONIC TURBULENT FLOW MODELLING OVER STEPS AND CAVITIES

ABSTRACT

Altitude compensating nozzles provide ideal conditions for propulsion, ignoring the effect of atmospheric pressure that will cause the reduction of efficiency. Past experiments discovered the effect of fuel combustion on the surface of nozzles, which causes cross-hatching patterns on the wall due to enthalpy, known as ablation. Cross-hatching patterns act as surface roughness that causes flow disturbance to the supersonic flow field within the nozzle. A numerical experiment is conducted to examine the supersonic turbulent flow behaviour over cross-hatching relief surfaces in order of steps and cavities with different length-to-depth ratios(L/D), including the induced drag as the direct physical effect of flow losses. After getting the fundamental concepts, the numerical experiment setup is started with reviewing the assumptions and limitations, which will be further applied in the simulation afterwards. 2D-modelling is done with Solidworks, following by the meshing in Pointwise. The simulation result from Ansys Fluent is further interpreted and presented with Techplot in the form of graph and contour. The numerical experiment is a parametric study which is 150% to the reference paper, with 14 variants of length-to-depth ratios in the range of 4.17 to 17.21. For closed cavity type flow ($15.39 \leq L/D \leq 22.99$), it yielded larger drag force due to the separation shock found which increased linearly with smaller L/D ratio. The drag force decreased drastically for transition cavity type flow ($L/D = 13.23$) as less flow separation was formed. For open cavity type flow ($4.17 \leq L/D \leq 11.61$), it yielded small drag force as shear layer was completely formed above the cavity with large recirculation regions inside the cavity.

SUPERSONIC TURBULENT FLOW MODELLING OVER 2-D PERIODIC WAVY SURFACES USING CFD

ABSTRAK

Muncung pampasan ketinggian memberikan keadaan yang ideal untuk penggerak dengan mengabaikan kesan tekanan atmosfera yang akan menyebabkan penurunan kecekapan. Eksperimen yang lalu mendapati kesan pembakaran bahan bakar pada permukaan muncung, yang menyebabkan corak penetasan silang di dinding kerana entalpi, yang dikenal sebagai ablasi. Corak penetasan silang bertindak sebagai kekasaran permukaan yang menyebabkan gangguan aliran ke medan aliran supersonik di dalam muncung. Satu eksperimen berangka dilakukan untuk memeriksa tingkah laku aliran turbulen supersonik di atas permukaan pelepasan palang mengikut urutan langkah dan rongga dengan nisbah panjang-ke-kedalaman yang berbeza (L/D), termasuk daya tarikan yang disebabkan sebagai kesan fizikal langsung dari kerugian aliran . Setelah mendapat konsep asas, penyediaan eksperimen berangka dimulakan dengan meninjau andaian dan batasan, yang akan diterapkan lebih lanjut dalam simulasi selepas itu. Pemodelan 2D dilakukan dengan Solidworks, diikuti dengan penyambungan di Pointwise. Hasil simulasi dari Ansys Fluent ditafsirkan lebih lanjut dan disajikan dengan Techplot dalam bentuk grafik dan kontur. Eksperimen berangka adalah kajian parametrik yang 150% dari kertas rujukan, dengan 14 varian nisbah panjang hingga kedalaman dalam lingkungan 4.17 hingga 17.21. Untuk aliran jenis rongga tertutup ($15.39 \leq L/D \leq 22.99$), ia menghasilkan daya seret yang lebih besar kerana kejutan pemisahan yang didapati meningkat secara linear dengan nisbah L/D yang lebih kecil. Daya tarik menurun secara drastik untuk aliran jenis rongga peralihan ($L/D = 13.23$) kerana semakin sedikit

pemisahan aliran terbentuk. Untuk aliran jenis rongga terbuka ($4.17 \leq L / D \leq 11.61$), ia menghasilkan daya seret kecil kerana lapisan ricih sepenuhnya terbentuk di atas rongga dengan kawasan peredaran semula besar di dalam rongga.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

2-D	: 2-Dimensional
3-D	: 3-Dimensional
CFD	: Computational Fluid Dynamics
GIT	: Grid Independence Test
RANS	: Reynolds Averaged Navier-Stokes
SST	: Menter’s Shear Stress Transport
AUSM	: Advection Upstream Splitting Method
OSW	: Oblique Shock Wave
EW	: Expansion Wave
SWBLI	: Shock-wave/boundary-layer interaction
FP	: Flat Plate

LIST OF SYMBOLS

M	: Mach number
H	: thickness of the plate
A_e	: nozzle exit area
q	: specific flow rate in nozzle
M^*	: critical Mach number at nozzle throat
A^*	: critical cross-section area at nozzle throat
h	: depth of the cavity
d^*	: diameter of nozzle throat
l_r	: length of the recession
L	: length of the plate
B	: width of plate model
h_e	: height of flat nozzle exit
h^*	: height of flat nozzle throat
k	: tubulence kinetic energy
ω	: specific dissipation rate
β	: half diamond angle/half rhombic angle
Re	: Reynold's number
y^+	: dimensionless wall distance

S	: effective temperature/Sutherland's constant
T	: static temperature
T_0	: reference temperature
μ	: dynamic viscosity
μ_0	: reference value of viscosity
ρ	: density of the fluid
Pr_L	: laminar Prandtl number
Pr_T	: turbulent Prandtl number
σ^*	: closure coefficient
β^*	: closure coefficient
γ	: blending factor
F_1	: auxiliary function

CHAPTER 1

INTRODUCTION

1.1 Introduction and Overview

With a fixed geometric expansion ratio by design, a nozzle is able to give out optimum thrust which increases the efficiency in performance. At different altitudes, the nozzle will undergo over or under-expansion if the ambient pressure is beyond the designed exit pressure. For instance, Energia with RD-0120 engine, Space Shuttle with RS-25 engine, and Ariane 5 launch vehicle with Vulcain 2 engine are designed for nozzle exit pressure of 0.2 MPa. The nozzles will lose 7 to 9 percent of thrust at over or under-expansion condition. At altitude, a solution to this is using a nozzle attachment although it is prone to ablation due to complex interactions of the combustion products with the materials of the walls.

This study concerns surface roughness. All surfaces have a certain degree of roughness to it, and can have a range of forms and shapes, ranging from roughness on the surface to externally machined shapes on the surface itself. In this work, the focus is on pre-machined roughness elements characteristic of the wall irregularities post ablation. The particular effect of a larger cavity depth, while maintaining the length and the other freestream parameters are focused on in this work.

1.2 Research Background and Motivation

The review by Rudolph J. Swigart which concluded the requirement of the occurence of cross-hatching roughness elements. Worth noting that for supersonic freestream conditions, the compressibility effects, affects the flow field, where changes are highly non-linear pending on various parameters including the those of the roughness elements and the freestream conditions. The thrust lost caused by the surface roughness might reduce the functionality of the nozzle attachment.

To further investigate the consequences, detailed studies were done about the effect of surface roughness in the form of steps and cavities to the flow. The cavity flow is categorized into open cavity, transition cavity and closed cavity based on the flow behaviour within the cavity. Ch'ng Kia Chuan had his investigation on the 2-dimensional geometry cases with different L/D ratio and obtained the drag caused by respective cases via numerical method (Ch'ng *et al.*, 2020). Further studies can be done with some changes on geometry to examine the difference in the results while maintaining the freestream conditions. Meanwhile, the flow behaviour of surface roughness in the form of steps and cavities can be observed in more detailed with the visualization of flow characteristics in the form of contours or plots.

1.3 Problem Statement

Ablation can help to prevent the overheating on the nozzle wall by partially removing the heat from combustion with the ablative material. However, ablation might leave surface roughness with cross-hatching pattern on the wall that will cause additional disturbance to the flow in the nozzle. The flow separation caused by the surface roughness will induce more drag and further increase the thrust lost. Although altitude compensating nozzle can prevent the thrust lost caused by over or under expansion, the ablative material on the wall of nozzle attachment might be a concern on the functionality of the equipment.

Cross-hatching pattern comes with several forms while the effect of surface roughness in the form of steps and cavities is interesting to study as the sudden change in the geometry would cause complex consequences to the flow. The flow behaviour and the drag induced of cavities with different L/D ratio can be well observed to study their effect in qualitative and quantitative method.

1.4 Research Objective

Based on the research problem stated, the study is done with numerical method to achieve objectives including:

- To study the characteristics of the supersonic flow field over the steps and cavities form of surface roughness under over-expanded condition with CFD, and
- To investigate the effect of a smaller length-to-depth ratio (L/D) of the roughness geometry on the flow field properties and corresponding drag induced.

1.5 Overall scope

The scope of this research is about the result of a numerical simulation of a 2-dimensional geometry of steps and cavities. Structured meshing is done to input into the simulation to acquire an accurate result for post-processing. A few contours and plots are plotted to investigate the flow behaviour of surface roughness in the form of steps and cavities with different L/D ratio in a supersonic flow. The drag force and drag coefficient are calculated in simulation for each case to be compared and discussed in the results. Conclusion is made at the end to summarize the observations found in the discussion section and future research recommendations are suggested for further study.

1.6 Thesis Outline

This thesis has a total of five chapters to precisely explain and discuss the whole research topic. The thesis outline includes:

Chapter 1. Introduction – Provide an overview and research background, at the same time stating the objectives of the research and the scope of the study.

Chapter 2. Literature Review – Discussion about the previous studies that is related to this research by pointing out the crucial finding of the respective studies. Starts with cross-hatching pattern, the review highlights the understanding in the effect of surface roughness to the flow as well as the numerical method used in the previous research.

Chapter 3. Methodology – Description of the overall procedure in the research including the review of related past work, problem setup, CFD simulation and post processing of the study. The method used in the simulation is well explained with solid reason on their functionality in solving the related problem.

Chapter 4. Result and Discussion – Express the understanding on the result of the numerical simulation in the form of contours and plot for a better visualization on the flow behaviour of each case. Selected cases are further discussed to examine the abnormal behaviour with the comparison to general behaviour. Data of drag force for respective cases are plotted to investigate the trend of the variation in different L/D ratio.

Chapter 5. Conclusion and recommendations – Concluding remarks are presented by summarizing the highlighted finding in this research. Recommendations on related research in future study are suggested to perform more detailed result and discussions.

CHAPTER 2

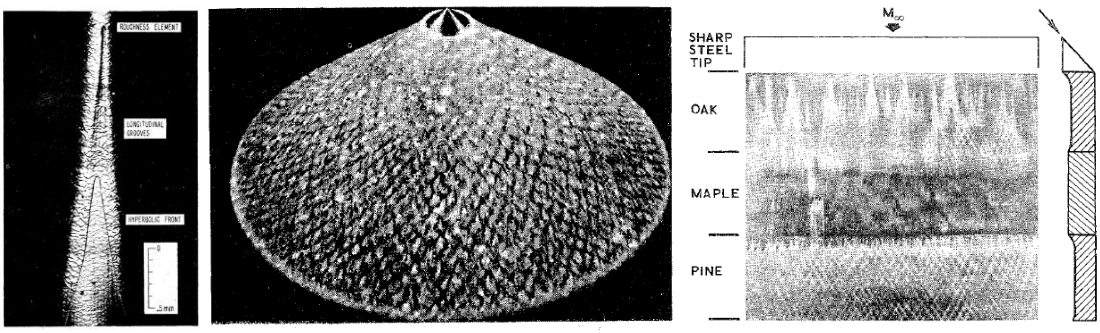
LITERATURE REVIEW

2.1 Formation of Cross-hatched Patterns

An early review is done by Rudolph J. Swigart on the cross-hatching phenomena in 1974 which summarized and concluded theoretical and experimental findings on mechanisms behind the formation of cross-hatched patterns (Swigart, 1974). Experimental research is crucial to complement numerical studies as they are always compared to examine the accuracy and study the restrictions and limitations in different methods. Canning and his co-workers ran one of the earliest experiment on the cross-hatching phenomena to investigates the flow conditions and model the development of cross-hatched patterns (Canning, Wilkins and Tauber, 1968). Streamwise grooves that were observed during the formation of patterns in regions of turbulent flow indicates the presence of longitudinal vortices. Moreover, the research concluded that cross-hatching should not exist in subsonic flow as no spiral angle of intersecting grooves was observed at Mach numbers less than 1.

Larson and Mateer carried out a test on the formation of cross hatching with a hypersonic wind tunnel at Mach 7.4 to identify the aspects of the phenomena including the necessity of supersonic flow for the formation and the effect of geometry on it (Larson and Mateer, 1968) It was clearly shown that supersonic flow is required as the pattern was only formed on the model with a freestream inflow of Mach 1.3 but not on the model tested with a Mach 0.9 inflow. Further observations were done with the increment of pattern depth with time and the morphing and transition into the regmaglypt pattern which is similar to the surface roughness found on meteorites. Moreover, Larson and

Mateer found that both a supersonic flow and a turbulent boundary layer is required but alone they are not sufficient for the cross-hatching formation. A thin boundary layer was concluded as another requirement when a 5° cone without the presence of the patterns was observed to be free of shock waves. Laganelli and Nestler then strengthened the necessity of turbulent flow with their study on the pattern formation on materials made out of various different types of woods where the portion made of oak showed clear signs of turbulent wedges (Laganelli and Nestler, 1968).



(a) (b) (c)

Figure 2.1: Previous experimental results with (a) Turbulent wedge on Lexan model, (b)50° half angle Lucite cone with cross-hatched pattern, and (c) cross-hatched patterns in wood.

2.2 Formation of Cross-hatched Pattern

As mentioned before in the research result of Larson and Mateer, cross-hatching patterns would resemble into another form of roughness called regmaglypts with the increase of time which is similar to the reliefs found on meteorites. Lin and Qun had their studies on the ablation phenomena appearing on various kinds of meteorites (Lin and Qun, 1987). They observed the formation of ablation wedges between the smooth nose region and the regmaglypt region. The ablation wedges had a number of diamond-shaped patterns, which are similar to cross-hatched patterns, which clearly indicated the ablation phenomena in the transition region of the boundary layer. Ablation is a process of the removal of material from the surface of an object under high temperature by pyrolysis which helps to prevent excess heating on the wall.

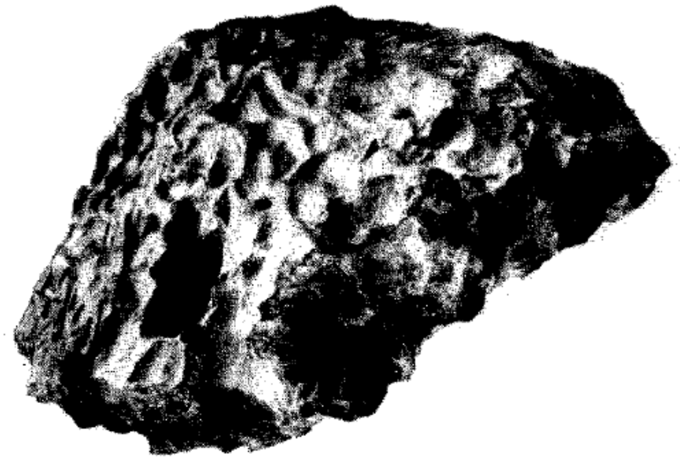


Figure 2.2: Regmaglypt pattern found on meteorites.

Lin and Qun further ran an experimental study on the sequence of the appearance of the ablation patterns before the complete formation of regmaglypts. With Mach numbers in the region of of hypersonic velocities, there was a series of various ablation patterns formed in succession, from smooth surface, narrow grooves, ablation pits, abaltion grooves, ablation wedges, cross hatching and in the end, regmaglypts. Since the Mach number remained unchanged all the way, the results found that Reynolds number of the boundary layer is the main factor affecting the type of ablation pattern (Lin and Qun, 1987).

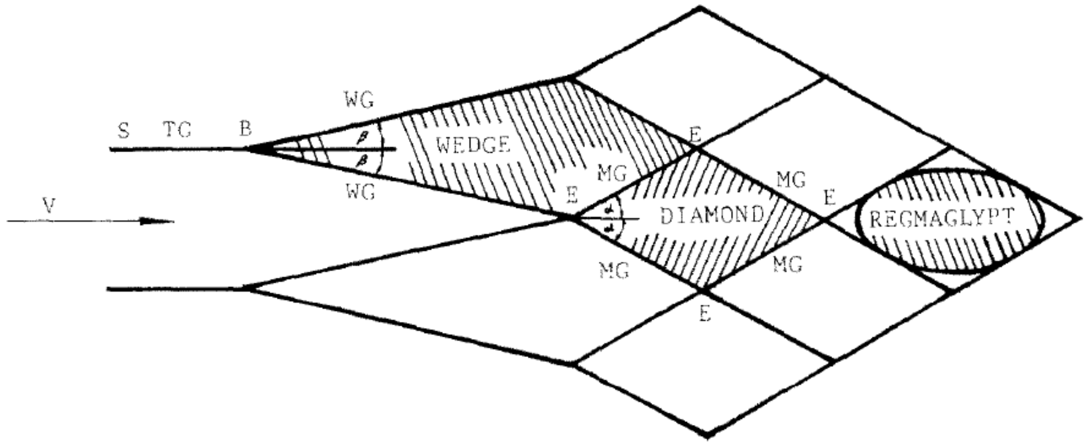


Fig. 7. Schematic of formation of regmaglypt:
V: On coming velocity, *B*: Bifurcation of trailing vortex, α: Mach angle,
S: Turbulent spot, wg: Wedge vortex groove, β: Half wedge angle,
tg: Trailing vortex groove, *E*: Emerged disturbance, mg: Mach vortex groove.

Figure 2.3: Illustration of regmaglypt formation

Lin and Qun illustrated the formation of regmgalypt and gave their understanding based on the review and observation of experimental works which included the cross hatched patterns. The formation starts with a small disturbance caused by any small irregularity on the ablation surface and it will enlarge and induce a streamwise trailing vortex in the transition region. The further growing of the trailling vortex will bifurcate into pair of vortices which are called as wedge vortices. When two neighnouring vortices intersect, a disturbance is generated in the external supersonic flow field and forms a conical Mach wave. After the intersection of conical Mach waves and the turbulent boundary layer, Mach vortex grooves are formed and grows until two neighbouring Mach vortices intersect. This process will repeat and continuously generate new disturbance with the intersections. With higher ablation rate is caused by the pressure rise from the shocks inside the turbulent boundary layer, where in the end, cross hatched patterns are then formed on the ablated surface. The diamond-shaped pattern will gradually become oval or elliptic shaped as called regmaglypts with the elapse of ablation time after the effect of the local wave strength (Lin and Qun, 1987).

2.3 The Onset of Turbulence and Corresponding Flow Behaviour

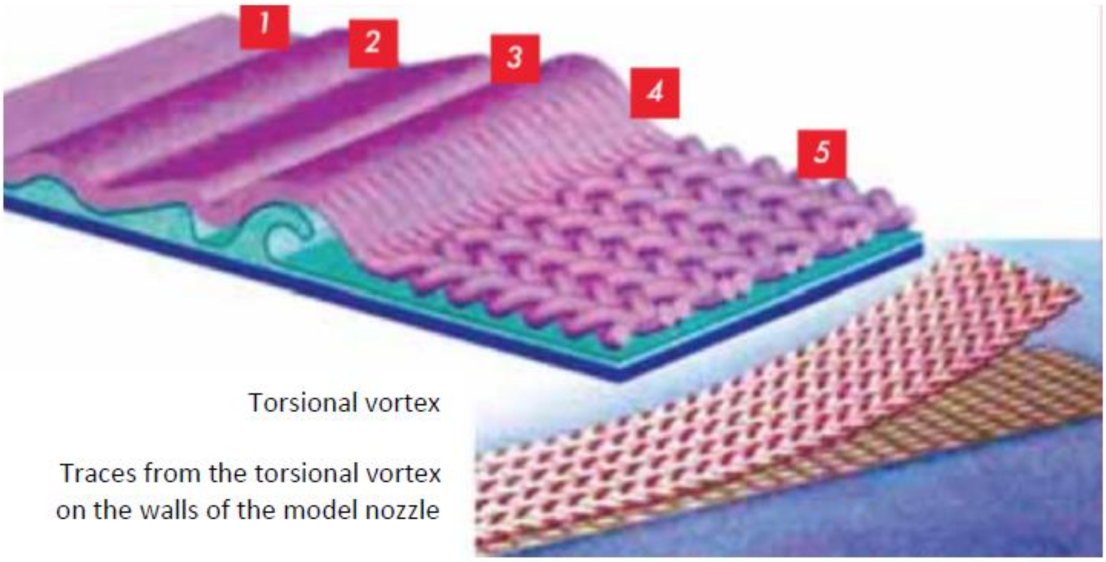


Figure 2.4: Formation of turbulent flow.

Kochetkov conducted a mass ablation experiment within the walls of a solid propellant rocket motor and proposed the following flow development based on the results observed (Kochetkov, 2018). The sequence of the development starts with 1 – laminar flow, 2 – Tolmin-Schlichting waves, 3 – Kelvin-Helmholtz waves, 4 – Taylor-Gertler vortives and 5- torsion harnesses. The vortices are caused by the positive pressure gradient of the flow, forming steep configuration as shown in the Kelvin-Helmholtz waves whereafter Taylor-Gertler vortices are developed. Further pressure gradient will rotate the Taylor-Gertler vortices in pairs into a helical form, as evidenced in Figure 2.3.

2.4 Effects of Surface Roughness the Flow Field

2.4.1 Incompressble Flow

Several studies on the surface roughness effects on an incompressible boundary layer are done. With the experiment of rough pipes with different roughness heights in incompressible turbulent flow, Nikuradse observed that the Reymold number had only a slight effect on the velocity profile on the surface while it was more affected by the relative roughness (Nikuradse, 1950). Meanwhile, Wu and Chritensen had their studies on the mean and turbulent flow features caused by distributed surface roughness element on a subsonic boundary layer (Wu and Christensen, 2007). They discovered that the velocity profile of the roughned wall had a downward shift compared to smooth wall while the velocity in the inner boundary layer was also reduced. On the other hand, the Reynold stress values increases drastically at the region with high roughness scales.

Mejia-Alvarez and Christensen had a similar study in subsonic flow but extended it to investigate the difference in developing flow and developed flow (Mejia-Alvarez and Christensen, 2010). The results found that for a developing flow, only the inner boundary layer was affected by the roughness. When the flow was fully developed, the roughness effects spreaded to a larger portion until entire boundary layer and the flow became self-similar.

2.4.2 Compressible Flow

Bowersox and Latin had their first study on the turbulent measurement for surface roughness effects on a supersonic boundary layer with six variations of surface topologies (Latin and Bowersox, 2000). They observed that there are shock waves and expansion fans occuring when the roughness elements protrude into the supersonic boundary layer. Another investigation found that the dependence is on the roughness elements as the Reynold stresses did not collapse after adjusting the outer parameter, which has a similar result with the research of Wu and Christensen in incompressible flow.