7. ANKARA INTERNATIONAL AEROSPACE CONFERENCE
11-13 September 2013 - METU, Ankara TURKEY

AIAC-2013-049

A 2-D THEORETICAL AND COMPUTATIONAL STUDY FOR PRELIMINARY DETERMINATION OF GRID FIN GEOMETRIC PARAMETERS AT LOW TRANSONIC SPEEDS

Erdem Dikbaş[1] and Özgür Uğraş Baran[2]
TÜBİTAK-SAGE
Ankara, Turkey

Cüneyt Sert[3]
Middle East Technical University
Ankara, Turkey

ABSTRACT

This paper describes a two dimensional computational study of flow field characterization of grid fins using FLUENT solver for low transonic flight conditions. The main objective of the study is to understand the effects of geometric parameters on the drag and flow field characteristics of a grid fin and to ease determining geometric parameters in terms of requirements. Width-to-thickness and width-to-depth ratios are the ones that are to be determined. Required width-to-thickness ratio is driven by the area rule of the gas dynamics, while major property that governs required width-to-depth ratio is the boundary layer profile. Results are to show that prediction of geometric parameters by theoretical relations is possible.

INTRODUCTION

Grid fin concept for aerodynamic control of missiles has been studied for a few decades. Various Soviet ballistic missiles was designed and produced with grid fins in 1970s. Moreover, they were used on missiles instead of conventional planar fins such as the Vympel R-77 air-to-air missile. A typical configuration of grid fins on a generic missile body is shown in Figure 1. Apart from being a control device, grid fins were also utilized as an emergency air brake on Soyuz TM-22 spacecraft in the past.

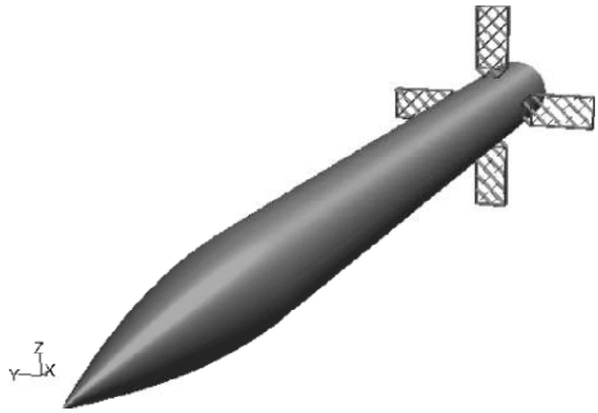

Figure 1: *A schematic of grid fin control surfaces on a missile body* [DeSpirito et al., 2001]

Grid fin control surfaces are counted as advantageous in packaging convenience, control capabilities and hinge moment requirements for subsonic and supersonic flight regimes. However, using grid fin control system creates some drawbacks in terms of range requirements for transonic flight condition [Fleeman, 2001]. The major reason for this is high drag force encountered by grid fin because of the internal flow field within the grid cells.

---

[1] Research Engineer, Email: erdem.dikbas@tubitak.gov.tr
[2] Senior Research Engineer, Email: ugras.baran@tubitak.gov.tr
[3] Assistant Professor in Mechanical Engineering Department, Email: csert@metu.edu.tr

There are various theoretical, experimental and computational researches and studies on grid fins in the literature. Most of them are limited to subsonic and supersonic flight regimes. Aerodynamic performance and characteristics of grid fins in these regimes have already been resolved in details. On the other hand, studies on transonic design issues have not reached at a satisfactory level, although articles on transonic drag reduction techniques and prediction methodologies are available.

In this study, a method which describes an initial set for geometric properties of a grid fin is investigated. Results of this study are to be used as a starting point of a detailed design of a grid fin operating within a low transonic flight regime.

METHOD

Relationship with Gas Dynamics

Flow field within a grid cell is basically governed by dynamics of air; therefore, inviscid gas dynamics relations seem to be applicable. Free stream air behaves as if it is exposed to a cross sectional area reduction, similar to a converging nozzle flow. Choking is a well-known occurrence, which is to happen when area reduction exceeds a certain limit.

In Figure 2, a simple 2-D schematic of a grid fin and two physical phenomena related to its behavior are presented. The distance between the two cell walls is named as width and the length of the individual cell walls is defined as depth in this study. Cell wall thickness can be taken as the third important parameter to define the geometry of interest.

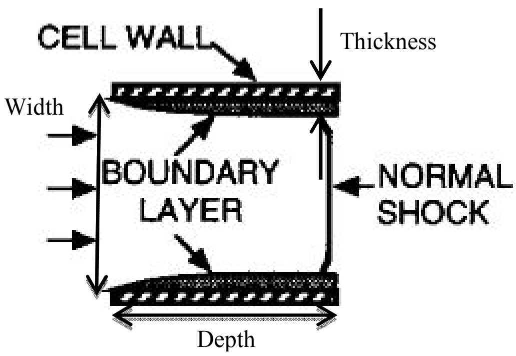

Figure 2: *Low transonic choking* [Washington and Miller, 1998]

The main concern in designing a grid fin for low transonic flight is to avoid the choking phenomenon within individual cells due to drag considerations. This limits the possible dimensions of the cells. It is known that there exists a Mach number at which choking starts to take place for certain cell width and wall thickness parameters. The aim of this study is to predict this Mach number and make the preliminary design of the grid fin accordingly.

Estimation Based on Isentropic Gas Dynamics Relations

Area rule is the determining factor of the critical Mach number, beyond which choking appears. The area reduction – Mach number relation of isentropic flows is given as:

$$\frac{A}{A^*} = \frac{1}{M}\left[\frac{2}{k+1}\left(1+\frac{k-1}{2}M^2\right)\right]^{(k+1)/[2(k-1)]} \tag{1}$$

where k stands for specific heat ratio and A^* represents the throat area where Mach number reaches to unity [Aksel and Eralp, 1993]. The area reduction to which flow field around a grid fin is exposed can also be expressed in terms of geometrical parameters as:

$$\frac{A}{A^*} = \frac{w}{w-t} \tag{2}$$

where w is width and t is thickness.

Critical Mach number denotes a minimum value at which choking occurs. This condition defines the Mach number that the drag created by pressure waves increases rapidly. Thus, this condition presents invaluable information on determining the design parameters of a grid fin at transonic flight.

To investigate if the critical Mach number is a function of spacing between the solid walls and wall thickness, an arbitrary design Mach number is selected. Then, the area ratio that causes the design Mach number to increase to sonic condition is determined using Eq. (1). This area ratio yields a relation between width and thickness parameters, as shown by Eq. (2). Finally, simulations are run for a few free-stream Mach numbers that are close to design Mach number.

Computational Study and Verification

FLUENT 2D density based Euler solver accompanied by an unstructured grid having 63,411 cells is utilized throughout the study. Sea-level atmospheric condition is applied at a desired Mach number using pressure-far-field boundary condition at upstream and downstream boundaries of the computational domain. Specified Mach number for these boundaries starts with 0.78 and it is increased by 0.01 steps until 0.93. The other two external boundaries are defined as translational periodic. At last, wall boundary condition is applied to solid boundaries of the domain.

Mach number of 0.80 has been chosen as design Mach number, for which the critical width-to-thickness ratio is estimated. In Figure 3a, for which free-stream Mach number is low, Mach number inside the grid fin does not reach sonic state, i.e., flow is not choked. On the other hand, examining Figures 3b and 3c, choking phenomenon is determined by observing the Mach number distribution between grid fin walls. Flow field inside the cells almost does not change once the flow is choked. Different downstream flow conditions are observed in Figures 3b and 3c. The reason for this is that the choked flow inside the grid fin is independent from the downstream conditions since downstream flow is supersonic and is not affected from downstream far-field.

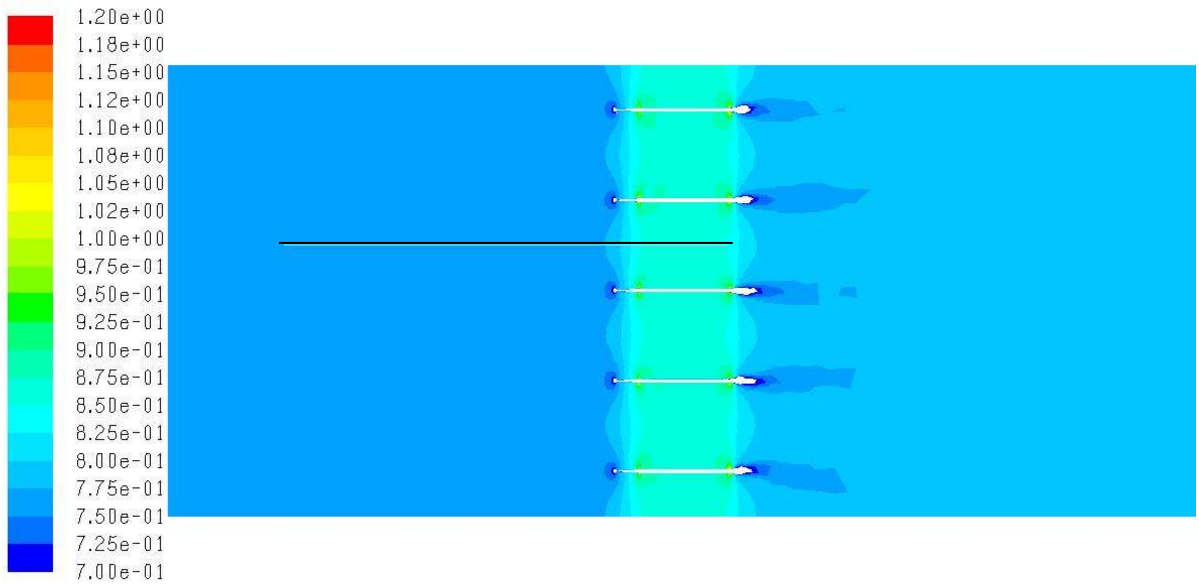

Figure 3a: *Mach number contours for estimated width-to-thickness ratio and free-stream Mach number of 0.78*

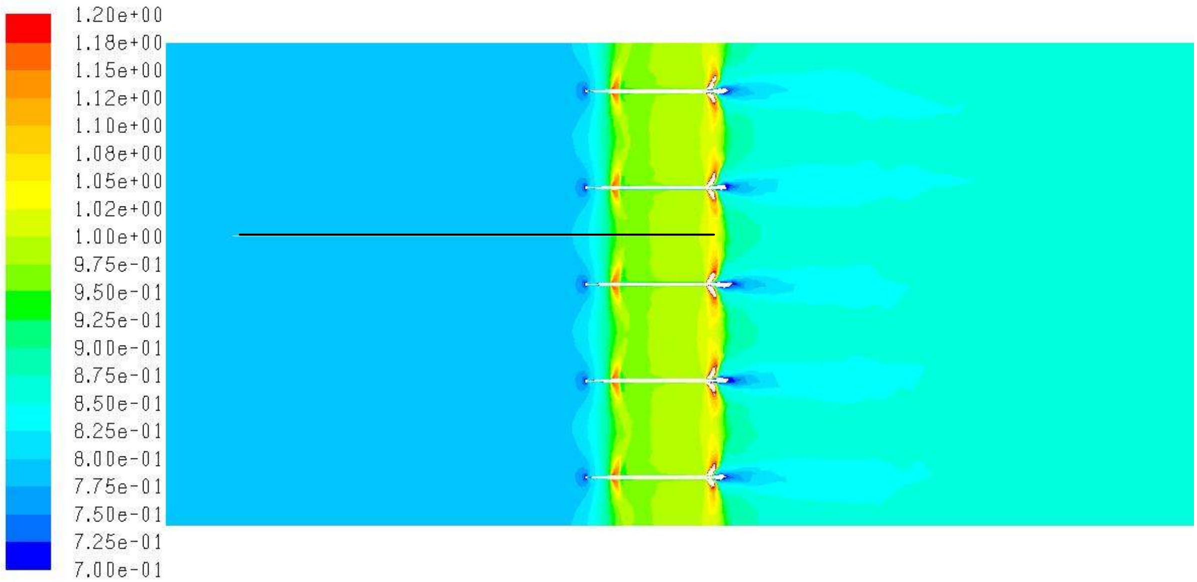

Figure 3b: *Mach number contours for estimated width-to-thickness ratio and free-stream Mach number of 0.85*

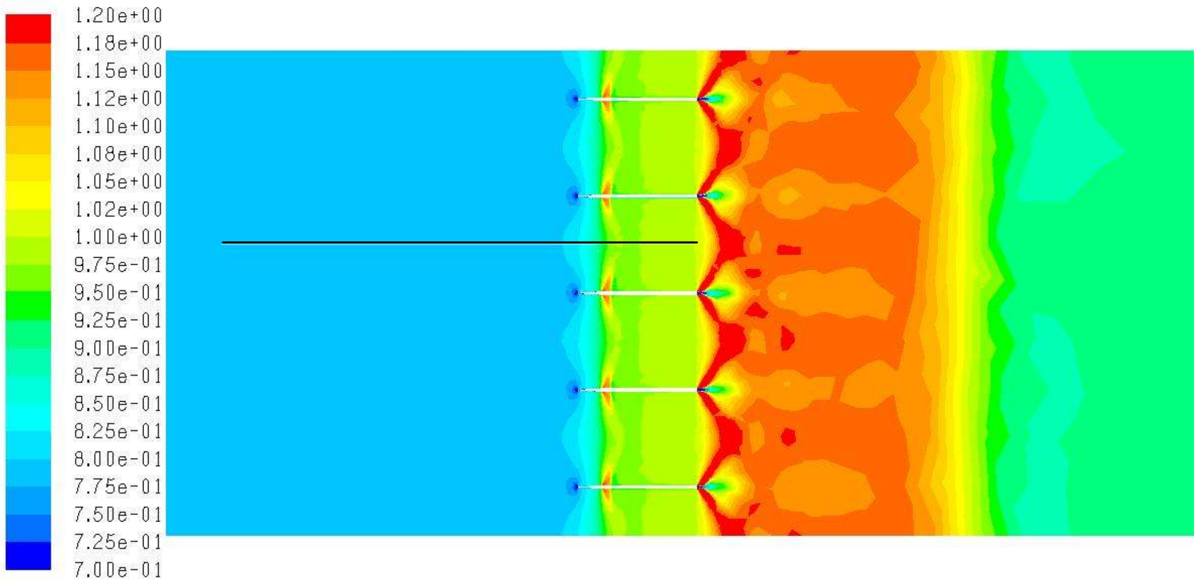

Figure 3c: *Mach number contours for estimated width-to-thickness ratio and free-stream Mach number of 0.90*

Flow between the grid fin walls encounters choking as free-stream Mach number is increased. Figure 4 shows how the solutions become independent of free-stream Mach number, as the upstream Mach number is increased. The data are taken along the centerline between two parallel walls, as it is shown on Figures 3a to 3c. Mach number in upstream region is converged to 0.79. As it is previously stated, the geometric design parameters are determined for a design Mach number of 0.80. CFD results verify the selection of geometric design parameters with a small offset.

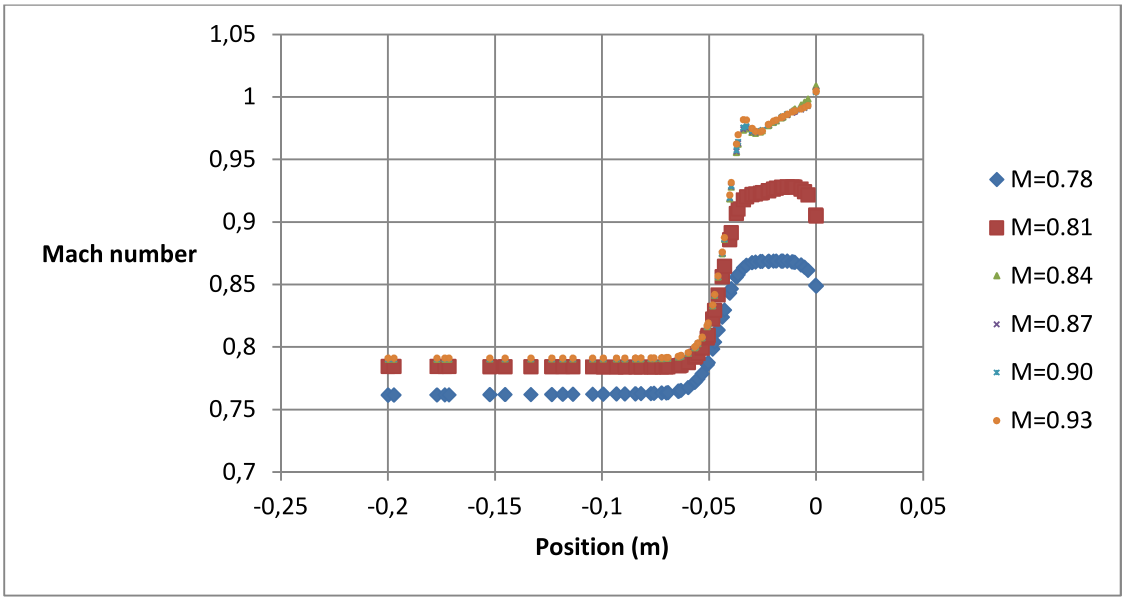

Figure 4: *Mach number along the centerline of model grid fin for different upstream boundary conditions*

Locus of Mach number values at the fin exit, which is denoted by 0 in Figure 4, is plotted in Figure 5. It is seen that the Mach number does not exceed unity significantly.

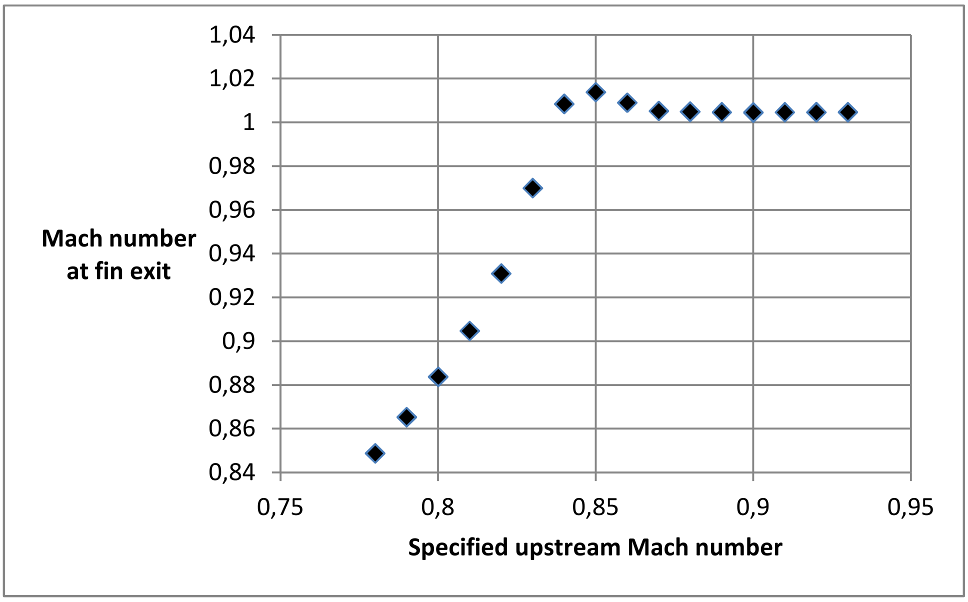

Figure 5: *Mach number at fin exit vs. specified upstream Mach number*

It should be noted that Mach number in upstream region converges to a limit value and deviate from the specified value for pressure-far-field boundary condition, as seen in Figure 6. Gas dynamics and choking concept explain this behavior. Indeed, it is not possible to fix pressure and density to an arbitrary value at the upstream boundary. Therefore, there is always an offset between the specified Mach number and the calculated one. Since the model seen in Figures 3a – 3c resembles a shock tube problem rather than a far field problem, application any boundary condition fixing the upstream flow conditions is not directly applicable and results should be evaluated accordingly. That is, it should be noted that, the critical Mach number is not the specified Mach number at which choking starts to occur; rather, it is the calculated upstream Mach number that yields choking.

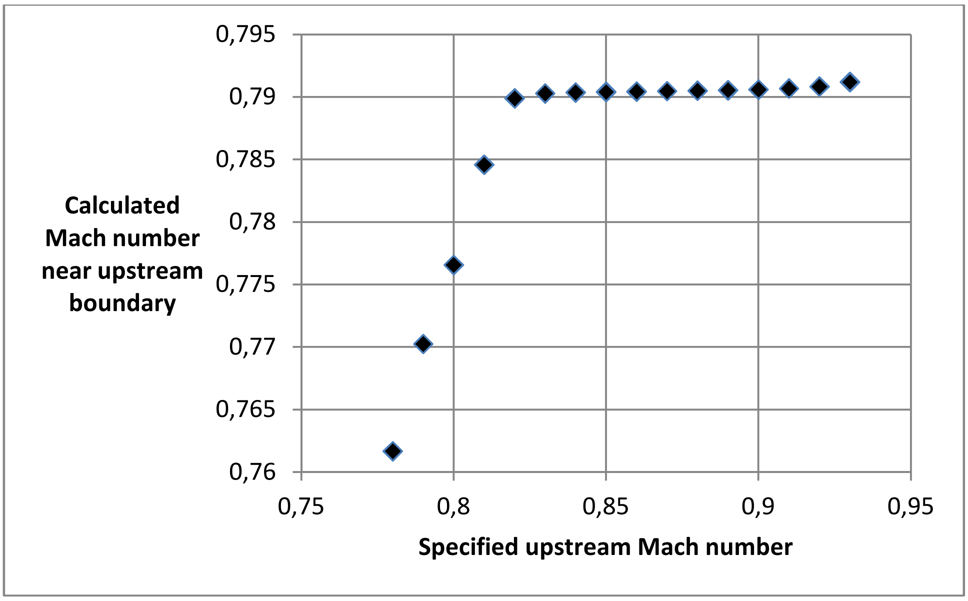

Figure 6: *Calculated Mach number near upstream boundary vs. the specified upstream Mach number*

Determination of the condition that yields high drag penalty is an important motivation for this study. Figure 7 shows the condition where drag increases dramatically. The result shows that the converging nozzle analogy is successful in drag prediction. That is, both choking and high drag penalty occurs at the same declared Mach number, which is close to 0.83. This value corresponds to M = 0.79 when the zone near upstream boundary is considered.

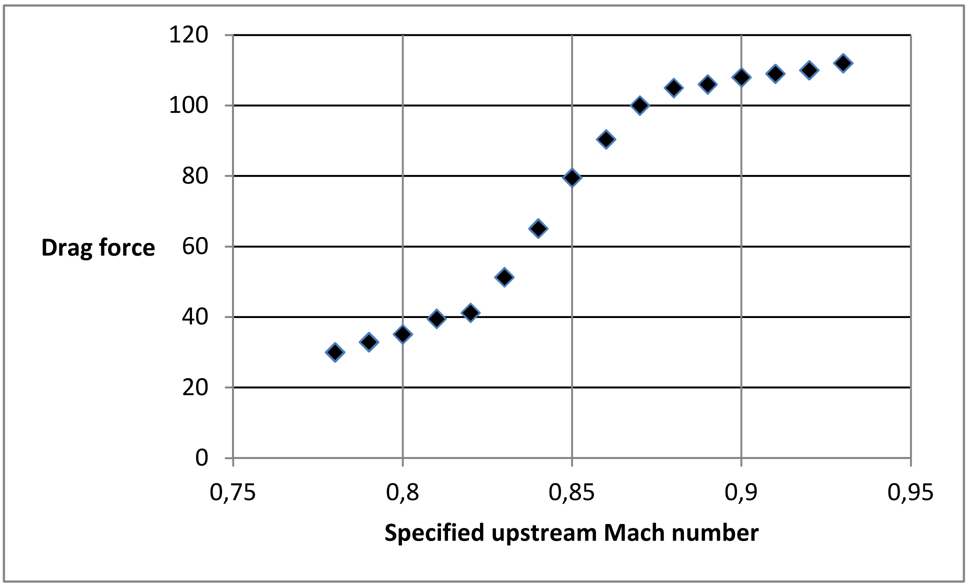

Figure 7: *Drag force vs. specified upstream Mach number*

Contribution of Boundary Layer Development

Although in the previous part choking has been considered as a result of area reduction due to wall thickness, formation of boundary layer also has a contribution. Decrease in mass flow in a region near the walls causes the boundary layer to act as an inner wall, as seen in Figure 2, which reduces the flow cross sectional area in the converging duct model. This can be taken into account in the area reduction concept investigated in this study. The dominant parameter is considered to be 'boundary layer displacement thickness' [Kretzschmar and Burkhalter, 1998], which is the thickness related to the mass flow deficiency due to boundary layer formation. This causes the critical Mach number to be dependent on the depth parameter, in addition to width and thickness. The magnitude of displacement thickness gives an idea about how the boundary layer formation affects the critical Mach number.

Equation 3 [Stratford and Beavers, 1961] provides the order of magnitude of displacement thickness for a typical grid fin geometry.

$$\delta^* = 0.046(1 + 0.8M^2)^{0.44} x\, {R_X}^{-1/5} \quad (3)$$

Displacement thickness is a function of free-stream Mach number, local Reynolds number and position in flow direction. Once typical values are imposed into the empirical relation, the order of magnitude of displacement thickness can be predicted. One might calculate this value at the trailing edge of 2 cm-deep grid fin and M=0.85 condition. If calculated, a value around 10^{-4} meters is found. This much of difference does not create even 1% change in the critical Mach number, when it is used in Eqn. 1. Therefore, it can be inferred that effect of boundary layer on the critical Mach number is insignificant.

A CFD model that has unrealistically small wall thickness and very large depth is used to visualize choking due to boundary layer formation. Spalart-Allmaras turbulence model is used as the viscous model. As seen in Figure 8, Mach number gradually increases and reaches to unity. This is an expected result because flow cross-sectional area gets narrower as boundary layer displacement thickness increases.

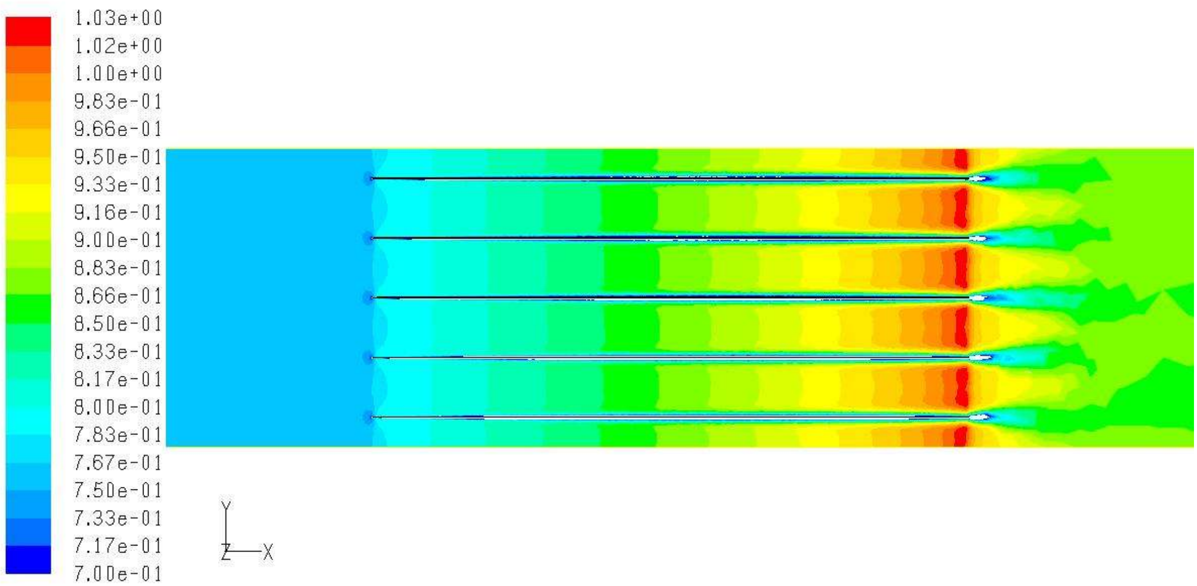

Figure 8: *Mach contour showing choking due to boundary layer formation*

CONCLUSION

Grid fin parameterization is studied in two-dimensional space and an area-rule-based prediction method is explained in this paper. Effect of grid cell width-to-thickness ratio is investigated and a simple 2-D design methodology is presented. This methodology uses the area rule equation and allows the designer to determine width and thickness properties of a grid fin. As the computational study has shown, using the methodology enables the design to avoid causing an extremely high drag, which is a crucial problem specific to transonic flight regime. Contribution of depth parameter is observed and concluded to be insignificant. This study provides a starting point for a preliminary design of grid fin control surfaces.

When the problem is considered in three dimensional space, a different area reduction problem is expected. Wall thickness effects should be handled at four edges of fin cell. Moreover, secondary flow fields, such as vortices, are expected to occur along the edges of 3D grid cells. Those three dimensional problems are addressed as the subject of future studies.

References

DeSpirito, J. , Edge, H. L. , Weinacht, P. , Sahu, J. (2001) *Computational Fluid Dynamics Analysis of a Missile with Grid Fins,* Journal of Spacecraft and Rockets, Vol. 38, No. 5, p: 713, Sep.-Oct. 2001

Fleeman, E. L. (2001) *Tactical Missile Design*, AIAA Education Series, Reston, VA, p: 41 Jan 2001.

Washington, W. D. and Miller, M. S. (1998) *Experimental Investigations of Grid Fin Aerodynamics: A Synopsis of Nine Wind Tunnel and Three Flight Tests,* NATO RTO AVT Symposium on Missile Aerodynamics, Sorrento, Italy, p:10-9, May 1998

Kretzschmar, R. W. and Burkhalter, J. E. (1998) *Aerodynamic Prediction Methodology for Grid Fins,* NATO RTO AVT Symposium on Missile Aerodynamics, Sorrento, Italy, p:11-5, May 1998

Aksel, M. H. and Eralp, O. C. (1993) *Gas Dynamics,* Prentice Hall, New York, NY, p: 99, 1993

Stratford, B. S. and Beavers, G. S. (1961) *The Calculation of the Compressible Turbulent Boundary Layer in an Arbitrary Pressure Gradient – A Correlation of Certain Previous Methods,* Aeronautical Research Council R&M, No. 3207, p: 7, 1961