ANSA AND META TOOLS FOR CFD SIMULATION OF SUPERSONIC AND HYPERSONIC WINGS

[1] Kaleeswaran Balasubramaniam*, [2] Shiva Kumar Biradar
[1, 2] Xitadel CAE Technologies India Pvt. Ltd, INDIA,

KEYWORDS –
Supersonic, Hypersonic, Aerodynamics, Heat generation, Stability.

ABSTRACT –

The design of supersonic and hypersonic wings and their significance in Aerospace and Automotive industries has a lot of present and future applications. Supersonic wings and aerofoil designs are already well in use in the present world. Various Computational Fluid Dynamics (CFD) studies have already been conducted analysing various wing designs for future aerospace and automotive applications. But very limited work has been done on the concept of “Swept front wings”. In this study ANSA/mETA environment was used along with Fluent to analyse heat generation of these wings during high velocity regimes. Findings revealed that the swept front wings were capable of attaining more stability at higher altitude than normal swept back wings. The heat resistance capabilities and stability characteristics of these wings were also noted to be efficient. Similarly in hypersonic wing analysis, modified diamond wedge wings were analysed to understand the flow at such Mach regimes and was found to be supportive. The results indicate the feasibility of swept front wings in aerospace industries. The possibility of its inclusion in formula one race is a case for future research.

I. INTRODUCTION

CFD is the science of predicting fluid flows, heat, mass transfer etc. Nowadays CFD data are used extensively for developing new designs, trouble shooting etc. CFD acts as a virtual laboratory to investigate complex cases. Key elements of this Virtual Laboratory are analysis software like Fluent supported by powerful Pre-preprocessor ANSA and Post processor mETA.

Hypersonic studies started in the modern century with the advent use of Re-entry capsules, Satellite launchers, Fighter planes etc. Hypersonic regime starts when the flight speed reaches an air speed of Mach 5.Similarly, a supersonic regime starts from Mach speed of 1.2 and ends up with 5.

A standard wedge aerofoil/wing was analysed to study the hypersonic features. To add further enhancement to the standard model two other wedge foils were designed at different chord positions to have a promising study of hypersonic flows in future of automotive racing industry.

Similarly, in conjunction with hypersonic studies a Supersonic study was also conducted with Swept Front wing model. The main advantage of FSW is the maneuverability. This is because shock waves gets formed at the root chord. Thus, this type of wings produces better reliable conditions to varying climatic changes. The main advantage of FSW is the decrease in drag. This is because in FSW air flows from the wing tip and then it flows towards the body of the model.

The main capabilities of ANSA, mETA are discussed in this paper. ANSA’s CFD and volume algorithms are also discussed, along with specialised features of mETA in plotting, reporting etc.

1.1. Literature Survey

The paper on use of flat plate and concave models for re-entry models [5] tells about new concepts in drag reduction. The use of concave nose reduces stagnation regions and heat transfer [15]. A detail study of concave windward surface was studied. A McCormack's predictor corrector technique was used to study the supersonic flow over a wedge [6].A detail analysis of CFL number with the meshing was analysed. The hypersonic flow conditions were analysed for a circular and hyperbolic probes of capsules [7].A blunt body with cavity was analysed for better drag reduction and decrease in heat transfer was analysed.

A supersonic analysis of Scramjet engine with cowls was analysed [8].The oblique shock relations were studied and pressure distributions were noted. The presence or ramp and cowl in supersonic flow was analysed. A study of supersonic analysis of canards in missiles was analysed [2].The paper deals with the use of concavity in nose cones and similar such models in canard arrangement. An analysis of transonic aerodynamics [10] with oblique shocks and separation regions was studied. W.H.Manson informs about the drag divergence number and pressure coefficient values for high cambered wings. A detail analysis of the use of productive turbulent flows for automotive/aerospace applications was written by J.D.Anderson [4], [11].

1.2. Capabilities of ANSA and mETA used in project design

The meshing of the 2D models was done with ANSA pre-processor. The model was completely meshed with quad elements. The special capability of ANSA to increase uneven nodal counts helped to achieve quad elements throughout the domain. Thus, there was no need of any splitting the domain into rectangular, squared regions. The CFD surface mesh algorithm produced a fine region of elements near to the model for better capturing of the flow. The Tetra Rapid algorithm was used to create volume generation, it's faster and produced good quality volume mesh. The meshing algorithm reduced the time to merely 2 /10 when compared to other pre-processors.

Clear pressure distribution over the FSW model was predicted by using the animation and cutting plane option in mETA. The animation helped to predict the stagnation regions and pressure zones. Similarly, viscous streamlines zone was also predicted. The report making was made simple with drag and drop method and even a video was made with ease using Report generation tool in mETA.

II. DESIGN SPECIFICATIONS AND ISA TABLE

A double wedge aerofoil model was designed and analysed .The total chord length was taken to be as 2 m and with a camber height of 0.15 m. In similar to this two wedge models were designed with wedges protruding at a length of 0.5 m from the front and 0.5 m from the rear side respectively. These models were analysed at hypersonic condition of Mach 5 and at tropospheric and stratospheric conditions.

The values obtained from the ISA tables for the tropospheric and stratospheric conditions as follows
Mach 5 and 10,000 km with a temperature range of 223.15 K, Gauge Pressure range of 26436.3 Pa.
Mach 5 and 20,000 km with a temperature range of 221.65 K, Gauge Pressure range of 2511.023 Pa [1]

Fig.2.1.Double Wedge –design 1

Fig.2.2.Double Wedge-design 2

The Swept front wind design was analysed based on the model details of Grunmann X 29 concept plane model. A total wing span of 5.4 m was considered and a NACA series aerofoil 23042 was scaled to obtain the desired Forward Swept Wing (FSW) model.
ISA table values for a Mach speed of 3 was taken with a Pressure value of 101325 Pa and with a reference temperature of 300 K respectively.

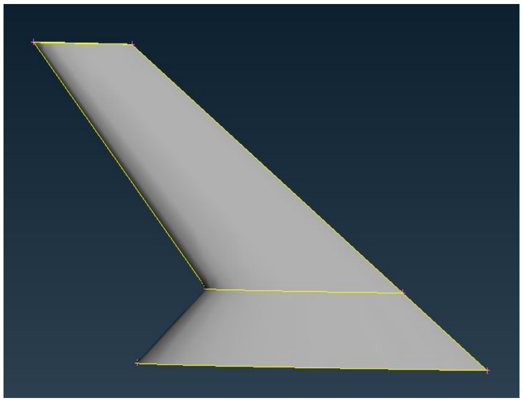

Fig.2.3.Forward Swept Wing

III.NUMERICAL METHODOLOGY

Numerical methodology involves the Pre-Processing, Solver and Post Processing techniques applied to the model for the analysis.

3.1. Mesh Considerations

The Wedge aerofoils were meshed using ANSA © pre-processor [9] and 2D quad elements. A fine mesh with an elemental length of 0.08 was created. A fine quality/refined mesh was used to capture the shock/air flow near to the model. This quadrilateral cell arrangement supports the capture the shock and can reduce the skewness factor.

For a Swept front wing analysis, a surface mesh was generated using CFD algorithm and with an elemental length of 0.5.A Tetra Rapid volume mesh algorithm was considered for this analysis. A prism layer growth was made with a growth factor of 1.2 and with 5 growth layers. Thus, a region of fine mesh around the wing was created for better flow analysis.

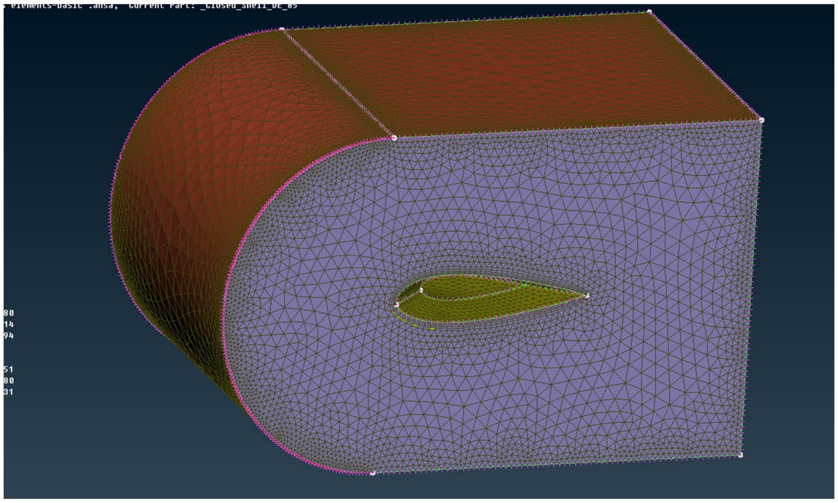

Fig.3.1.Unstructured mesh over FSW model

3.2. Boundary Conditions

The boundary conditions used in the design models includes the following;

S.No	Boundary Conditions	PID Taken (Fluent 2D,Fluent)
1	Pressure Farfield	Wind tunnel edges
2	Iso thermal wall	Model

3.3. Initial Conditions

The mesh files were saved after giving the boundary conditions and then exported as 2D mesh in the case of Wedge models and 3D mesh for FSW model. The mesh case files were studied using Ansys Fluent ©15.0, a commercial FVM coded software [4]. The initial conditions were same for both the models.

A density based solver was chosen for the study. This is because density based solver possess inbuilt criteria of solving high heat and viscous effect problems [2]. Energy equation was taken into consideration (because this high speed sonic study involves high heat and viscous effects).In the viscous models, a Spalart Alamaras one equation model was taken into consideration for Hypersonic studies. A Kω-SST model was used in the case of supersonic cases. The two equation model has got the kinetic energy and heat dissipation rate as two solvable equations. Thus, the problem was solved under Ideal gas conditions. Ideal gas condition values; Cp value 0f 1.006 J/kg-K, with a viscosity of 1.789*10^-05 Kg/m-s and with a thermal conductivity value of 0.0204 W/m-K.

In the boundary condition Pressure farfield with Mach number of 3 for Supersonic and Mach number of 5 was set for hypersonic conditions. A varying gauge pressure of 26436.3 Pa, 2511.023 Pa was applied for varying altitude of 10 Km and 20 Km Hypersonic conditions.

A gauge pressure of 101325 Pa was applied for Supersonic condition (these values were found from the ISA tables).The convergence criteria was set at 10^6 and courant number was kept at 0.1(this determines stability of the solution).The solution was set with highest order of convergence for Navier-stokes solution (i.e. Second order discretization).An Initial iteration was done the for upto 1000 iterations for seeing the degree of convergence. Then, the value was shifted to about 10,000 for complete convergence.

Short summary of above details; 1. Density based solver. 2 Energy equation. 3 Realizable K ω flow model, SA model. 4 Ideal gas flow.

3.4. Equations used

The models were solved with the equations of Continuity; Momentum with turbulent model of Spalart Alamaras was taken into study.In the case of Supersonic analysis Kω-SST model was considered.The models comes under the conservative form of viscous flow as the models were fixed within the flowing velocity field **[Source: Introduction to CFD by J.D.Anderson].**

The given equations suits well to the turbulent cases and the turbulence model used suits well for studying flow around bodies.

1. Spalart Alamaras suits well to predict the flow around hypersonic bodies.
2. Kω –Shear stress models performs well for Supersonic analysis

1. Continuity Equation

$$\frac{d\rho}{dt} + \square.(\rho V) = 0 \qquad (3.1)$$

2. Momentum Equation

X axis:

$$\partial\left(\frac{\rho U}{\partial t}\right) + \nabla.(\rho u V) = -\frac{\partial P}{\partial x} + \frac{\partial \tau xx}{\partial x} + \frac{\partial \tau xy}{\partial y} + \frac{\partial \tau xz}{\partial z} \qquad \textbf{(3.2)}$$

Y axis:

$$\partial\left(\frac{\rho V}{\partial t}\right) + \nabla.(\rho v V) = -\frac{\partial P}{\partial y} + \frac{\partial \tau xy}{\partial x} + \frac{\partial \tau yy}{\partial y} + \frac{\partial \tau yz}{\partial z} \qquad \textbf{(3.3)}$$

Z axis:

$$\partial\left(\frac{\rho W}{\partial t}\right) + \nabla.(\rho w V) = -\frac{\partial P}{\partial z} + \frac{\partial \tau xz}{\partial x} + \frac{\partial \tau zy}{\partial y} + \frac{\partial \tau zz}{\partial z} \qquad \textbf{(3.4)}$$

3. Spalart-Alamaras one equation turbulence model

The Spalart-Alamaras model [14] consists of only one transport equation to represent kinematic eddy viscosity parameter υ and a specification of a length scale providing computations of boundary layers in external aerodynamics. [**Source: CFD by Finite volume approach: Versteeg and Malasekara,13]**

The Reynold's Stresses are computed with
$\tau_{ij} = -\rho u_i u_j = 2\mu_t S_{ij} = \rho\upsilon f_{v1}[\partial U_i/\partial x_j + \partial U_j/\partial x_i]$ **(3.5)**

The transport equation for υ is given by
$\partial(\rho\upsilon)/\partial t + \mathrm{div}(\rho\upsilon U) = 1/\sigma_v \mathrm{div}\ [(\mu+\rho\upsilon)\ \mathrm{grad}(\upsilon) + C_{b2}\rho(\partial\upsilon/\partial_{xk})(\partial\upsilon/\partial_{xk})] + C_{b1}\rho\upsilon\Omega - C_{w1}\rho(\upsilon/ky)^2.f_w$
(3.6)

Rate of change of viscosity) + (Transport of υ by convection) = (Transport of υ by turbulent diffusion) + (Rate of production of υ) – (Rate of dissipation of υ).

Ω= Mean vorticity tensor and values of constants are σ_v=2/3,k=0.4187,C_{b1}= 0.1355,C_{b2}=0.622
$C_{w1} = C_{b1} + k^2 (1+C_{b2}/\sigma\upsilon)$ **(3.7)**

K ω model is a two equation model with two transport equations. This helps to calculate even the effects related to convection and diffusion. K represents the turbulent kinetic energy and ω represents the specific dissipation respectively.

IV.RESULTS AND DISCUSSION

In this section various obtained contour plots of the results were shown and the aerodynamic variations were discussed,

1. Pressure contours. 2. Velocity contours 3. XY plots.

4.1. Pressure Plots

Fig.4.1 depicts about pressure distribution over the design 1 foil analysed using ANSYS Fluent and Post processed using Beta CAE-mETA [12]. There occurs a region of high pressure shock flow near the front part due to stagnation and less region of separation exists at the rear.

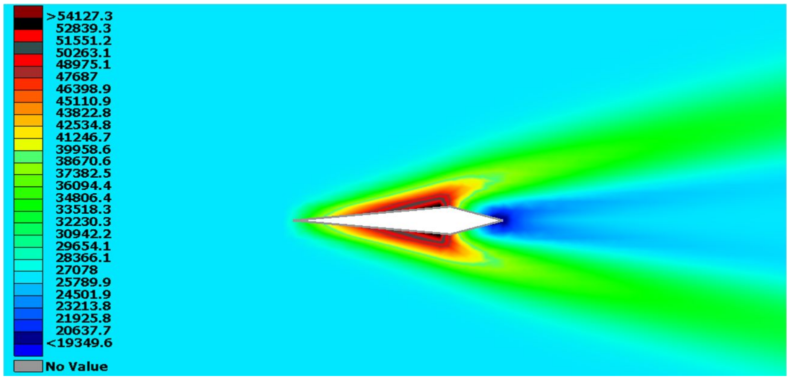

Fig.4.1.Pressure plot over Design 1-Hypersonic foil

Similarly, the standard and design 2 models were also analysed for an altitude range of 10 Km (tropospheric conditions).Fig .4.2 depicts about the pressure distribution over design 2 hypersonic wedge aerofoil analysed at tropospheric conditions. From the figure one can analyse that there occurs more region of separation and less high pressure region. According to Hypersonic studies lesser the reattached region results in ideal model for analysis [3].

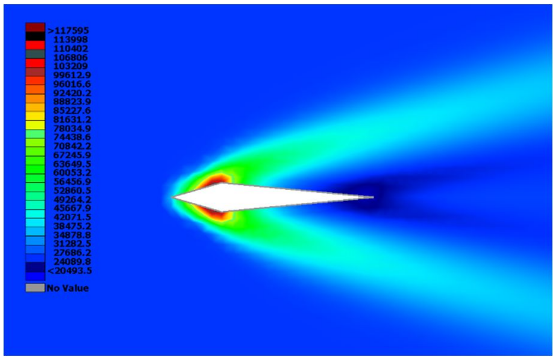

Fig.4.2.Pressure plot over Design 2-Hypersonic foil

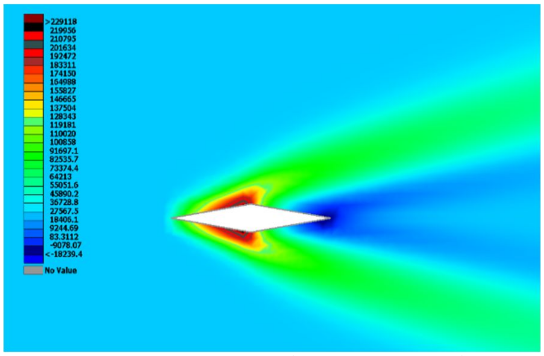

Fig.4.3.Pressure plot over Standard wedge foil

Similarly, pressure distribution over standard model was analysed and found that very high pressure region exists over the stagnation region and high region of separation also exists. The pressure distribution was also analysed with altitude conditions of 20 Km with necessary inputs. From the fig.4.4 the similar design 1 model was analysed with various altitude conditions and pressure contour was obtained. The pressure contours predicted that less region of separation existed in the model and value of stagnation force was also low.

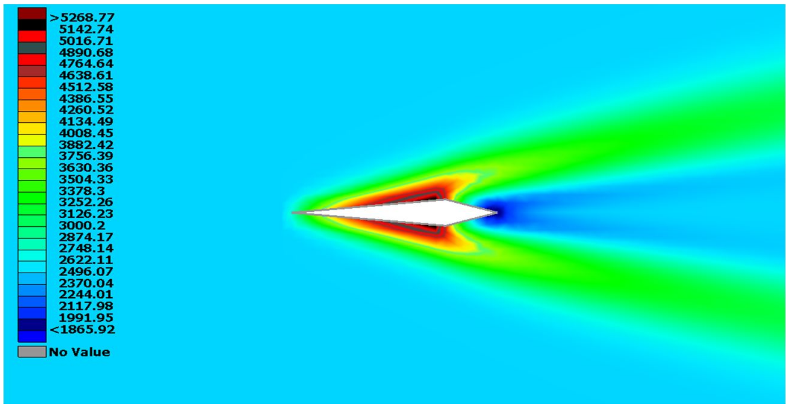

Fig.4.4.Pressure plot over Design 2-Hypersonic foil

Pressure contours of another two models were also analysed and found similar characteristics as discussed with the values of 10 Km range. Thus, design 1 was found to be a good model when compared to other models and could be an ideal case in designing future combat planes. This design models can also be used in automotive industry in transport vehicles in future.

Fig.4.5.Japanese bullet train (courtesy: Daily Mirror)

Fig.4.6.Nissan nismo (Courtesy: Nissan Auto)

The supersonic wing design was analysed at mean sea level conditions and the following observations were made.

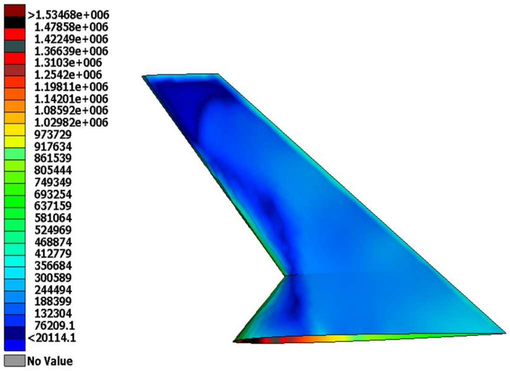

Fig.4.7.Pressure plot over the bottom surface

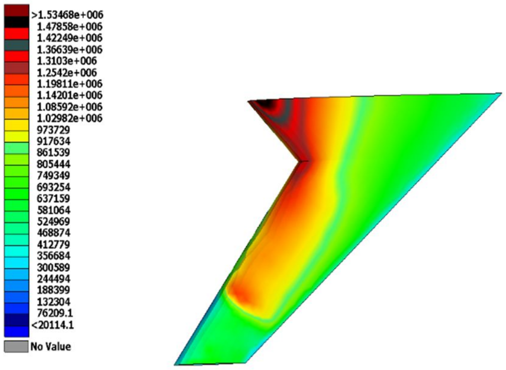

Fig.4.8.Pressure plot over the top surface

The figures 4.7, 4.8 predicts reduced drag after the region of stagnation due to high recirculation of the flow. The Mach flow distribution of the models was studied to find out the position of the shock region and the region of stagnation conditions. This is because increase in Mach causes more drag flow over the object.

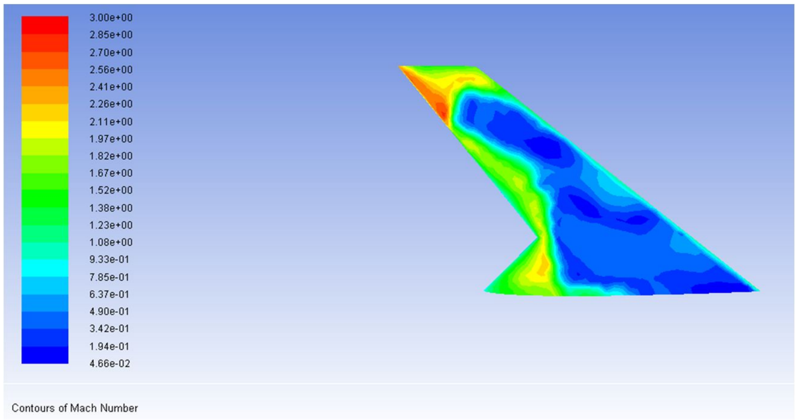

Fig.4.9.Mach Contour over the FSW

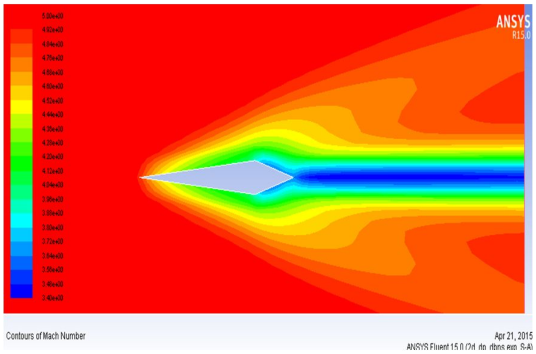

Fig.4.10.Mach Contour over the Design 1 model

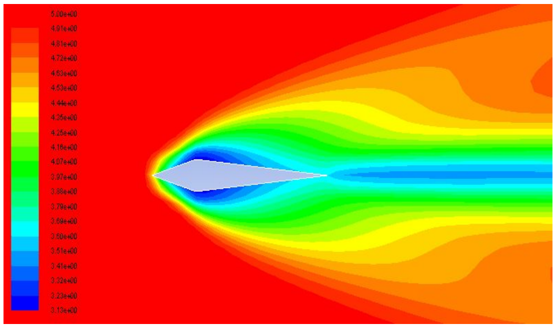

Fig.4.11.Mach Contour over the Design 2

From the fig 4.10 and 4.11 Mach flow over the models and the region of stagnation, separation was well noted. Mach flow analysis of the standard double wedge revealed high region of shock stagnation near to the camber tips. A similar flow profile was also found in the Mach flow with the stratospheric region values.

The Cd values of the models were analysed and the values of high Cd and its correlating models were understood. The fig 4.12 predicts the Cd values of design 1, the Cd reduces drastically and models suits perfectly for high speed flows.

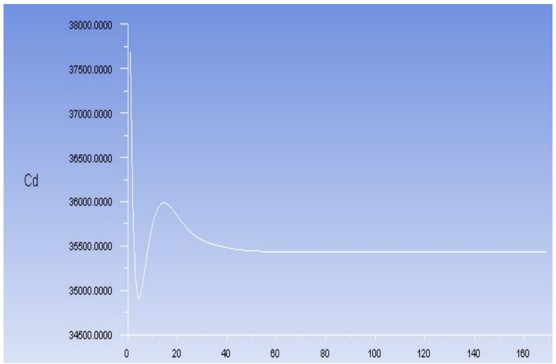

Fig.4.12.Cd values-design 1

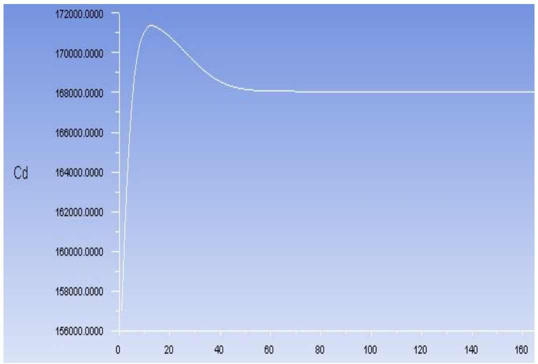

Fig.4.11.Cd values –diamond wedge

The Cd values of standard diamond wedge and design 2 was also analysed. It was found that Cd values of diamond wedge soars to high values when compared to design 2, 1 models.

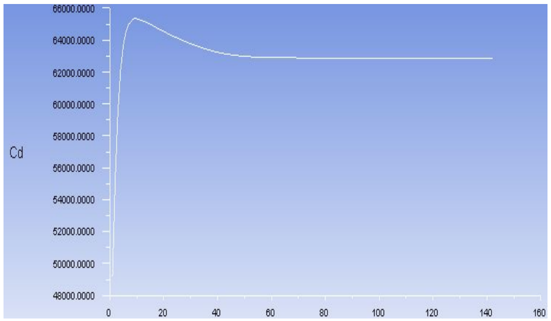

Fig.4.12.Cd values-design 2

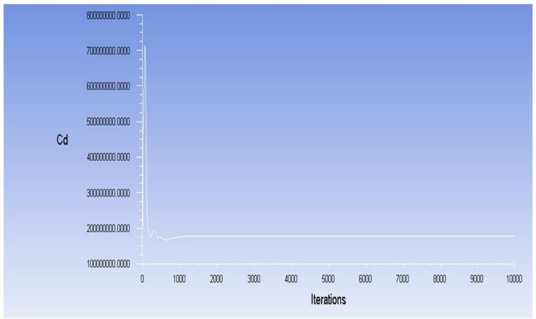

Fig.4.13.Cd values-FSW model

From the CD graphs it was analysed that coefficient of drag is low for design 1 model when compared to standard and design 2 models. This is based on the region or the percentage of attachment in the models. As discussed model 1 has less CD due to less region of attachment when compared to other models. Similarly, fig 4.13 of FSW model also is an ideal case study for lesser drag formation in Supersonic analysis.

VII. CONCLUSION

Mach number in the range of 1 to 3 constitutes the region of supersonic flows. Body under a supersonic regime experiences shock waves. Normal or an oblique shock may pertain based on the profile. The supersonic model of FSW was analysed and stagnation regions were found. The model correlates with the theoretical datas of high stagnation near to root chord and thus leading to stability conditions. This supersonic design can also be considered for formula one cars to create a stable spoilers at rear and front. The modified wedge design makes a conclusion that it creates less region of separation, making a good high streamlined flows. These designs also helps to reduce the percentage region of high Mach stagnation regions. The results from the pressure contours and Mach distribution also invokes the same, further Cd values were also analysed to note down the drag reduction in the design models.

The highest stagnation pressure value for design 1 model was found to be 54127 Pa when compared to standard model value of 229118 Pa and design 2 model with a value of 117595 Pa. Stratospheric conditions of the same designs were also analysed and found correlative results of model's behaviour with respect to tropospheric conditions.

The better use of numbering technique in ANSA helped to create quad meshes, which would have been a time consuming process in other Pre-Processing (as it requires splitting the domain). ANSA's error fixing methods and volume generation algorithms made the problem easier to solve for simulation setup. Further, ANSA's Fluent 2D deck setup helped to implement the boundary conditions easily on the edges. Post-Processing techniques from mETA paved for a better understanding of the flow around the models. Viscous flows were predicted using scalar option and by using cutting plane technique. Better animation and drag, drop techniques from mETA was an added advantage for flow analysis in this paper. Thus, the overall use of ANSA and mETA tools not only helped to solve the problem with ease, but also with less time consumption.

List of Abbreviations

Cd, CD	- Coefficient of Drag
FSW	- Forward Swept Wing
Mach	- Mach number (1 Mach = 340.12 m/s)
Pa	- Pascal
SST	-Shear Stress Transport model
FVM	- Finite Volume Method
ISA	-International Standard Atmosphere
NACA	-National Advisory Committee for Aerofoils

Appendix

Fortran program

```
programtroposphericcalculations
!to calculate the pressure below and above the troposphere
real p0,h,t0,p real p11,g,h11,r,t11
print*,'enter the value of initial pressure p0'
read*,p0
print*,'enter the value of initial temperature t0'
read*,t0
print*,'enter the value of specified height h'
read*,h
h11=11  ! troposphere starts from this height
p11=226.32  ! pressure above the tropospheric layer
t11=216.52  ! temperature above the tropospheric layer
r=287 !gas constant
call below(p0,h,t0,p)
call above(p1,p11,g,h,h11,r,t11)
end program troposphericcalculations

subroutinebelow (p0,h,t0,p)
p=p0(1-0.0065*h/t0)**(52561)
print*,'the value of pressure below the troposphere is::',p
end subroutine
subroutine above(p1,p11,g,h,h11,r,t11)
p1=p11*exp(-(g)*(h-h11)/(r*t11))
print*,'the value of pressure above the troposphere is::',p1
end subroutine
```

Acknowledgement

We would like to express our deep acknowledgement to Mr. Prakash ‘Krish’ Krishnaswamy, CEO Xitadel group for his constant support. Also, our sincere thanks to Mr. Nikolaos Christodoulou Beta CAE, Mrs. Efthymia Chatzivasiloglou Beta CAE, Dr.Malky, Dept. of Aerodynamics, University of Leicester, staff members of Xitadel India Ltd and Beta CAE S.A.

REFERENCES

(1) Mustafa Cavcar, Anadolu University, Turkey. (2009). The International Standard Atmosphere. Pages 1-12.

(2) Kowtham Kumar, Kaleeswaran, Ranjith. (2013). CFD analysis of notched canard arrangement in missiles. IJERD. , Pages. 103-111.

(3) J.D.Anderson (2013). Hypersonic and high-temperature gas dynamics. 2nd ed. USA: AIAA education. 24-58.

(4) Fluent version 15.0 User’s Guide, ANSYS Inc. March 2015

(5) Ugur Guven. (2012). Effect of Eccentricity on heat transfer rate of reentry vehicle with concave windward surface. . International Astronautical Congress. , Pages 1-6.

(6) Saif Ullah et.al. (2009). Modeling and Simulation of Supersonic flow using Mccormack's Technique. Proceedings of the world congress., Pages1-5.

(7) Sourabh Bhat et.al. (2012). Effect of Nose cavity on heat transfer rates to surface of an aeroshell . International Astronautical Congress., Pages 1-6.

(8) Manivannan et al. (2013). CFD analysis of Scramjet. International Journal of Innovative Research., Pages 2730-2734.

(9) ANSA version 15.2.X User’s Guide, BETA CAE Systems S.A., March 2015.

(10) W.H.Manson. (2006). Transonic Aerodynamics of Airfoils and Wings.Configuration Aerodynamics. USA: AIAA Education.

(11) J.D.Anderson. (2006). nil. In: nil Introduction to Computational Fluid Dynamics. 5th ed. USA: Tata Mcgraw hill.

(12) μETA Postprocessor version 15.2.X. User’s Guide, BETA CAE Systems S.A., March 2015.

(13) H.Versteeg, W.Malalasekra (2009). An Introduction to Computational Fluid Dynamics: The Finite Volume Method. 2nd ed. New Delhi: Pearson Education Ltd.

(14) Mikhail Pavlovich Bulat and Pavel Victorovich Bulat. (2013). Comparison of Turbulence Models in the Calculation of Supersonic Separated Flow. World applied science journal., 1263-1267.

(15) Maharshi Bharadwaaja, English version by Subbaraya Shastry,G.R.Josyer. (1973).Vymaanika-shaastra book of aeronautics. Mysore: Coronation press.