Swept-back Grid Fins for Transonic Drag Reduction

Marco Debiasi1,§, Zeng Yan2, and Chng Tat Loon3
Temasek Laboratories, National University of Singapore, Singapore, 117411

A swept-back grid-fin configuration was proposed in a previous study as a mean for reducing the drag associated to flow choking in the lattice cells at transonic speeds. The current follow-on study aims to further reduce the unfavorable effects of the transonic choking phenomenon by adding a 20° sharp angle to the leading edges of the lattice walls. Viscous computational fluid dynamic simulations were performed to investigate the flow of an ogive-cylinder body with conventional and with swept-back grid fins with sharp leading edges at transonic and supersonic speeds and zero angle of attack. These data were further validated with wind-tunnel measurements in the transonic regime of models of the two types of fin. The results indicate that swept-back grid fins with sharp leading edges offer a significant drag reduction.

Nomenclature

$C_p$ = pressure coefficient
$C_D$ = overall vehicle drag coefficient
$C_{D,fin}$ = grid-fin drag coefficient
$c$ = grid-fin chord (web element chord)
$D$ = diameter of the ogive-cylinder body
$D_{fin}$ = fin drag force
$E$ = total energy
$h$ = grid-fin height
$H$ = total enthalpy
$k$ = turbulent kinetic energy
$L$ = length of the ogive-cylinder body
$M$ = Mach number
$M_{\infty}$ = freestream Mach number
$p$ = static pressure
$Re_D$ = Reynolds number, $U_{\infty} D / \nu$
$s$ = grid-fin span
$t$ = time
$T$ = static temperature
$u_i$ = velocity vector component ($i = 1, 2, 3$)
$u_i'$ = fluctuation of velocity vector component
$U_{\infty}$ = freestream velocity
$w$ = thickness of the web elements (walls of the lattice cells)
$x_i$ = position vector component ($i = 1, 2, 3$)

Greek letters
$\alpha$ = angle of attack
$\gamma$ = ratio of specific heats at constant pressure and constant volume

1 Senior Research Scientist, Temasek Laboratories, National University of Singapore, Singapore, Member AIAA.
2 Research Scientist, Temasek Laboratories, National University of Singapore, Singapore.
3 Associate Scientist, Temasek Laboratories, National University of Singapore, Singapore; currently Ph.D. Student at Princeton University, Princeton, NJ, USA.
\[ \xi = \text{angle of sharp leading edge} \]
\[ \Lambda = \text{swept-back angle} \]
\[ \mu = \text{dynamic viscosity} \]
\[ \nu = \text{kinematic viscosity} \]
\[ \rho = \text{flow density} \]

**I. Introduction**

A grid fin is an aerodynamic control surface consisting of an outer frame with an internal lattice of intersecting thin walls of small chord. Unlike conventional planar fins that are aligned parallel to the direction of the airflow, grid fins are mounted perpendicular to the flow allowing the oncoming air to pass through the cells of the lattice.

The main advantage of grid fins is that they have significantly smaller chord than conventional planar fins. Thus, they generate smaller hinge moments which require smaller actuators to rotate them in a high-speed flow. Their small chord also makes them less likely to stall at high angles of attack which increases their control effectiveness compared to conventional planar fins. Another advantage of grid fins is that they can easily be folded against an aerodynamic body to make them more compact and easier to store or transport.

Both theoretical and experimental studies have been performed on grid fins. The first investigation on grid fin aerodynamics was carried out by Belotserkovskiy et al.\(^1\) by using a theoretical method which relies mainly on empirical formulas. Burkhalter et al. used the vortex lattice type approach to analyze the grid-fin flow in the subsonic regime.\(^2\) Washington et al.\(^3\) conducted wind-tunnel experiments to study the aerodynamic effects of fin curvature and angle of protrusion from the body at some subsonic and supersonic Mach numbers. Their data indicate that “fin curvature has small effect on fin normal force, hinge moment, and bending moment”. Miller and Washington\(^4\) extended the experiments to study the effects on drag of the cross-section shape of the outer-frame walls and of the thickness of the walls of the lattice cells. Their findings indicate that simple shaping of the cross section of the frame, reduction of the wall thickness, or a combination thereof can considerably reduce drag levels with minimal impact on the grid-fin lift and other aerodynamic properties. Later experimental works by Abate et al.\(^5\) and by Fournier\(^6\) focus on overall aerodynamic performance of an ogive-cylinder body with grid fins.

The above studies rely mainly on force and moment measurements from individual fins and from the whole vehicle. However, no attempt was made to resolve in detail the flow characteristics inside and around the lattice cells. Recently Theerthamalai and Nagarathinam\(^7\) developed an aerodynamic prediction method for the estimation of the aerodynamic characteristics of grid-fin configurations at supersonic Mach numbers. The shock and expansion relations and the interactions between the shock and expansion waves were used to predict the pressure distribution inside each grid-fin cell.

Besides theoretical and experimental studies, computational fluid dynamics (CFD) has been used for investigating the aerodynamic characteristics of grid fins. Chen et al.\(^8\) simulated an ogive-cylinder body with grid fins by using the NPARC code of NASA. The issue of the grid fin size, in terms of both the walls’ thickness and the frontal shape of the lattice was addressed. Lin et al.\(^9\) performed computations of turbulent flows past a grid fin alone and past various fin/body combinations at Mach numbers of 0.7 and 2.5. The computations provided the detailed flow fields including Mach-number contours, pressure contours, and streamline patterns as well as the integrated aerodynamic coefficients. DeSpirito and Sahu\(^10\) conducted CFD studies by using the commercial CFD code FLUENT with the objective of investigating what advantages grid fins offer over conventional control surfaces. Good agreement was observed between CFD data and the aerodynamic coefficients measured in wind-tunnel tests of an ogive-cylinder body with grid fins. The simulations successfully calculated the flow structure around the fin in the separated-flow region at the higher angles of attack.

Depending on the speed of the airflow, grid fins can have a higher or lower drag compared to conventional planar fins.\(^5,\ 6\) The drag and control effectiveness of a grid fin at low subsonic speeds are similar to those of a conventional fin since the thin walls create very little disturbance in the flow of air passing through the lattice. However, the same behavior does not hold true in the transonic regime. Hughson et al.\(^11,\ 12\) conducted CFD studies to investigate the transonic flow fields about and through the cells of a vehicle with lattice grid fins. The CFD results illustrate that at transonic Mach numbers a normal shock forms at the back of the grid cells. The flow inside the cells chokes thus reducing the flow rate through the lattice which effectively acts as an obstacle to the flow with attendant increased drag. At higher speed, the normal shocks that form in front of the grid are swallowed by the lattice which gives the fin good drag and control characteristics.
Reducing the drag of grid fins in the transonic regime is the main objective of the present study. A modified configuration consisting of a swept-back grid fin was proposed by the authors and studied via numerical simulations with results indicating a promising drag reduction. This study considers the additional benefit of adding a 20° sharp leading edge to the lattice walls of the swept-back grid fin. This configuration is different but in some way parallels that proposed by Guyot and Schülein which has shown to reduce the drag by 30-40% at freestream Mach numbers from 2 to 6.

II. Numerical Simulations

A. Grid fin configurations

The fin geometry investigated by Abate et al., DeSpirito and Sahu, and Hughson et al. and shown in Fig. 1 a) and b) has been selected as the baseline configuration in our study. All the dimensions in Fig. 1 are relative to the diameter $D$ of the ogive-cylinder body. This consists of a 3D long tangent ogive nose with a 13D long (for comparison with data from literature) or 7D long (for comparison with experimental measurements) cylindrical afterbody with four grid fins mounted in a cruciform orientation. The pitch axis of the grid fins is located 1.5D from the rear of the cylinder. The grid fin has a rectangular shaped outer frame with span $s$, height $h$ and chord $c$ of size $0.75D$, $0.333D$ and $0.118D$ respectively. The thickness $w$ of the walls of the lattice is $0.007D$.

Fundamental aerodynamic considerations similar to those presented by Guyot and Schülein, as well as in the design of transonic and supersonic inlets, suggest that adding a swept-back angle to the fin lattice should improve its drag characteristics. As stated in our previous study, the grid-fin geometry is modified by sweeping back the frameworks of grid cells along the chord direction with the same projected structure and dimensions as shown in Fig. 1 a). Figure 1 c) shows the top view of the swept-back grid fin configuration (SB), with the swept-back angle $A = 30°$, which has also been reported in our previous study. The swept back angle also slightly staggers the leading edges of the lattice with respect to each other. This in conjunction with adding sharp leading edges (SB-sharp) allows for improved flow ingestion by the lattice at transonic and supersonic speeds, as illustrated in Fig. 2. In the present study, the angle of the sharp leading edges is $\xi = 20°$, i.e. the same value used in the configuration studied by Guyot and Schülein. In our numerical simulations the swept-back grid fins are mounted in the same cruciform orientation and position on the ogive-cylinder body as the baseline grid fins.

The merit of using the SB-sharp design described above is evaluated by comparing the baseline and the SB-sharp fins at freestream Mach numbers $M_{\infty}$ between 0.817 and 1.7. The baseline and the SB fins (without sharp leading edges) have been already studied numerically with results indicating a drag reduction for the SB geometry. Thus, this study further considers the effect of adding the sharp leading edges.

B. Computational mesh

The creation of a mesh of satisfactory quality is the most demanding task of the numerical simulation. Due to the complex geometry of the grid fins, creating a structured mesh with good quality would have been a long and tedious process, especially within the region of the grid fins. Similar to the mesh of the baseline and swept-back models in our previous study, we used GAMBIT, the grid generation software supplied in the FLUENT CFD suite, to generate unstructured meshes of satisfactory quality. By taking advantage of the model symmetry at zero angle of attack, only one quarter of the geometry was used for the calculation domain in the current CFD study.

As illustrated in Ref. 13, the wall function of FLUENT was used for generating the boundary layer mesh near the body and fin surfaces. The spacing between the first mesh point and the surface is 0.001D, with a growth factor of 1.2 and at least 9 points distributed within the boundary layer corresponding to the Reynolds number used by the free-flight test of Abate et al.

Figure 3 shows the details of the mesh topology in the $x$-$z$ (longitudinal) and in the $y$-$z$ (transversal) planes for the SB-sharp fin. Figure 3 c) is an enlargement of Fig. 3 a) showing the detail of the sharp leading edges. Inside the leading edge part, there are tetrahedrons with triangle surfaces and the Wedge Primitive meshing scheme was employed to create a radial mesh on a three-sided face. With the application of the Wedge Primitive meshing scheme, GAMBIT creates a mapped mesh that includes a group of triangular mesh elements emanating from common endpoints. Triangle surface mesh can be observed in the fin surface, as shown in Fig. 3 c). Meshes within the fin are highly clustered to the walls’ surfaces as well as to their leading and trailing edges. Three transitional parts on the top of, before, and after the fin smoothly transform the very fine mesh close to the fin to a coarser mesh at a distance from it which significantly reduces the total number of mesh nodes for the whole domain. Within these
transitional parts, T-Grid type was used, as seen in Fig. 3 a). The advantage of this type of meshing is that it allows different mesh number and mesh type in different surfaces, whereas the disadvantage is that the volume mesh strongly depends on the mesh quality at each surface. Thus great care was put in generating high-quality meshes at all the surfaces. In the remaining part of the domain, which mainly covers the ogive-cylinder body, structured mesh was used.

The upstream and downstream boundaries are located 12D away from the nose and tail of the ogive-cylinder body, and radially 16D away from the cylinder surface. The total number of grid cell elements for the computational domain is about 1.2 million with 118 volumes.

C. Governing equations and flow solver

3D Navier-Stokes equations coupled with a turbulence model were used to simulate the turbulent flow field.\(^{13}\) The governing equations for the unsteady compressible turbulent flow in 3D are expressed as follows:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0
\]  \hspace{1cm} (1)

\[
\frac{\partial}{\partial t}(\rho u_j) + \frac{\partial}{\partial x_j}(\rho u_j u_j) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial t}(\mu) - \frac{\partial \tau_{ij}}{\partial x_j}
\]  \hspace{1cm} (2)

\[
\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_j}(\rho u_j H) = \frac{\partial}{\partial x_j} \left[ u_j \tau_{ij} + (\mu + \sigma^e \mu_T) \frac{\partial k}{\partial x_j} - q_j \right]
\]  \hspace{1cm} (3)

where \(t\) is the time, \(x_j\) the position vector, \(\rho\) the density, \(u_j\) the velocity vector, \(p\) the pressure, \(\mu\) the dynamic viscosity. The total energy and enthalpy are \(E = e + k + u_j u_j / 2\) and \(H = e + p / \rho + k + u_j u_j / 2\), respectively, with \(e = p/((\gamma - 1)\rho)\). The term \(\gamma\) is the ratio of specific heats at constant pressure and constant volume. Other quantities are defined in the following equations:

\[
\mu_T = \rho v_t
\]  \hspace{1cm} (4)

\[
S_y = \frac{1}{2} (\frac{\partial u_y}{\partial x_j} + \frac{\partial u_j}{\partial x_j})
\]  \hspace{1cm} (5)

\[
\tau_{ij} = 2 \mu_T (S_{ij} - \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij}) - \frac{2}{3} \rho k \delta_{ij}
\]  \hspace{1cm} (6)

\[
\hat{\tau}_{ij} = 2 \mu (S_{ij} - \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij}) + \tau_{ij}
\]  \hspace{1cm} (7)

\[
q_j = -\left(\frac{\mu}{Pr_l} + \frac{\mu_T}{Pr_T}\right) \frac{\partial h}{\partial x_j}
\]  \hspace{1cm} (8)

\[
k = \frac{1}{2} u'_i u'_j
\]  \hspace{1cm} (9)

where \(\delta_{ij}\) indicates the Kronecker delta, and \(u'_i\) is the fluctuation of the velocity component \(u_i\).

The CFD software FLUENT was chosen to compute the flow. The governing equations were solved by using the finite volume method with unstructured-grid. The Spalart-Allmaras one-equation turbulent model was used. A second-order, upwind discretization scheme was employed for the flow variables and the turbulent viscosity. An implicit, density-based solver was adopted for the high-speed compressible flow investigated.

The results were tracked to ensure their convergence. Three orders of magnitude reduction of the maximum residuals of all the variables was adopted as a criterion to confirm the global convergence of the simulations. As an example, Fig. 4 shows the convergence history of the overall (i.e. body and fins) drag coefficient \(C_D\) of the SB-sharp configuration at \(M_{\infty}=1.19\). The figure shows that the value of \(C_D\) undergoes initial oscillations but after about \(10^4\) iterations it stabilizes and approaches a constant value thus confirming the convergence of the simulation. Similar results were obtained for all the other configurations and the Mach number studied.
Parallel computing was used to speed up the computations. A complete parallel simulation run in 8 processors and took approximately 120 CPU hours. Grid-independent studies were also carried out to ensure that all the numerical solutions are grid-independent.

D. Boundary conditions
Comparisons have been made with data available in literature for baseline grid fins mounted on a $L/D = 16$ ogive-cylinder body at free-flight conditions $^5, ^{11}, ^{12}$ and with data from the experimental measurements described in section III. The comparison with the free-flight data is detailed in Ref. 13. It suffices here to recall that for these cases the freestream static pressure and temperature are 101,325 Pa and 295 K, respectively. The focus of this study is comparing the results of experimental measurements conducted in a windtunnel with simulations matching the same conditions as summarized in Table 1.

In all cases the angle of attack of the body and the deflection of the fins with respect to the flow are zero. Far-field pressure boundary conditions were applied for the outer boundaries, symmetry conditions for the symmetry surfaces, and non-slip conditions for all the solid surfaces. At the downstream outflow boundary a used-defined function program is compiled to extrapolate all the flow quantities including pressure from the interior grid points.

E. Validation
The numerical simulations described above have been validated by comparing the results with data available in literature.$^{13}$ As an example, Fig. 5 shows the Mach number ($M$) contours along the $x$-$z$ plane of baseline fins mounted on a $L/D = 16$ body. Results are shown for three different Mach numbers simulating free-flight conditions. The figures on the right are obtained with the current simulation and match quite well those on the left from Ref. 12.

The drag coefficient $C_D$ was calculated by using the FLUENT postprocessor to integrate the viscous and pressure forces along the surfaces of the body and of the fins. The reference area is 1/4 of the cross-sectional area of the cylinder base and the reference length is the cylinder diameter. Figure 6 compares the overall drag coefficient obtained for baseline fins mounted on the $L/D = 16$ body with the results of free-flight tests from Abate et al.$^5$ and with comparable CFD results provided by Hughson et al.$^{11}$ The figure indicates that our CFD results agree well (within 4%) with those from Ref. 11. However, both CFD results predict about 10% lower drag coefficients than the values obtained from free-flight tests, possibly because the free-flight conditions differ somewhat from the conditions used in both CFD analyses. Nevertheless, both simulations follow quite well the trend of the experimental data indicating that the numerical methods are accurate enough to capture the main flow phenomena and physics of vehicles with grid fins. All the data in the figure show that the drag coefficient increases with the freestream Mach number for low transonic to sonic conditions. With further increase of the Mach number to supersonic conditions, the drag coefficient gradually decreases. The drag increase around Mach 1.0 is explained in the literature as a result of flow choking within the cells of the lattice at transonic conditions.

F. Numerical results
Having validated the numerical simulation process as described above, we used it to calculate the flow of several grid fin and body configurations at different values of $M_\infty$.

Our previous study reports additional results for the baseline fins and for the swept-back fins with blunt leading edges mounted on a $L/D = 16$ ogive-cylinder body at free-flight conditions in the $M_\infty$ range 0.905 to 2.0. The results clearly show that introducing the swept-back angle $\Lambda = 30^\circ$ is beneficial for drag reduction.$^{13}$

The current study compares the numerical results with experimental measurements. Thus the geometry and the flow conditions adopted for CFD should match those of the experiments. In this case a shorter ogive-cylinder body with $L/D = 10$ is required and the freestream pressure and temperature are not the atmospheric values at sea level but vary with the freestream Mach number as indicated in Table 1. For these conditions Table 2 summarizes the values of the overall drag coefficient $C_D$ obtained with the baseline and the SB-sharp fins mounted on the body. The last column shows the percentage reduction of $C_D$ offered by the SB-sharp fins compared to the baseline type, from which it is obvious the advantage offered by adding a sweep-back angle and sharp leading edges. For the same cases Table 3 summarizes the values of the drag coefficient $C_{D,\text{fin}}$ obtained for the fins alone by neglecting the effect of the body drag. The results indicate that the SB-sharp fins have at least 30% lower drag than the baseline fins in the transonic regime and that this further decreases at higher speeds.
III. Experimental Measurements

A. Experimental setup

Experiments were conducted to measure the drag of the baseline and of the SB-sharp grid fins. To this aim, models of these grid fins were fabricated and tested in a transonic wind tunnel. The models are made of stainless steel, Fig. 7, and their dimensions are $s = 85.7\text{mm}$, $h = 38.1\text{mm}$, $c = 13.5\text{mm}$, and $w = 0.8\text{mm}$, Fig. 1. The fins were installed at $1.5D$ from the rear of the cylindrical body of an AGARD-C model which has $L/D = 10$. In order to better discriminate the drag of the fins, the body does not have the usual AGARD C wings and empennage. The body was mounted on a sting close to the center of the wind-tunnel test section. The minimum distance of the fin tips from the test-section walls was more than 3 times the fin span thus wall interference and blockage effects were negligible.

Experimental measurements were taken at freestream Mach numbers between 0.75 and 1.7. The Mach number increment is 0.05 in the range $M_\infty = 0.75$ to 1.20 and 0.1 above this range. The wind tunnel is of the blow-down type and incorporates a control system that maintains the Mach number and the pressure in the test section within 1% of the desired values. In these conditions the total pressure in the test section was about 1.7 times higher than ambient. The variation of the temperature during the experiments was negligible due to the short duration of each run and to the use of a heat exchanger that minimizes the temperature drop with the depletion of the air in the reservoir. The flow conditions summarized in Table 1 for CFD are representative of the conditions experienced by the models during the experimental runs.

A balance with a resolution of 0.014N and an accuracy of 0.092% was used to measure the axial forces. These data were measured and recorded simultaneously to the total and static values of the pressure and temperature in the test section.

B. Experimental results

The drag force exerted by the fins at each freestream Mach number was obtained as the difference of the drag of the AGARD-C model with fins and without them. No significant differences were found between repeated measurements.

Figure 8 presents the values of the drag force $D_{fin}$ measured for the baseline and for the SB-sharp fins at different values of $M_\infty$ between 0.75 and 1.7. For both fins the drag increases with the freestream velocity and reaches a maximum at about $M_\infty = 1.1$ after which is gently decreases. This is consistent with the choking of the flow in the lattice cells at transonic conditions. At this Mach number the maximum drag measured for the baseline fin is 88 N whereas the maximum drag measured for the SB-sharp fin is 62 N at $M_\infty = 1.05$. It should be noted that the drag of the SB-sharp fin is more than 20 N lower than the drag of the baseline fin across the whole Mach range explored.

Similar to the numerical simulations, the drag coefficient of the fins was obtained by using as reference area 1/4 of the base of the AGARD C cylindrical body and as reference length its diameter. Figure 9 presents the values of the drag coefficient $C_{D,fin}$ of the baseline and SB-sharp fin from the experiments together with the CFD values from columns 2 and 3 of Table 3. Figure 10 presents the corresponding reduction of the drag coefficient (percent benefit) offered by the SB-sharp fin compared to the baseline type together with the corresponding values from the last column of Table 3. The experiments substantially confirm the results of the numerical simulations and indicate that, compared to the baseline fin, the drag of the SB-sharp fin is reduced by more than 30% in the Mach range explored with the possible exception of a smaller reduction (25%) close to $M_\infty = 1.1$ where the drag is higher as discussed above. Additional measurements should be performed around this Mach number to confirm the dip observed in Fig. 10 for this flow condition.

Interestingly, and similar to what previously observed in commenting Fig. 6, the numerical simulations seem to underestimate the value of the drag coefficient by about 10% compared to those from the experiments. Again, this may be the results of some differences, e.g. the presence of the sting support, between the experiments and the numerical simulations. Notwithstanding this discrepancy, it should be noted that the values of $C_{D,fin}$ from CFD and from the experiments follow the same trend confirming that the numerical solutions are accurate enough to capture the main flow phenomena of the grid fins.
IV. Discussion

Having verified that the results from the numerical simulations agree quite well with the wind-tunnel measurements, we utilize the numerical data to explore in more detail the features of the flow across the grid fins.

The flow fields through the baseline and the SB-sharp grid fins are fairly complex. Figures 11 to 13 show the detailed Mach number contours on the x-z plane passing through the baseline and the SB-sharp grid fins at $M_{\infty} = 1.045, 1.332,$ and 1.70, respectively. The walls of the baseline configuration have blunt leading edges which cause a local deceleration of the approaching flow. The boundary layer inside the cells, which grows from the leading edge, and the flow stagnating at the junctions of the walls introduce a decrease in the flow area. At $M_{\infty} = 1.045$ the flow inside the cells of the baseline lattice is accelerated, Fig. 11 a), thus reaching pressure values at the lattice exit that are not in balance with the pressure of the flow surrounding the fin. Thus the flow exiting the cells equalizes the pressure of the surroundings through a series of expansions and compressions. The X pattern past the lattice elements in Figs. 11a) clearly shows the expansion fans of the flow exiting the cells. When such expansions reach pressure values below the surrounding, they are followed by rather abrupt shocks across which the flow pressure matches the surroundings. These shocks create an arch past the fin cells. The energy dissipation associated with these processes limits the efficient passage of the flow through the lattice with attendant choking of the flow. The system of expansion fans and shocks from the SB-sharp configuration is similar, Fig. 11 b), but the strength of these phenomena is reduced compared to the baseline case.

Similar observations can be drawn for the grid fins at $M_{\infty} = 1.332$, shown in Figure 12. At this Mach number the normal shock in front of the SB-sharp fin is almost attached to the lattice, Fig. 12 b), as opposed to the detached bow shock in front of the baseline fin, Fig. 12 a), which worsens the drag by forcing additional air to spill around the fin.

At $M_{\infty} = 1.70$, i.e. past the transonic regime, the flow characteristics of the baseline and SB-sharp cases are quite different, see Fig. 13. The strong shock in front of the baseline fin, Fig. 13 a), induces large flow losses. The pressure of the flow past this strong shock rises at the expense of the velocity and the flow emerges at the exit of the cells with relatively low speed and higher pressure than the surroundings. Past the fin an expansion and shock structure similar to the lower speed cases is observed through which the flow pressure equalizes with the surroundings. The strong shock upstream of the baseline fin is replaced by oblique shocks at the leading edges of the SB-sharp fin, Fig 13 b). The flow remains supersonic within the cells and weak expansion waves form at the trailing edge of the cells. Flow choking is notably absent and lower drag is obtained.

These observations support the design rationale of sweeping back the lattice of the grid fins and of adding the 20° sharp leading edge to the swept-back lattice both of which were expected to reduce the losses and facilitate the passage of the flow through the lattice cells.

The current study indicates the advantage of using a swept-back with sharp leading edges grid-fin at transonic speeds and it hints, but does not specifically address, the benefit of this configuration at subsonic speeds or at higher supersonic speeds, Figs. 9 and 10. At the same time it does not provide any indication of the flow characteristics of the SB-sharp fin placed at an incident angle with respect to the freestream or of its control authority. These will be explored in a follow-on study which, in some ways, will parallel and complement the work of Guyot and Schülein. At the same time we will explore the flow characteristics of a swept-forward (SF) with sharp leading edges grid fin, Fig. 14, which can offer some advantages in practical applications.

V. Conclusion

At transonic flight conditions flow choking usually occurs in the lattice cells of conventional grid fins, which causes a significant increase of the drag force. The present study proposes an improved grid-fin design which combines a 30° swept-back lattice structure with sharp leading edges of the cells walls. The objective is to reduce the transonic flow choking and the associated drag. Viscous computational fluid dynamic simulations were performed to investigate the flows over an ogive-cylinder body with conventional (baseline) and swept-back with sharp leading edges (SB-sharp) grid fins in transonic and supersonic flow regimes with freestream Mach number ranging from 0.817 to 1.7, at zero angle of attack. The results agree well with data available in literature for the baseline case. Experimental measurements with models of the baseline and of the SB-sharp fin performed in a wind tunnel at Mach from 0.75 to 1.7 also confirm the CFD results and indicate that the SB-sharp fin has a drag coefficient about 30% lower than the conventional design. The results from simulations were used to analyze in more details the characteristics of the flow through the two types of grid-fin. At transonic and low supersonic speed the SB-sharp fin significantly reduces the strength of the dissipative phenomena at the back of the fin lattice which
cause the flow choking with attendant drag increase. At higher supersonic speed the SB-sharp fin promotes the formation of oblique shocks attached to the leading edges and a cleaner supersonic flow within the lattice cells.

A future study will explore the effect of the angle of incidence of the fins with respect to the freestream in order to assess their control capability as well as the use of a swept-forward (SF) grid-fin configuration with sharp leading edges.

References

Table 1. Flow conditions used in CFD matching the experimental measurements.

<table>
<thead>
<tr>
<th>$M_\infty$</th>
<th>$p$ (Pa)</th>
<th>$T$ (K)</th>
<th>$U_\infty$ (m/s)</th>
<th>$Re_D \times 10^6$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.817</td>
<td>1.11·10^5</td>
<td>265</td>
<td>267</td>
<td>2.50</td>
</tr>
<tr>
<td>0.905</td>
<td>1.01·10^5</td>
<td>256</td>
<td>290</td>
<td>2.56</td>
</tr>
<tr>
<td>1.045</td>
<td>8.63·10^4</td>
<td>243</td>
<td>327</td>
<td>2.59</td>
</tr>
<tr>
<td>1.190</td>
<td>7.20·10^4</td>
<td>229</td>
<td>361</td>
<td>2.54</td>
</tr>
<tr>
<td>1.332</td>
<td>6.25·10^4</td>
<td>226</td>
<td>401</td>
<td>2.48</td>
</tr>
<tr>
<td>1.400</td>
<td>5.43·10^4</td>
<td>218</td>
<td>414</td>
<td>2.31</td>
</tr>
<tr>
<td>1.477</td>
<td>4.71·10^4</td>
<td>208</td>
<td>427</td>
<td>2.16</td>
</tr>
<tr>
<td>1.600</td>
<td>4.20·10^4</td>
<td>201</td>
<td>455</td>
<td>2.13</td>
</tr>
<tr>
<td>1.700</td>
<td>3.57·10^4</td>
<td>193</td>
<td>473</td>
<td>1.96</td>
</tr>
</tbody>
</table>

Table 2. Drag coefficient $C_D$ from numerical simulations of the $L/D = 10$ ogive-cylinder body with baseline fins and SB-sharp fins at freestream conditions as in Table 1.

<table>
<thead>
<tr>
<th>$M_\infty$</th>
<th>Baseline</th>
<th>SB-sharp</th>
<th>SB-sharp reduction</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.817</td>
<td>0.638</td>
<td>0.518</td>
<td>18.81%</td>
</tr>
<tr>
<td>0.905</td>
<td>0.760</td>
<td>0.609</td>
<td>19.87%</td>
</tr>
<tr>
<td>1.045</td>
<td>0.892</td>
<td>0.744</td>
<td>16.59%</td>
</tr>
<tr>
<td>1.190</td>
<td>0.862</td>
<td>0.730</td>
<td>15.31%</td>
</tr>
<tr>
<td>1.332</td>
<td>0.823</td>
<td>0.684</td>
<td>16.89%</td>
</tr>
<tr>
<td>1.400</td>
<td>0.810</td>
<td>0.678</td>
<td>16.30%</td>
</tr>
<tr>
<td>1.477</td>
<td>0.797</td>
<td>0.659</td>
<td>17.31%</td>
</tr>
<tr>
<td>1.600</td>
<td>0.772</td>
<td>0.621</td>
<td>19.56%</td>
</tr>
<tr>
<td>1.700</td>
<td>0.753</td>
<td>0.602</td>
<td>20.05%</td>
</tr>
</tbody>
</table>

Table 3. Drag coefficient $C_{D,fin}$ from numerical simulations of a baseline fin and a SB-sharp fin at freestream conditions as in Table 1.

<table>
<thead>
<tr>
<th>$M_\infty$</th>
<th>Baseline</th>
<th>SB-sharp</th>
<th>SB-sharp reduction</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.817</td>
<td>0.098</td>
<td>0.061</td>
<td>37.76%</td>
</tr>
<tr>
<td>0.905</td>
<td>0.125</td>
<td>0.082</td>
<td>34.40%</td>
</tr>
<tr>
<td>1.045</td>
<td>0.111</td>
<td>0.073</td>
<td>34.23%</td>
</tr>
<tr>
<td>1.190</td>
<td>0.103</td>
<td>0.067</td>
<td>34.95%</td>
</tr>
<tr>
<td>1.332</td>
<td>0.098</td>
<td>0.063</td>
<td>35.71%</td>
</tr>
<tr>
<td>1.400</td>
<td>0.096</td>
<td>0.061</td>
<td>36.46%</td>
</tr>
<tr>
<td>1.477</td>
<td>0.094</td>
<td>0.057</td>
<td>39.36%</td>
</tr>
<tr>
<td>1.600</td>
<td>0.093</td>
<td>0.052</td>
<td>44.00%</td>
</tr>
<tr>
<td>1.700</td>
<td>0.092</td>
<td>0.051</td>
<td>44.57%</td>
</tr>
</tbody>
</table>
Figure 1. Schematic of the grid-fin configurations: a) front view of the grid-fin structure; b) top view of the baseline grid fin; c) top view of the swept-back grid fin.

Figure 2. Schematic of the flow approaching a conventional, blunt-leading-edged cell (left) and a swept-back, sharp-leading-edged cell (right).
Figure 3. Mesh of the grid fin region for the SB-sharp geometry: a) $x$-$z$ (longitudinal) plane; b) $y$-$z$ (transversal) plane; c) enlargement of a).

Figure 4. Convergence history of the overall drag coefficient with SB-sharp fins at $M_\infty = 1.19$. 

American Institute of Aeronautics and Astronautics
Figure 5. Mach number contours on the x-z plane of baseline fins mounted on a $L/D = 16$ body at simulated free-flight conditions: a) at $M_\infty=1.045$ from Ref. 12; b) at $M_\infty=1.045$ from current simulations; c) at $M_\infty=1.332$ from Ref. 12; d) at $M_\infty=1.332$ from current simulations; e) at $M_\infty=2.0$ from Ref. 12; f) at $M_\infty=2.0$ from current simulations.
Figure 6. Comparison of the overall drag coefficient $C_D$ from free-flight test data from Ref. 5, CFD results from Ref. 11, and current CFD results.

Figure 7. Grid-fin models for experimental measurements in wind tunnel: conventional flat design from Hughson et al. 11 (baseline for this study; left) and swept-back with sharp leading edges design (SB-sharp; right).
Figure 8. Fin drag from experimental measurements.

Figure 9. Fin drag coefficients from experimental measurements and from numerical simulations.
Figure 10. Reduction of the drag coefficient of the SB-sharp fin relative to the baseline fin from experimental measurements and from numerical simulations.

Figure 11. Mach number contours on the x-z plane of fins mounted on a $L/D = 10$ body at $M_{\infty}=1.045$: a) baseline fin; b) SB-sharp fin.
Figure 12. Mach number contours on the x-z plane of fins mounted on a $L/D = 10$ body at $M_{\infty} = 1.332$: a) baseline fin; b) SB-sharp fin.

Figure 13. Mach number contours on the x-z plane of fins mounted on a $L/D = 10$ body at $M_{\infty} = 1.70$: a) baseline fin; b) SB-sharp fin.
Figure 14. Representation of a swept-forward (SF) grid fin. The fin pictured here is a SB-fin model turned around (notice the blunt leading edges). Actual SF fins will have sharp leading edges.