

See discussions, stats, and author profiles for this publication at: <https://www.researchgate.net/publication/283714400>

# Comparative Study of varying Mach No. over Symmetric Airfoil with Flow through various Channel Designs

Conference Paper · August 2014

CITATIONS

0

READS

129

5 authors, including:



**Harshit Bhatia**

University of Udine

4 PUBLICATIONS 0 CITATIONS

SEE PROFILE



**Geetesh Waghela**

ARK Info Slutions Pvt. Ltd.

3 PUBLICATIONS 0 CITATIONS

SEE PROFILE



**Mohd Afeef Badri**

Atomic Energy and Alternative Energies Commission

15 PUBLICATIONS 21 CITATIONS

SEE PROFILE



**Rajesh Yadav**

University of Petroleum & Energy Studies

36 PUBLICATIONS 54 CITATIONS

SEE PROFILE

Some of the authors of this publication are also working on these related projects:



Inertial and Non inertial particles in turbulent flows [View project](#)



Aerothermodynamics of a Spiked Blunt Body with Multiple Aerodisks [View project](#)

## Comparative Study of varying Mach No. over Symmetric Airfoil with Flow through various Channel Designs

Harshit Bhatia<sup>1#</sup>, Geetesh Waghela<sup>1</sup>, Mohd. Afeef Badri<sup>1</sup>, Rajesh Yadav<sup>1</sup>, Prakash S Kulkarni<sup>2</sup>

<sup>1</sup> Department of Aerospace, UPES Dehradun, [hb.aero1803@gmail.com](mailto:hb.aero1803@gmail.com)

<sup>2</sup> Department of Aerospace, Indian Institute of Sciences, Bangalore, [psk@aero.iisc.ernet.in](mailto:psk@aero.iisc.ernet.in)

### ABSTRACT

This paper summarizes the study of aerodynamic parameters keeping aero-thermodynamic heating as prime focus. When an aircraft transits from subsonic to transonic speeds, at certain locations over the wing sonic flow may be encountered, this affects the lift and drag significantly, and also results in high heating. Thin symmetrical airfoil with internal duct was chosen to study the effects of duct inclusion in the airfoil. Such airfoils also possess leading edge curvature acting as blunt body for the airfoil with a bypass path through cross section. Introducing a ducted airfoil in supersonic flight enhances air circulation, there forth reducing aerodynamic heating leading to enhanced aerodynamic performance. Airfoils with two different variations i.e constant area ducted airfoil and slightly diverging ducted area airfoil formed the test cases for the study. Two dimensional steady state solutions are obtained, and cases observed with complete calculations at a free stream Mach number, pressure and temperature of 0.6, 1 Bar and 300 K respectively are at angle of attack 0 degree and 6 degree so as to observe more details in lift generating conditions.

**Keywords:** Symmetric Airfoil Design, Channeled airfoil, Flow through duct, aero-thermodynamics.

### Nomenclature:

M or $\bar{M}$	=	Mach number
AOA	=	Angle of Attack.
L	=	Lift
D	=	Drag
L/D	=	Lift to drag ratio
$\rho$	=	Density
c	=	Chord
$\Gamma$	=	Lift per unit span
$C_l$	=	Coefficient of Lift

**Introduction:** The concept of flow through duct (ducted airfoils) has already been tested over airfoils but those studies were mainly focused on supersonic drag reduction [1]. Tests revealed that introduction of duct reduced drag over airfoil thereby reducing the heating effect [2]. Supersonic aircraft which were first developed in the second half of the 20<sup>th</sup> century have been entirely used for research or military purpose [1]. There are only two such aircrafts (Concord and Tupolev TU-114) which were developed for civil transport use. Ruffin and Marshall investigated the effect of drag reduction by the concept of channeled airfoil at free stream Mach number 2.5 [2]. The concept of channeled airfoil can thus be applied to various vehicles with high speed aerodynamic usage, like supersonic aircrafts, atmospheric reentry vehicles, and missiles; but on larger scale it can be even used to increase the lift over simple airfoils at subsonic speed [2]. Control of aerodynamic heating also becomes a study of importance as controlling it involved in safety parameters of aviation industry [1,2]. One of the aerodynamic factors of all these vehicles that add to cost in terms of economical flight is Drag. Drag here is a sensitive issue and many of the conceptual designs in past have been rejected due to this drag factor. This drag adds on to the mass payload effect, additional force and power, and less maneuverability. Therefore, it is the base for economic feasibility also, while finalizing the design.

No vehicle as such can directly reach supersonic speed without going through subsonic speed. As we look at the flight region of subsonic and supersonic separately; the best known Diamond shaped airfoil is known, best to perform for supersonic cruise, have very low lifting ability at subsonic speed. To counter this if we use cambered airfoil to increase lift in subsonic region, it often leads to lower L/D ratio because of high wave drag influence (as wave drag hinder the cruise performance). If we try to improve L/D ratio of chambered airfoil then we have to compromise with the design cost, as we require extra lift enhancing devices. With sharp edged diamond airfoils, as

used in many previous studies, problems that are related, are not limited to, structural strength, cost involved in difficult manufacturing. For structural stability some blunting is preferred. This blunting has effect on heat transfer rate as well, resulting in less body temperature and more heat dissipation to atmosphere. This helps engineers to maintain low temperature of aerodynamic body.

Main factors of aero-thermodynamic heating are skin friction drag and compression of air at higher Mach number which may lead to formation of shock wave [1]. This phenomenon causes the temperature to increase as the kinetic energy near walls becomes close to zero where we obtain our stagnation point. It comes into concern in mainly supersonic and hypersonic flows where stagnation point is of great importance as kinetic energy drops significantly.

**Solution Methodology:** This problem discusses the effect of Mach No. variation over channeled airfoil in different flow regimes. It goes on to show that ducted airfoils have enhanced aerodynamic performance than supersonic airfoils without ducts. There are many such topics of concern such as selection of thin symmetrical subsonic airfoil that yields better results than supersonic diamond shaped airfoils so NACA 0010 has been selected as base airfoil and channeling effect has been applied on it.

**Governing Equations-** The equations of concerns (but are not limited to) that affect this kind of problem related to airfoil are described in short below

1) Bernoulli's Principle: It says that whenever there exist a pressure difference there has to be difference in the speed around body. It also says that for any body or object to fly there has to be a pressure difference. However, it fails to explain why there is velocity difference around airfoil.

With this discussion, we can say that following conservation laws hold for air flow around airfoil.

2) Conservation of Mass:- This states that mass flow rate of air is conserved around airfoil. Also it assumes that our airfoil is solid and air cannot pass through it (no air leakage).

3) Conservation of Momentum: This is based on the Newton's second Law, which gives direct relation between net force acting on air element and rate of momentum change.

4) Conservation of energy- Energy is conserved in our approach as we are using already validated code.

Other theories explaining the concept are

5) Kutta Joukowski Theorem: If lift coefficient at particular angle of attack is known then the lift around it is calculated by the formula 1.2

$$L = \frac{1}{2} \rho v^2 A C_L$$

————— Eq 1.2

Where L is lift,  $\rho$  is density, A is area,  $C_L$  is coefficient of lift at that particular angle of attack.

This theorem says that lift over airfoil is obtained directly by imposing circulation on it based on potential flow theory. This convinces other scientists as it satisfies thin airfoil theory and lifting line theory.

The circulation  $\Gamma$  is closed integral of tangential velocity on airfoil surface. The Lift per unit span is given by

$$\Gamma' = \rho v \Gamma$$

----- Eq 1.3

Lift circulation by pressure integration

$$L = \oint p n \cdot k \, dA$$

————— Eq 1.4

**Computational Setup-** Domain selection was done based on already tested ideas, so medium sized domain was selected which is ten times the cord length (10c). This selection can also be justified on the limited availability of the computational resource. Application of grid independence study on domain was performed to get a solution independent of mesh size, taken to solve the problem. Various meshes of different cell counts were tested and optimum size with independent effect was used in this problem.

Structured meshing strategy has been incorporated to yield better and faster results. Study of temperature variation is performed numerically by using Navier-Stokes solver. Other aspects of this study are to study change in Aerodynamic and Aero-thermodynamic characteristics of the airfoil when the duct inside is changed from straight one to diverging. As we need to study the temperature variation in the duct region and compare it with subsonic airfoil without duct, we need to enable energy equation that will bring temperature into consideration.

As no validation for low subsonic airfoil is available, so a case of supersonic airfoil for the purpose of validating the solver and boundary conditions is used. NACA 66206 airfoil, which is supersonic diamond shaped airfoil, is selected as the validating case airfoil. Analysis of this particular airfoil at Mach number 2.5, at an altitude of 35000 ft. is performed. All the parameters were calculated at same atmospheric conditions and were used in solver boundary conditions.

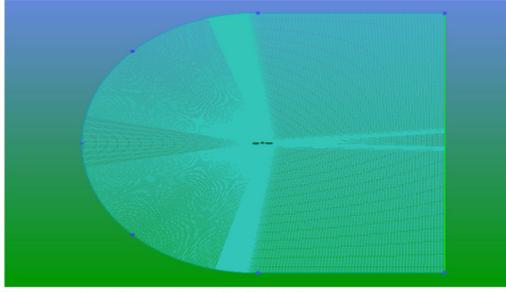


Figure 1a- Showing C Grid Domain used in grid.

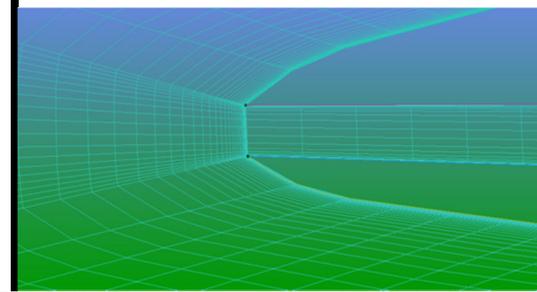


Figure 1b- Showing concentrated mesh in duct.

C- Grid with structured meshing strategy is selected as appropriate domain for this problem. Domain size has been taken as 10 times of chord in all the directions to ensure that boundary effects don't come into considerations. Shocks will occur when flow goes supersonic or even when it crosses transonic region so we need a turbulence model to capture the shock and the vortices. Appropriate turbulence model has been taken as  $K-\omega$  SST which perfectly captures shock near airfoil boundary and other flow properties away from the walls. Advantage in using  $K-\omega$  SST is that it is a two equation eddy viscosity turbulence model and can perfectly capture boundary effects as it converts into  $K-\omega$  near boundary and can be applied to viscous sub layers. It transforms itself into  $K-\epsilon$  model far from boundary which makes it easier and makes the  $K-\omega$  model free from errors like too sensitive to inlet boundary conditions. This model is suitable for flows with adverse pressure gradients where flow properties change dramatically with small distance. Second order discretization was performed to get more accurate solution. Density based solver has been used as flow is compressible and density changes. For achieving better performance, solution steering was done to make the solution stable by changing courant number as iteration progresses.

## Results and Discussion:

The circular path that air follows while airfoil is airborne increases the circulation over airfoil and also Also high pressure air around the leading edge of airfoil passes through airfoil center via the duct to the trailing part of airfoil, Which results in the decreases of pressure drag there by resulting in high lift-to-drag ratio for the airfoil.

Results in the form of Coefficient of Drag are shown below and it clearly indicates that as we increase duct thickness from 0% to 8%, Cd tends to reduce as the high energy air leaves the foil from duct hence resulting in reduction of temperature.

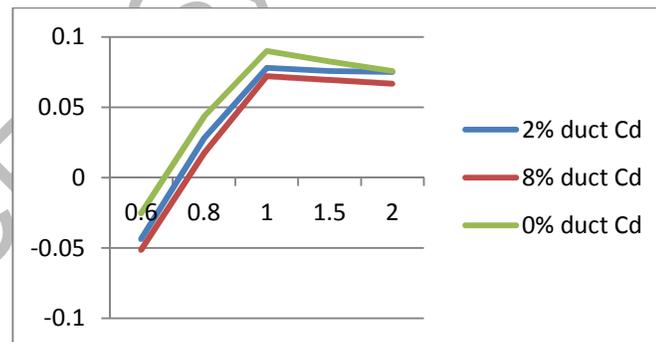


Fig (2) Cd vs Mach number

As we can see, as we increase Mach number from 0.6 to 2, coefficient of drag tends to increase but increase in duct thickness results in lower drag comparatively with other geometries. We may conclude that increase in duct thickness reduces the drag. Considerable drag reduction of over 11.71% has been noted down in case of 8% ducted airfoil at Mach 2.0 and maximum reduction of 13.42% has been observed in 2% ducted airfoil at Mach 1. Although there is not a significant increase in coefficient of lift values for ducted airfoil for zero degree as well as at lower angle of attack.

This analysis provides us with the preliminary view of the overall concept as described below- We get contours of static temperature and Mach numbers which describes visual information of how the quantities are changing for temperature and shock waves respectively.

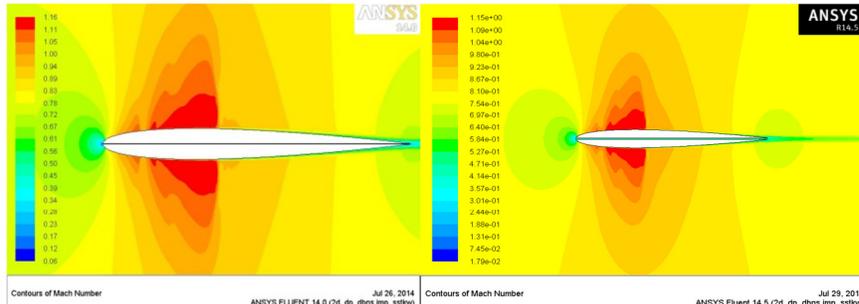


Fig (3a,b) Contours of Mach No.  $M=0.8$  (0AOA)      Contours of Mach No.  $M=0.8$  (0AOA)

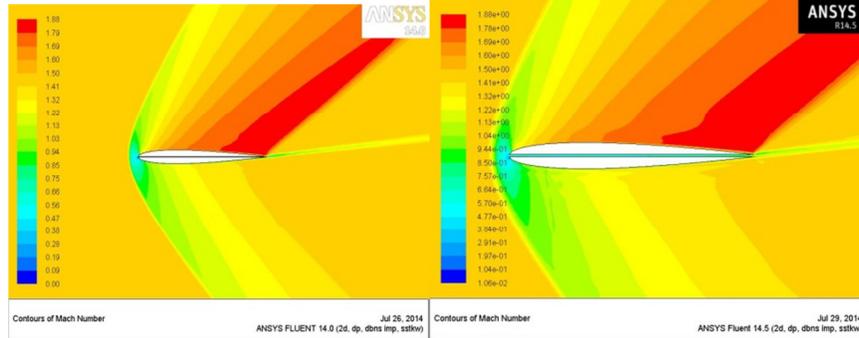


Fig (3c,d) Contours of Mach No.  $M=1.5$  (6AOA)      Contours of Mach No.  $M=1.5$  (6AOA)

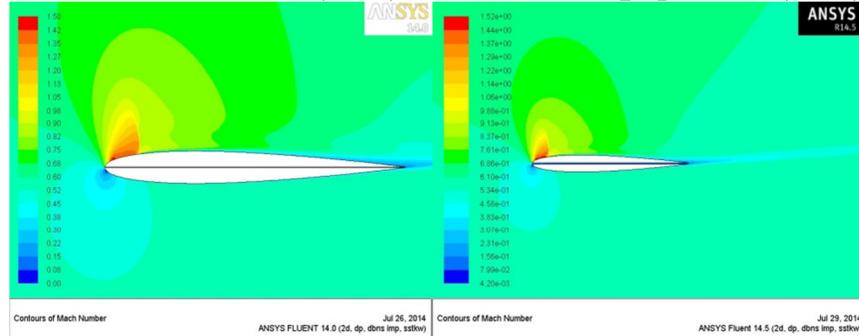


Fig (3e,f) Contours of Mach No.  $M=0.6$  (6AOA)      Contours of Mach No.  $M=0.6$  (6AOA)

From the figures shown above, we can see that how Mach number varies with changing duct thickness, it does not change rapidly with small increase in duct thickness. let us examine variation of temperature with increase in duct thickness.

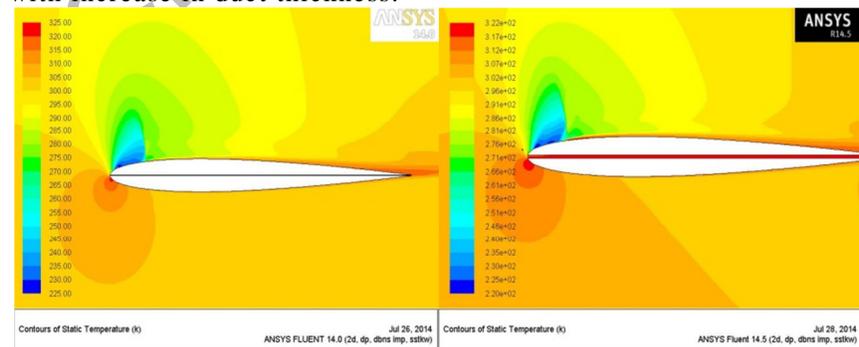


Fig (4 a,b) Contours of temperature at  $M=0.6$  (6AOA) 2% duct thickness and 8% duct thickness

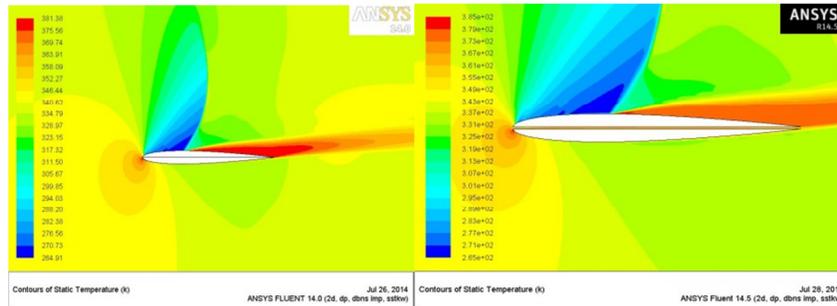


Fig (4 c,d) Contours of temperature at M=0.8 (6AOA) 2% and 8% duct thickness

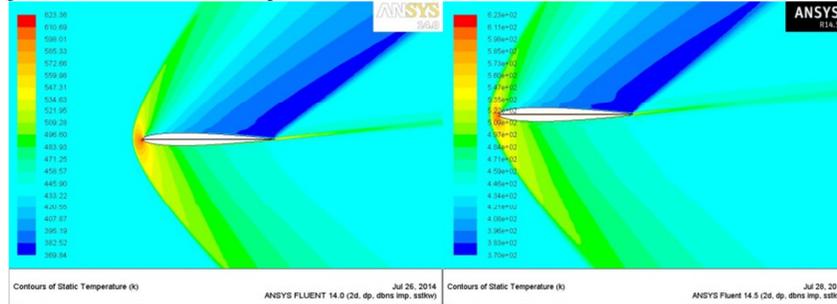
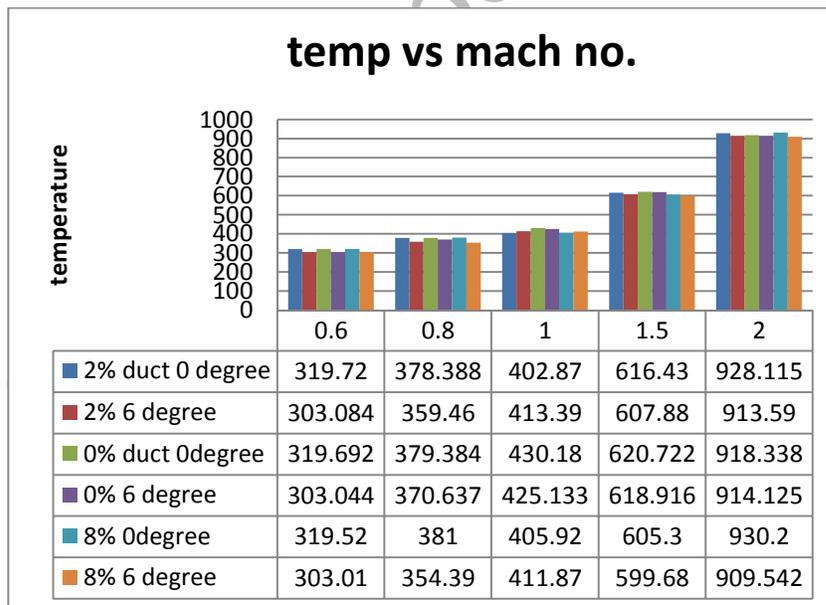


Fig (4, e,f) Contours of temperature at M=1.5(6AOA) 2% and 8% duct thickness.

We can clearly see from above contours that temperature at stagnation point decreases as the high energy air passes through duct. As we increase the duct thickness, the temperature reduces. A comparative graph has been shown below which shows the details.



Figure(5) temperature vs Mach Number

It can be noted down that by introducing a small duct in airfoil there is a significant decrease in temperature at the stagnation point which thereby decreases the drag force acting on the airfoil. A total of 15-20k reduction can be noted down by introducing duct.

Skin friction coefficient on all the three cases are found out to be changing with change in duct thickness. This can be observed from below figures. Another idea to plot the skin friction function vs location is that it gives us the idea of location of stagnation point.

Fig(6) Skin friction Coefficient at lower airfoil surface for (a) No duct at 0.8 M and 6 degree AOA; (b) 2 percent duct and; (c) 8 percent.

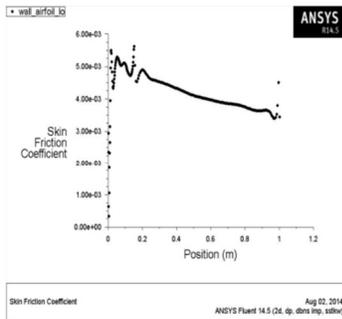
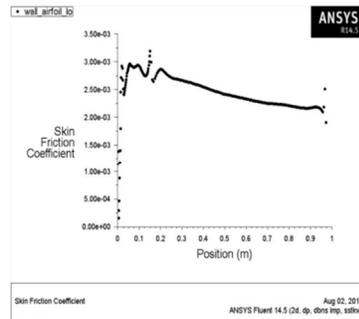
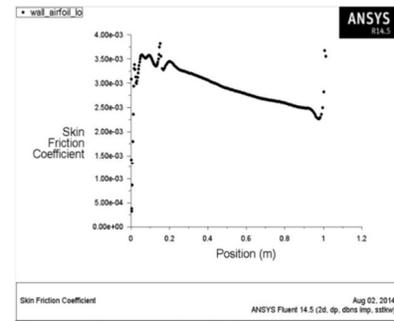


Fig (6a)



Fig(6b)



Fig(6c)

## Conclusion:

The main focus has been given to drag reduction on the subsonic airfoil with introducing duct so that it can be used in supersonic conditions as well and get optimal results. Although there is no significant lift increase with this improvement but it can be noted down that a total of 11.71% drag reduction is obtained by introducing 2% thickness duct and a total of 13% drag reduction can be obtained by introducing 8% thickness duct in airfoil.

When high energy air passes through duct, it results in decrease in temperature at stagnation point and at duct inlet. A maximum reduction of 20 K can be noted down in temperature at stagnation point.

Further work is possible on analyzing airfoil for more conditions and different duct geometries to obtain higher drag reduction and generate more lift. Focus will be given to use same airfoil for subsonic and supersonic conditions to make aircraft multi-functional.

## References-

- [1] David M. Giles and David D. Marshall; Aerodynamic Performance Enhancement of a NACA 66-206 Airfoil Using Supersonic Channel Airfoil Design; Paper No. AIAA 2008-300; 46th AIAA Aerospace Sciences Meeting and Exhibit 7 - 10 January 2008, Reno, Nevada
- [2] Stephen M. Ruffin, Anurag Gupta, and David Marshall; Supersonic Channel Airfoils for Reduced Drag; AIAA JOURNAL Vol.38, No. 3, March 2000
- [3] Anderson, J.D. Jr., Introduction to CFD
- [4] VERSTEEG, and MALALSEKERA, AN INTRODUCTION TO CFD, THE FVM Approach.