International Journal of Advancements in Mechanical and Aeronautical Engineering– IJAMAE
Volume 2: Issue 1 [ISSN : 2372-4153] pp. 61-65
Publication Date : 30 April, 2015

Vortex Generator Effect on Mittigation of Separation in Transnic Flow Nozzle

Zare Shahneh, A
School of Aeronautics, Transportation and Manufacturing
Cranfield University
United Kingdom

***Abstract*— The effect of tetrahedral sub-boundary layer vortex generator on plummeting normal shock-induced turbulent boundary layer separation has been investigated in computational environment and the result is presented. A good agreement achieved in this work between the pre-assessed experimental and current CFD results, opens the way for further utilization of CFD in the area of shock/boundary layer interaction with 3-D mean compressible flow field, particularly for complicated geometries. The outcome style sheet is illustrated by the portions given in this document.**

Keywords- separation, normal shock wave, vortex generator, CFD, STAR-CD

Introduction

This paper presents the computational investigation of the control of shock induced separation by a Tetrahedral Vortex Generator using STAR-CD. The purposes of the simulation are

- To increase the confidence in the experimental results be achieving good agreement (if possible) between those results and the CFD results

- To verify the physical parameters of flow in a wide area. The experimental measurement facilities are limited in a specific area of the test section, where CFD can show the flow field physical parameters in the entire test section.

- To provide further insight into the underlying physics by obtaining and analyzing flow field behaviour that could not have been obtained using the experimental apparatus that was used in this project.

- To lay the foundation for future numerical work on this topic as is further discussed in the Conclusion section.

I. MOTIVATION

The interaction between a turbulent boundary layer and a shock wave over the wing is a feature frequently encountered in modern aerodynamics. It can be responsible for a large loss in momentum energy. As already mentioned the flow speeds up as it proceeds about the airfoil, the local Mach number at the airfoil surface will be higher than the free-stream Mach number. There eventually occurs a free-stream Mach number called the critical Mach number at which a supersonic point appears somewhere on the airfoil surface, usually near the point of maximum thickness, and indicates that the flow at that point has reached Mach 1. As the free-stream Mach number is increased beyond the critical Mach number and approaches Mach 1, larger and larger regions of supersonic flow appear on the airfoil surface. When regions of subsonic and supersonic flow exist simultaneously on the surface of a body, the flow is transonic. As a transonic flow passes over the upper surface of the wing, the flow initially accelerates and therefore the surface static pressure falls. Therefore, the upper surface usually posses a large region of supersonic flow, which is terminated by a normal shock wave. To attain the equal upper and lower pressure value at the trailing edge, by the effect of shock wave, the flow decelerates and the surface pressure rises toward the trailing edge. Transonic flow pertains to the range of speeds in which flow patterns change from subsonic to supersonic or vice versa, about Mach 0.8 to 1.2. Transonic flow presents a special problem area as neither equations describing subsonic flow nor those describing supersonic flow may be accurately applied to the regime. In order to this supersonic flow to return to subsonic flow, it must pass through a shock wave. Typical shock thickness is in the order of 0.01mm. Underneath the shock location, a separation zone is expected. To verify the effect of s vortex generator to reduce this effect, and following an experimental investigation[1], a computational analysis is recommended.

II. Approaching facilities

Computational Fluid Dynamics in general is the study of a system involving fluid flow and heat transfer by numerical calculation/simulation rather than analytical and experimental approaches. The idea is to use appropriate algorithms to find solutions describing the fluid motion according to the discretized Navier-Stokes equations. It employs mathematical model in the form of a set of partial differential or integral equations and boundary conditions. The discretization is being approached by Finite Element method. Depending on the problem whether the regime is steady or unsteady state, and mostly non-linear equations needed to be applied, a method of solution can be selected.

A. Solution method

By calculating the statistical average of the solution directly, called Reynolds Averaged Numerical Simulation (RANS)

Direct Numerical Simulation (DNS).

By calculating the low frequency modes in space directly, called Large Eddy Simulation (LES)[2],[3].

Applying Unsteady Reynolds Averaged Numerical Simulation (URANS), Semi-Deterministic Simulation (SDS), Very Large Eddy Simulation (VLES) and sometimes Coherent Structure Capturing (CSC).

Compressible flows quite often exhibit near-incompressible flow properties in boundary layers, once the variation of the molecular viscosity with the temperature has been taken into account, as predicted by Morkovin in his famous hypothesis in incompressible flow studying [4] ,[5]

B. STAR-CD code

Many CFD programs have been introduced for general purposes [6], [7], [8]. In the meantime, the idea of investigation of shock induced separation has been also numerically studied by simulation in CFD programs. Aktin et al [9] studied the normal shock wave / boundary layer interaction at Mach between 1.3 to 1.55 both experimental and numerical. BVGK is the one which is used to predict the effect of separation control devices. It is a transonic aerofoil code to model a passive control device. This code combines a numerical solution of the exact potential equation with integral methods for solving the boundary layer and wake equations. The code is developed for passive control aerofoil is called BVGK(P).

Recently, jBAY Vortex Generator has been investigated in Swedish Defence Research Agency [10]. jBAY is a developed and modified model of the Bender-Anderson-Yagle (BAY) model, a vane type sub boundary vortex generator. The Edge CFD flow solver was applied for the study. The first test carried out by a single vortex generator with 7mm high and 49mm long was mounted on a flat plate where the boundary layer thickness at the vortex generator location is 45mm (vortex generator height=0.16 δ). δ is the boundary layer thickness at shock location. The angle of the vortex generator to the freestream flow was 23 degrees and the speed of flow is 34m/s. The second test case was the flow through a circular duct (the RAE M2129 S-duct) and the third case was flow over a flap. An excellent agreement of CFD and experimental investigations are claimed. The model is very attractive as it is easy and simple in implementation with a high capability to predict the results.

The current study is loaned the SAR-CD code which have been used for the numerical investigation of shock induced separation of air passing sub boundary layer vortex generator in the wind tunnel [11].

III. Result and discussion

A. Baseline analysis

In the experimental investigation, all experiments have been done in a supersonic wind tunnel. The wind tunnel is simulated with the same input and boundary conditions as experimental case. It is considered a two dimensional flow regime. Fig. 1 shows the shock wave captured and Lambda form has appeared.

Based on theory, since the flow upstream of shock is supersonic, so for all gaseous with specific heat capacity greater than one, downstream Mach number should be subsonic. Also variations of pressure, density and temperature equations with Mach, it is resulted that the ratio of each of these parameters depends only on upstream Mach number for air. Fig. 2 shows that pressure and temperature are increasing across a shock wave, while the Mach number is decreasing.

Fig. 3 shows the separated area induced by boundary layer shock wave interaction commencing from the beneath of the shock location. It is quite similar to the formation of separation produced in PREPHA program, shown in Fig. 4. [11]

B. Flow simulation with vortex generator

Fig. 5 shows the shock wave captured in the velocity contour, as a sharp reduction of flow velocity is occurred. The slice is the centerline along the tunnel.

Fig.6 shows relative static pressure contour zoomed at shock wave location. It illustrates a sudden increasing of pressure at shock location.

Static pressure contour across the tunnel in the area commencing by shock location to considerably far downstream shock location is almost constant.

The effect of vortex generator can be observed with velocity contour in different height slices (Fig. 7). Slice 0.0001/150, which is the first possible y-coordinate slice to the surface, shows a high variation of velocity magnitude around the vortex generator pair, which can be seen precisely in the zoomed picture as well (Fig. 8).

III. Conclusion

Simulation of a pair of Tetrahedral Vortex Generator in a supersonic wind was performed for flow parameters matching a reference experiment showed the possibility of an accurate numerical prediction of a shock wave/turbulent boundary layer interaction. The shock wave position according to the numerical prediction was only 2.5% away from the experimental finding (that ratio is defined as the difference between shock location and vortex generator location of experiment and CFD over the distance in experiment). The numerical calculation also clarified the effects of the shock wave on the static pressure, total pressure and Mach number distributions. Velocity contours at different stream-normal slices demonstrated the variation of velocity, showing a decrease in the velocity upstream of the vortex generator as it faced a high pressure zone on the obstacle, and a jet flow downstream of the vortex generator location. Velocity contours in streamwise slices showed counter-rotating vortices that evolved along the wind tunnel. The flow analysis showed that three-dimensional effects inside the boundary layer started from the upstream vortex location and progressed up to 400mm downstream of shock wave location. This finding imposes new requirements for any future planned experiment. The work can be extended by further evaluation of the phenomena in the CFD environment, and also other CFD codes where then being compared with experimental outcomes.

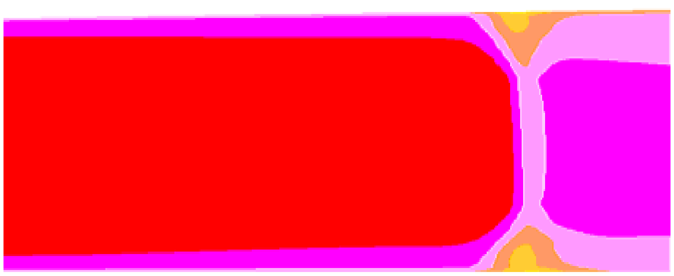

Fig. 1. Formation of shock wave and Lambda in a simulated supersonic wind tunnel

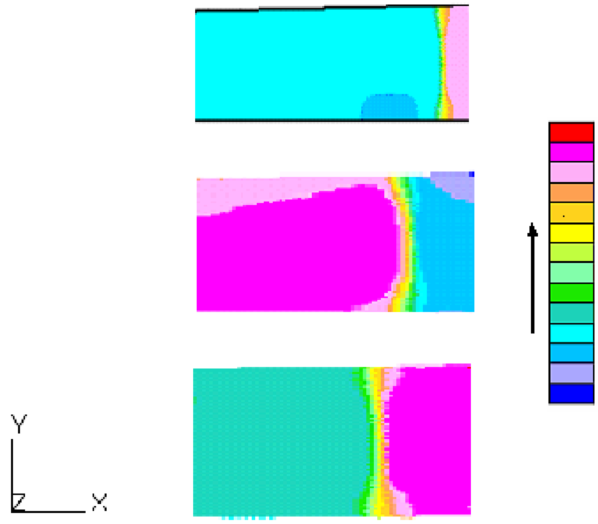

Fig. 2. Effect of shock wave on the fluid properties. Top: static pressure increases, middle: Mach number decreases, bottom: temperature increases. The contour is on the right side, shows value increasing from bottom to top.

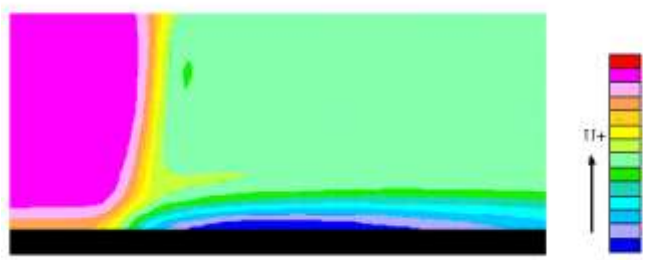

Fig. 3. Separated flow beneath the shock wave. Contour shows velocity rise direction.

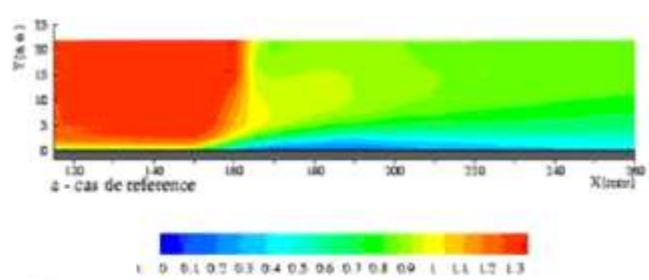

Fig. 4. Shock wave and separation produced in PREPHA program [11]

Fig. 5. Velocity contour at centreline along the test section

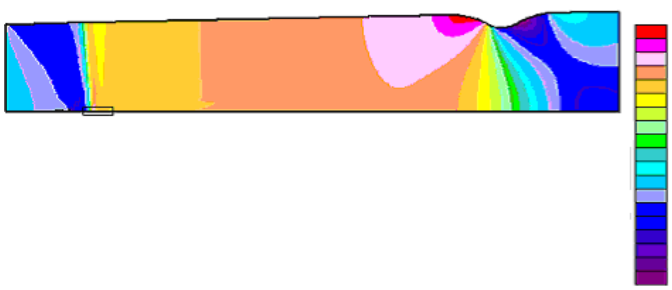

Fig. 6. Relative static pressure contour at centreline, above: along the wind tunnel, bottom: zoomed at shock wave location

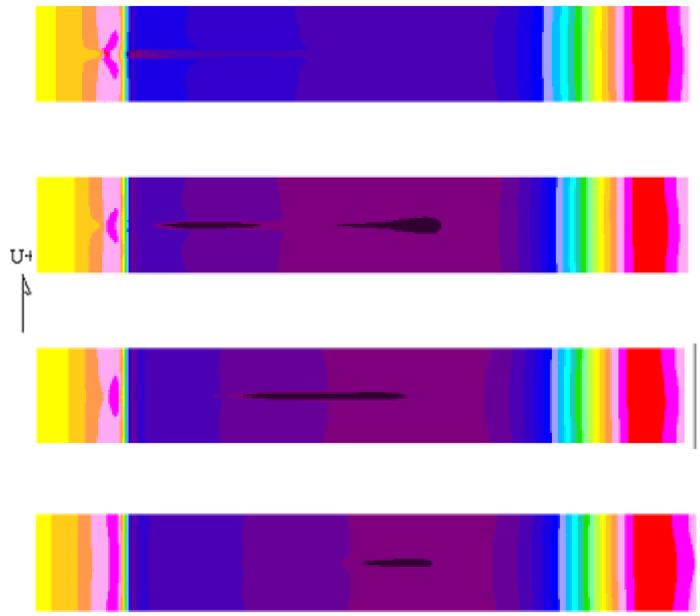

Fig. 7. Top view of velocity contour at different slices in y-coordinate. From the top are slices 0.0001/150(at the surface), 2mm, 4mm, 8mm, 15mm and 25mm above the surface. The contour colour variation is as Fig. 6.

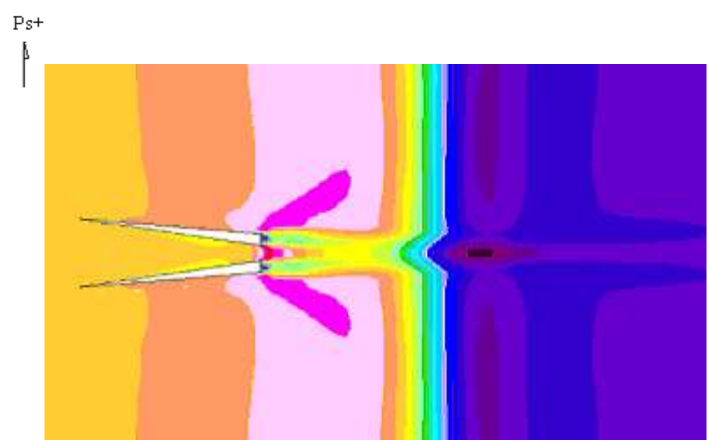

Fig. 8 . Velocity contour zoomed on the shock location zone at 0.0001/150(at the surface). The contour colour variation is as Fig.6.

REFERENCES

[1] Zare Shahneh A., Motallebi F., "Effect of Submerged Vortex Generator on Shock Induced Separation in Transonic Flow" , QMU of London, AIAA, Journal of Aircraft, ISSN 0021-8669, JAIRAM, Vol. 46, pp. 856-863, 2009

[2] Wng, Z.K., "A Source-Extraction Based Coupling Method for Computational Aeroacoustics", U. of Greenwich, 2003

[3] Pierre Sagaut , "Large Eddy Simulation for Incompressible Flows", Springer,1998

[4] Morkovin M.V. "Mecanique de la Turbulence", CNRS, pp.367-380 (1962)

[5] Lesieur Marcel, Foreword for "Large Eddy Simulation for Incompressible Flows", 2000

[6] Fluent Incorporated, 10 Cavendish Court, Lebanon, NH 03766-1442, 2005

[7] Integrated Environmental Solution, 141 ST. James Road, Glassgow, G4 0LT

[8] A. Kneer_ , E. Schreck_ , M. Hebenstreit_ , A. G¨oszler, Industrial mixed OpenMP / Mpi, ICCM Institute of Computational Continuum Mechanics GmbH, Hamburg, Germany

[9] Aktin C. J., Squire L. C., " A Study of the Interaction of a Normal Shock Wave with a Turbulent Boundary Layer at Mach numbers between 1.3 and 1.55 ", European Journal of Mechanics, V. 11, No. 1, 1992

[10] Jirasek A., "A vortex Generator Model and its Application to Flow Control", Swedish Defence Research Agency FOI, AIAA 2004-4965

[11] PREPHA programme, "control of shock wave boundary layer interaction", S8 CH wind tunnel

2015-04-30

Vortex generator effect on mitigation of separation in transonic flow nozzle

Zare Shahneh, Amir

Institute of Research Engineers and Doctors

A. Zare Shahneh. Vortex generator effect on mitigation of separation in transonic flow nozzle. International Journal of Advancements in Mechanical and Aeronautical Engineering, 2015, Volume 2, Issue 1, pp61-65

https://www.seekdl.org/journals-details.py/98?years=2015&volumes=62&issue=142