Vinoth.M Int. Journal of Engineering Research and Applications www.ijera.com
ISSN : 2248-9622, Vol. 5, Issue 5, (Part -5) May 2015, pp.93-98

RESEARCH ARTICLE OPEN ACCESS

CFD Based Investigation on Effects of Compression Surface At Fighter Aircraft Engine Intake

Vinoth.M*, Maniiarasan.P**
*(Department of Aeronautical Engineering, Anna University, Chennai.)
** (Department of Aeronautical Engineering, Anna University, Chennai.)

ABSTRACT

The purpose of the intake of an aircraft is to supply the engine with a proper airflow during various flight conditions. A good intake design is characterized by providing high pressure recovery and low distortion. Therefore it is essential to divert as much of the boundary layer as possible since it is a factor which affect the quality of the airflow. On aircrafts with engines installed on wing pylons, which is the most common configuration on transport and passenger aircraft, the inlet is short and leads directly to the engine and the pressure recovery is good. For engines that are integrated with the body, for example on fighter aircrafts, the airflow is travelling along the body of the aircraft before it reaches the air intake. A boundary layer builds up along the body, something which is not desirable, especially in the part of the flow that supplies the engines. The pressure recovery is lower because of this, something that has a negative effect upon engine thrust. There are, however, ways to prevent the boundary layer from entering the inlet, or at least to minimize the amount that does. It is common to use a boundary layer diverter. It affects the aircraft performance in so many ways. So, it needs some other provision or technology to overcome intake problem in fighter crafts. In the present work, a well-designed compression surface is installed in the entry of the engine intake to redirect boundary layer and create shock wave for getting desired flow in the compressor (When a flow crosses a shock wave, its velocity got reduced, which in term increases pressure). The compression surface is placed at the entry of the diffuser to perform the above mentioned operation. The work extents to, a comparative investigation is proposed for with and without compression surface. ANSYS-Fluent is a commercial CFD code which will be used for performing the simulation and the simulation configuration contains two different Mach speeds (0.7 & 2) with three different angles of attacks (0°, 7.5° and 15°). The simulation results are evaluated to find out pressure recovery in the engine intake between with and without compression surface.

Keywords – compression surface, pressure recovery, engine intake, boundary layer.

I. INTRODUCTION

The purpose of the intake of an aircraft is to supply the engine with a proper airflow during various flight conditions which it can be subjected to. A good intake design is characterized by providing high pressure recovery and low distortion. Therefore it is essential to divert as much of the boundary layer as possible since it is a factor which affect the quality of the airflow. Pressure recovery is defined as the average total pressure at the engine face, Aerodynamic Interface Plane (AIP) divided by the free stream total pressure (2 0 *PT*). Distortion is a measure of how uniform the total pressure is at the AIP. Factors which reduces the recovery is flow separation, boundary layer ingestion and shock interactions. At high speeds, needs to slow down the flow before it reaches the engine face, favourable around Mach 0.5. For engines that are integrated with the body, for example on fighter aircrafts, the airflow is travelling along the body of the aircraft before it reaches the air intake. A boundary layer builds up along the body, something which is not desirable, especially in the part of the flow that supplies the engines. The pressure recovery is lower because of this, something that has a negative effect upon engine thrust. There are, however, ways to prevent the boundary layer from entering the inlet, or at least to minimize the amount that does. The diverter separates the inlet from the fuselage and the boundary layer, but it is a design feature causing the inlet weight and drag to increase and with higher maintenance requirements.

II. LITERATURE SURVEY

M.Siva prakhasam et. al devoted research to analysis aircraft engine intake. They found that distortion of flow in diffuser due to boundary layer effect leads to compressor vibration and reduced surge margin. The Fluent, commercial computational fluid dynamic software is used to predict flow behaviour. Charles speaks that pressure distortion in intake leads to surging of turbo fan and compressor at same time in turbo jet engine experimentally.

Asmelash Haftu Amaha conducted research over shock wave turbulent boundary layer interaction has been analyzed computationally in a two-dimensional compression ramps for a free stream Mach numbers of 2.85 and 2.94. Ramp angles ranging from 8° to 24° were used to produce the full range of possible flow fields, including flows with no separation, moderate separation, and significant amount of separation. Moderate to significant discrepancies occur in the strong and very strong interactions. The data curves for the computed and experimental pressure ratios (20° and 24°), of strong interactions, show over prediction at separation and under prediction at reattachment locations. The stronger the interaction, the more the numerical solutions deviates from experimental results since, the turbulence model itself is not accurately modelling the flow physics of the problem for stronger interactions. This is the limitation of the turbulence model.

III. DESIGN

Fighter aircraft JAS 39 Gripen designed with Boundary Layer Diverter at engine intake.It is modified without BLD at engine intake and compression surface at engine intake.Both configurations is studied by using CFD simulation and comparison is done.In this work, only diffuser and part of body is considered to study. Flow at diffuser inlet and outlet with and without compression surface is our concern.Therefore, other parts of aircraft are unwanted load to computation and increase computation time and cost.Besides, it can go for finer mesh to maximize accuracy of results.

Dimensions of compression surface is 25 mm * 23 mm * 5mm. Cone angle is 15°.

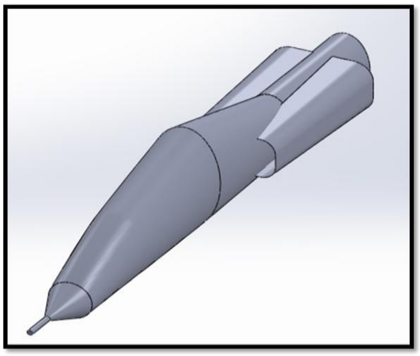
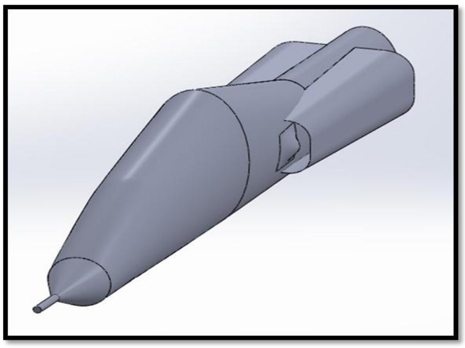


Fig. 1: JAS Gripen 39 Configuration without Compression Surface



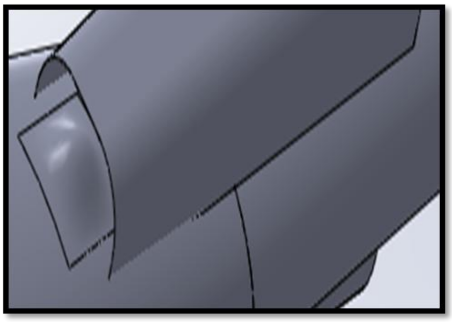


Fig. 2: JAS Gripen 39 Configuration with Compression Surface

IV SIMULATION

Simulation Details

Type of Analysis	– External Flow
Inlet Boundary in Pressure Far Field	- Free Stream Mach number
Atmospheric Pressure	– 18,180 Pa
Temperature	– 216 K
Model	- High Speed Aerodynamics Model
Solver	- CFD

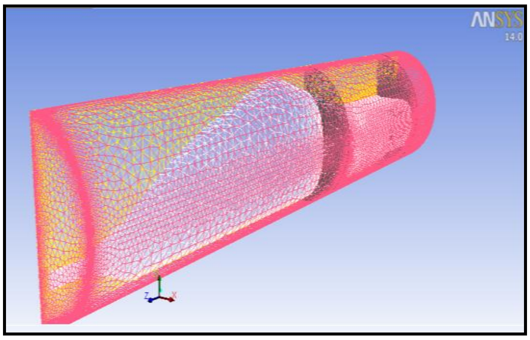


Fig.3: Mesh with Pressure Far Field

a. Elements – 889714
b. Nodes – 171964

Simulation Results

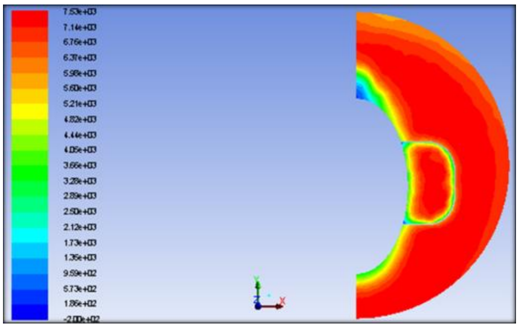


Fig.4: Inlet Total Pressure at 0 AOA and 0.7 Mach without CS

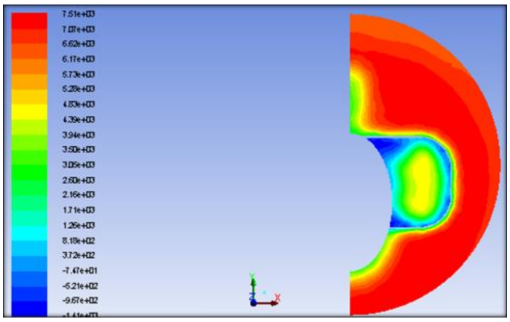


Fig.5: Outlet Total Pressure at 0 AOA and 0.7 Mach without CS

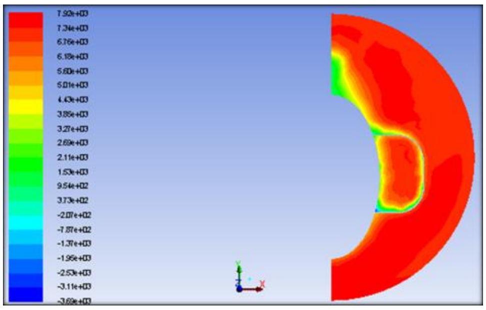


Fig.6: Inlet Total Pressure at 7.5 AOA and 0.7 Mach without CS

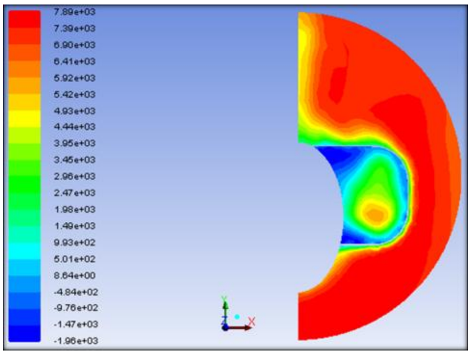


Fig.7: Outlet Total Pressure at 7.5 AOA and 0.7 Mach without CS

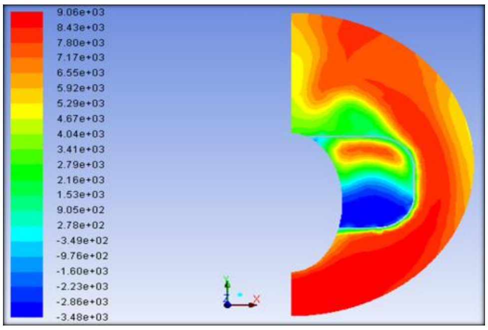


Fig.8: Inlet Total Pressure at 15 AOA and 0.7 Mach without CS

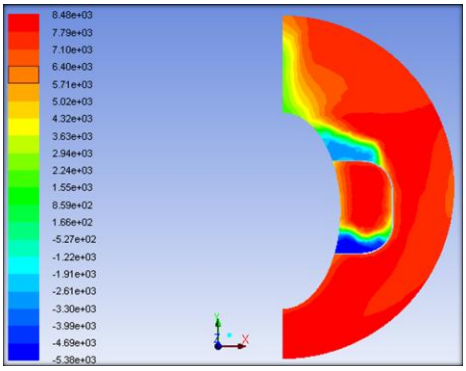


Fig.9: Outlet Total Pressure at 15 AOA and 0.7 Mach without CS

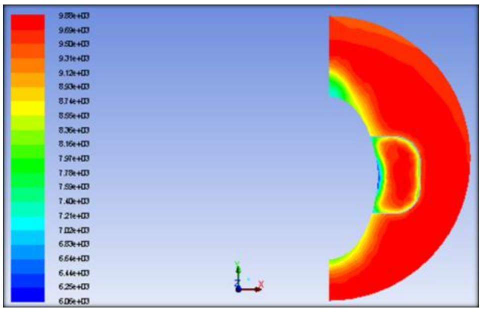


Fig.10: Inlet Total Pressure at 0 AOA and 0.7 Mach with CS

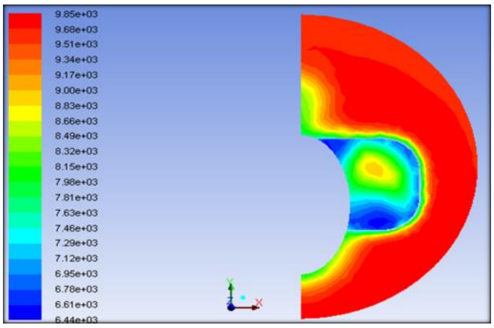


Fig.11: Outlet Total Pressure at 0 AOA and 0.7 Mach with CS

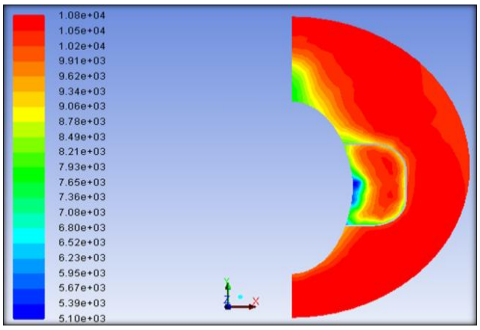


Fig.12: Inlet Total Pressure at 7.5 AOA and 0.7 Mach with CS

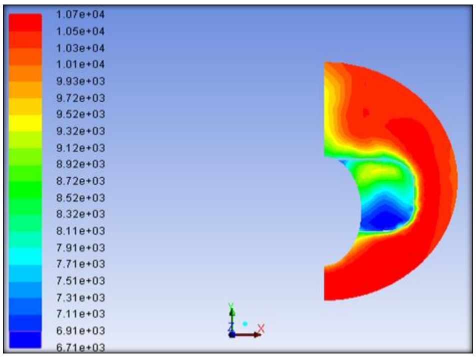


Fig.13: Outlet Total Pressure at 7.5 AOA and 0.7 Mach with CS

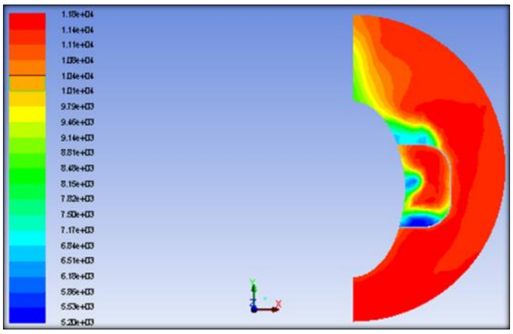


Fig.14: Inlet Total Pressure at 15 AOA and 0.7 Mach with CS

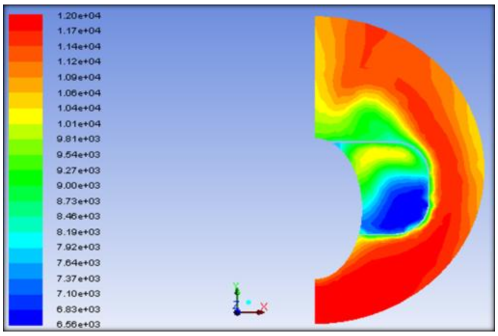


Fig.15: Outlet Total Pressure at 15 AOA and 0.7 Mach with CS

Table1: Pressure Recovery for 0.7 Mach

	AOA 0	AOA 7.5	AOA 15
Without CS	55.31%	53.79%	43.54%
With CS	66.46%	69.69%	64.83%

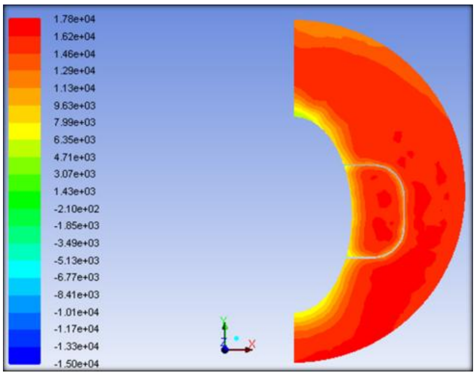


Fig.16: Inlet Total Pressure at 0 AOA and 2 Mach without CS

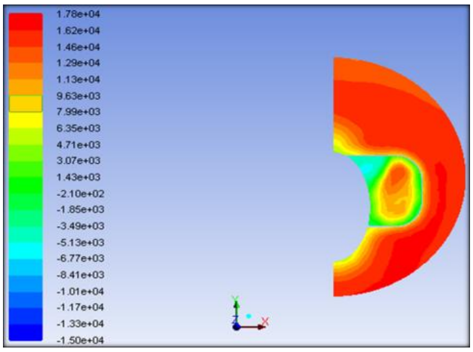


Fig.17: Outlet Total Pressure at 0 AOA and 2 Mach without CS

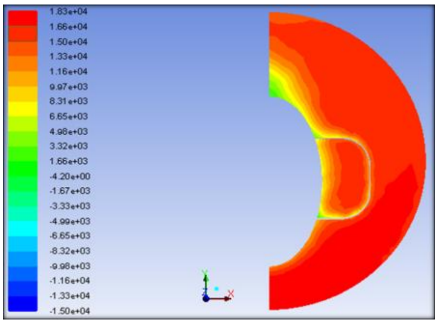


Fig.18: Inlet Total Pressure at 7.5 AOA and 2 Mach without CS

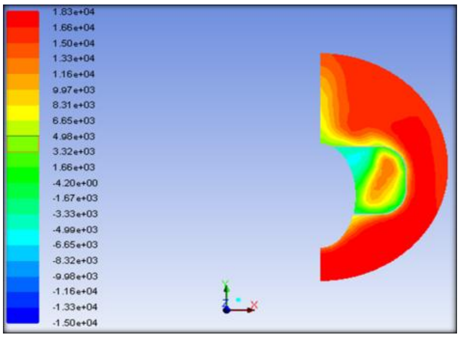


Fig.19: Outlet Total Pressure at 7.5 AOA and 2 Mach without CS

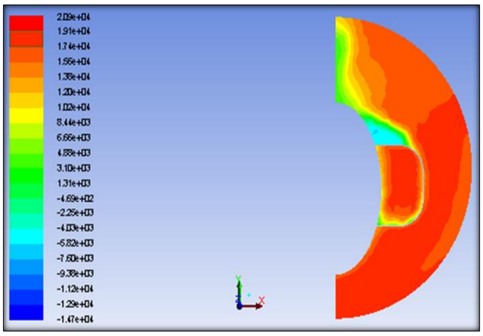


Fig.20: Inlet Total Pressure at 15 AOA and 2 Mach without CS

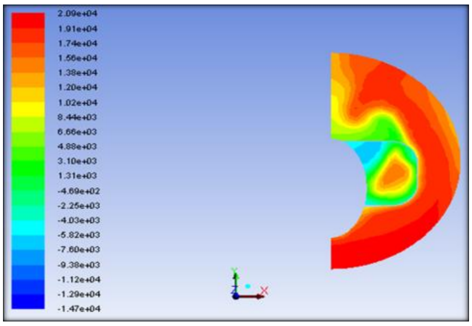


Fig.21: Outlet Total Pressure at 15 AOA and 2 Mach without CS

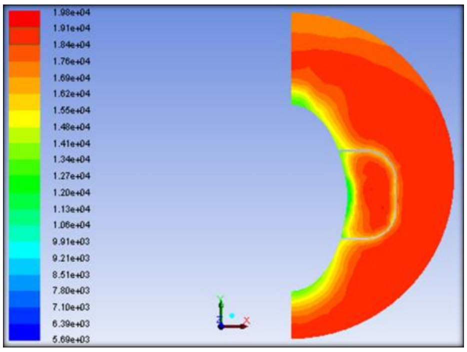


Fig.22: Inlet Total Pressure at 0 AOA and 2 Mach with CS

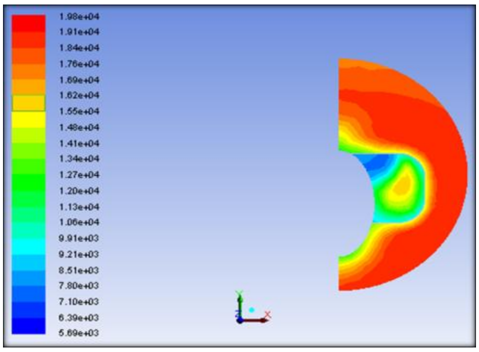


Fig.23: Outlet Total Pressure at 0 AOA and 2 Mach with CS

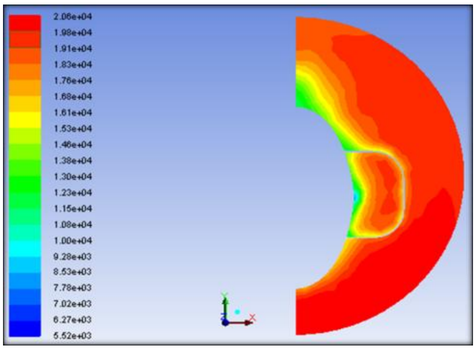


Fig.24: Inlet Total Pressure at 7.5 AOA and 2 Mach with CS

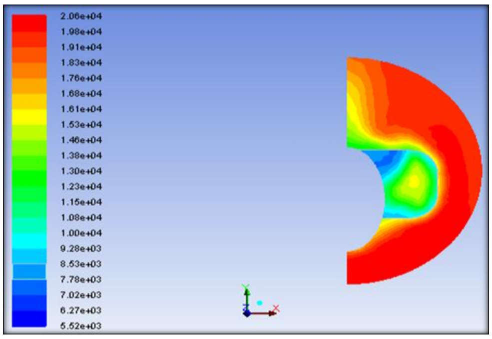


Fig.25: Outlet Total Pressure at 7.5 AOA and 2 Mach with CS

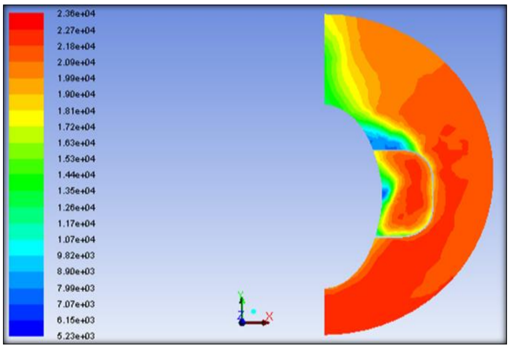


Fig.26: Inlet Total Pressure at 15 AOA and 2 Mach with CS

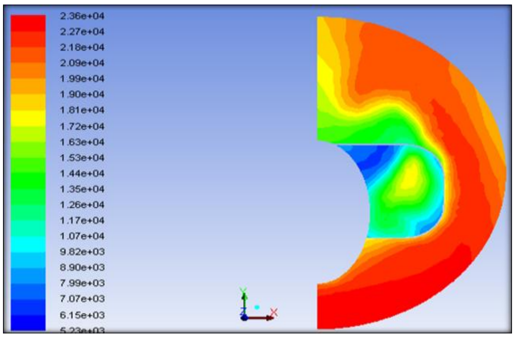


Fig.27: Outlet Total Pressure at 15 AOA and 2 Mach with CS

	AOA 0	AOA 7.5	AOA 15
Without CS	57.31%	71.42%	62.7%
With CS	69.5%	72%	66.15%

Table2: Pressure Recovery for 2 Mach

IV. CONCLUSION

The present work proposes a compression surface that reduces the boundary layer separation around the intake and changes flow properties to desired one. This design has several advantages compared to the diverter. It decreases the inlet weight, since the structure becomes less complex and it has no movable parts therefore requiring less maintenance. This further reduces the cost of the aircraft and is better concerning radar issues. The compression surface has pressure gradients which are span-wise and these help to redirect the boundary layer. It is theoretical model, to understand actual flow behavior; flow simulation is conduct through CFD.

Results are presented in previous chapter. With compression surface configuration has better pressure recovery than without compression surface configuration. Pressure recovery is ratio between total pressures at diffuser inlet to diffuser outlet (at compressor inlet). Here, compression is giving good pressure recovery. Analyzing variation in compression surface height, angle and width is future concern of this project.

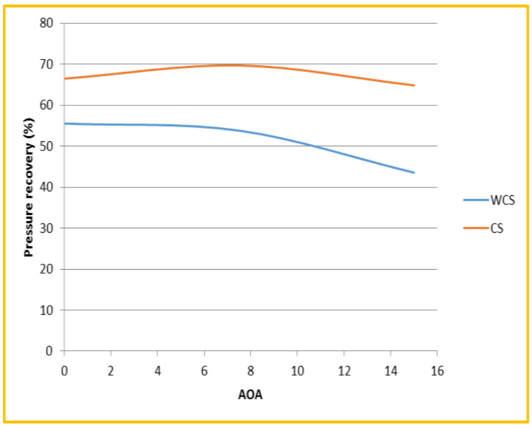


Fig.28: Pressure recovery Vs AOA at 0.7 Mach

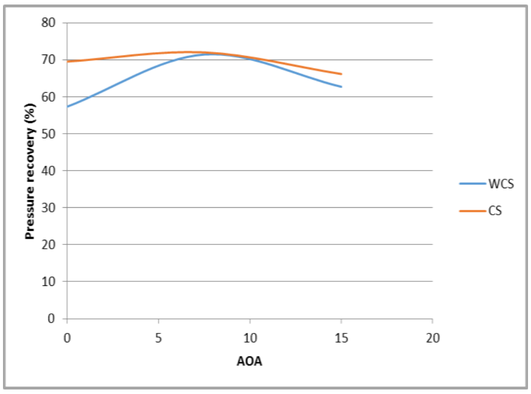


Fig.29: Pressure recovery Vs AOA at 2 Mach

REFERENCES

[1] Sivaprakasam M. et. al (2007) ‘Numerical and Experimental Investigation on Flow Distortion in an Aero Engine Air Intake’, Vol 4, No.1, AAIA.
[2] Amaha A.H. (2014) ‘Numerical Investigation of Shock wave Turbulent Boundary Layer Interaction over a 2D Compression Ramp’, Advances in Aerospace Science and Applications, Volume 4, Number 1, pp. 25-32.
[3] Catalona D.G. et. al (2001) ‘A numerical investigation of supersonic and subsonic flow over axisymmetric body’.