High Performance Supersonic Missile Inlet Design Using Automated Optimization

G.-C. Zha* D. Smith† M. Schwabacher‡ K. Rasheed§ A. Gelsey¢ D. Knight‖
Rutgers University - The State University of New Jersey
PO Box 909 • Piscataway NJ 08855-0909

Marty Haas
United Technologies Research Center
East Hartford, Connecticut 06180

January 11, 1996

Abstract

A multi-level design strategy for supersonic missile inlet design is developed. The multi-level design strategy combines an efficient simple physical model analysis tool and a sophisticated CFD Navier-Stokes analysis tool. The efficient simple analysis tool is incorporated into the optimization loop and the sophisticated CFD analysis tool is used to verify, select and filter the final design. Because of the non-smooth design space, the optimization combines both a non-gradient method and a gradient method. A geometry model for the supersonic missile inlet is developed. Significant improvement of the inlet total pressure recovery has been obtained. Detailed flow field analysis is also presented.

1 Introduction

With the advent of powerful computers, aerodynamic design with automated optimization has become more and more popular with the motivation to achieve a better quality design. The background of our current work is to develop a technology for supersonic missile aerodynamic design with automated optimization including component performance optimization and mission analysis, etc. The purpose of this paper is to optimize the supersonic inlet design as a first step of the long term goal. The total pressure recovery coefficient, which represents the aerodynamic performance of the inlet, is chosen as the optimization objective. The external cowl shape and cowl diameter are unchanged and hence the external cowl wave drag remains fixed.

The missile inlet in this design work is an axisymmetric mixed compression inlet with cruise Mach 4 at 60000 feet altitude. Minimum manufacture cost for this inlet is required. Therefore, the techniques such as boundary layer bleed and variable geometry are not used. The performance of the inlet will solely rely on the aerodynamic design of the rigid geometry.

The performance of an supersonic missile inlet depends on many factors such as the extent of external and internal compression, contraction ratio, inlet start throat area, throat location, shock train length, divergence of subsonic diffuser, etc. The conventional design of such complex aerodynamic systems mainly depends on the experience of the designers who may find it difficult to take into account all of the factors needed to achieve
an optimum design. A computer aided automated optimization technique is therefore very useful.

For aerodynamic design optimization, a CFD flow field solver is usually used as the analysis tool. The optimization technique may be generally classified into two types: gradient based and non-gradient based methods. Gradient based methods usually require that the objective functions and constraints are smooth. Conventional gradient based methods generally introduce a line search to find the optimum. Gelse et al [1] and Shukla et al [2] used a CFD solver with the gradient based line search optimizer CFSQP [3] to redesign two hypersonic inlets. A similar method has also been used for the design of 3D wings with nacelles [4] for a supersonic transport. However, such approaches usually require a number of flow field computations proportional to the number of design variables to obtain the gradients. The optimization would be extremely difficult with a large number of design variables due to the limitation of the computer power. An alternative gradient approach based on control theory suggested by Jameson seems promising to overcome this difficulty [5]. This method uses the flow field governing equations as constraints and introduces the adjoint equations to make the gradient computation independent of the number of design variables. In principle, the true optimum can only be obtained with unlimited design variables. The optimum approach procedure of control theory method is much more efficient than the line search methods. Control theory has been successfully used to design 2D and 3D wings and 3D wing/body junctions [6]. However, the line search method is simpler and the application is more straightforward as long as the geometry can be modeled without using too many design variables.

The advantage of the gradient based method is that it is more efficient than the non-gradient method if the design space is smooth. The disadvantage is that it is more likely to stop at non-global local optima. The non-gradient based methods (such as Genetic Algorithms (GA), Simulated Annealing (SA) and Random Probes (RP)) are more capable of treating a design space with discontinuities and approaching the global optimum. Yamamoto, Dervieux et al. have used Genetic Algorithms to optimize the design of airfoils [7] [8]. The disadvantage of the non-gradient based methods is that they are generally more CPU-intensive. A good strategy may be to use a non-gradient method to get close to the global optimum and then use a gradient method to reach the global optimum. Aly et al. used such a method combining Simulated Annealing and a gradient optimizer for airfoil designs [9]. The optimization is switched to a gradient method from SA when it is near the possible global optimum using a criterion defined by Ogot and Alag [10].

Supersonic inlet design is a difficult task due to the complex flow physics and CPU requirements. This paper may represent the first attempt to design a supersonic missile inlet with automated optimization. For a typical steady state flow field computation (e.g., flow past a wing), a single converged computation is required for an evaluation of the objective function. For a supersonic missile inlet, it is different. Because the downstream back pressure is not known a priori, a root search iteration is needed to obtain the back pressure under which the so called critical flow can be established. A critical flow is defined as the one in which a stationary terminal shock is located downstream of the inlet throat. It is the aerodynamically desirable case for which the maximum mass flow rate and total pressure recovery can be obtained. It is very time consuming to use a CFD solver for the back pressure search. In particular, when the back pressure is close to the critical value, the shock wave system moves very slowly and a very long execution time is necessary to ensure the stability of the shock wave. In addition, it is not like the wing design for which the pressure forces are of the main interest and hence an Euler solver can be used. The total pressure recovery depends in part on the viscous losses such as the boundary layer separation induced by shock wave/turbulent boundary layer interaction. Thus, a Navier-Stokes solver with turbulence modeling is essential. Using an DEC ALPHA 2100 work station and GASP CFD code, it usually requires nearly a week to complete a back pressure search for a given inlet geometry. Thus, it is too expensive to use a CFD flow field solver alone as an analysis tool for supersonic inlet de-
sign. It would be infeasible due to our current computer power limitation if a line search gradient based optimization is implemented. The control theory optimization method in principle is possible. But it is not straightforward and more work needs to be done to develop the adjoint equations taking account of the Navier-Stokes solver and multiple shock discontinuity system inside an inlet geometry.

To develop the technology for supersonic inlet automated design optimization based on the current computation capability, we use the following multi-level design strategy.

2 Multi-Level Design Strategy

The multi-level design strategy includes two level analysis tools: an efficient, simple physics model analysis and a more sophisticated time consuming CFD Navier-Stokes analysis. The efficient simple analysis tool is incorporated into the design loop and guides the sophisticated CFD Navier-Stokes solver to verify, select and filter the final design.

As described above, it is infeasible to use a Navier-Stokes solver as the optimization analysis tool for the supersonic missile inlet design. We therefore employed a simple inlet analysis code, NIDA, in the optimization loop. NIDA was developed at United Technology Research Center (UTRC) as an inlet analysis/design tool [11]. It uses a 1D aerodynamic model with the methods of characteristics for the supersonic part upstream of the throat, and empirical correlations based on experimental data downstream of the throat for the region of the terminal shock wave/turbulent boundary layer interaction and subsonic diffuser. One evaluation of the objective function (NIDA) only needs a few seconds on the DEC ALPHA 2100 work station. NIDA is therefore suitable as an analysis tool in the optimization loop. However, due to the simple aerodynamics model used in NIDA, the accuracy of the NIDA results needs to be verified by a more complete CFD solver. In addition, NIDA does not give any details of the flow field while a Navier-Stokes computation can provide the details of the flow field.

We can also use the multi-level design strategy to push the low level optimization to the extreme using very little time. Then we can use the high level analysis tool to optimize fewer but important variables for which the low level analysis tool can not take into account. Such work is not done at this stage but is a direction for future research.

The CFD solver used in this work is the GASP code from Aerosoft Inc. which solves the Reynolds averaged Navier-Stokes equations with Chien’s $k-\varepsilon$ low Reynolds number correction turbulence model. The convective terms are evaluated using the Roe's scheme.

3 Geometry Model

A set of parameters is needed to define the missile inlet geometry so that a design space can be formed for the optimization. Fig. 1 shows the model of the missile geometry which is composed of 8 variables and 6 fixed parameters given in Table 1 and 2.

The external compression is achieved using an initial cone (o-a) followed by an isentropic compression ramp (a-b). The internal compression starts at the entrance of the duct (c-h) and followed by the curve on the center body (c-d) and the curve on the inner cowl (h-i). The connection between b and c is a straight line. The throat is located at (d-i) with the minimum area of the duct. After the throat, the duct (d-e,i-j) is called the “constant cross-section area”. This part does not really have a constant cross-section, but is slightly divergent. Its purpose is to stabilize the “shock-train” downstream of the throat. The shock-train broadens the compression wave system after the throat and compresses the flow more gradually than a single normal shock would. Therefore, the shock-train may yield smaller boundary layer separation and higher total pressure recovery than a normal shock. The duct downstream of the constant cross-section area is the divergent subsonic diffuser which further slows down the flow going into the transport duct (f-g). The transport duct has a constant cross-section area and transports the flow to the combustor with low flow energy loss. Upstream and at the throat, the curves are
Table 1: Fixed Parameters

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$D$</td>
<td>cowl diameter</td>
</tr>
<tr>
<td>2</td>
<td>$r_f$</td>
<td>centerbody radius of constant cross section region</td>
</tr>
<tr>
<td>3</td>
<td>$x_g$</td>
<td>length of inlet for computation</td>
</tr>
<tr>
<td>4</td>
<td>$x_l$</td>
<td>length of inlet for computation ($= x_g$)</td>
</tr>
<tr>
<td>5</td>
<td>$x_n$</td>
<td>length of inlet for computation ($= x_g$)</td>
</tr>
<tr>
<td>6</td>
<td>$r_m$</td>
<td>external diameter</td>
</tr>
</tbody>
</table>

Table 2: Parameters to Optimize

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$\theta_1$</td>
<td>initial cone angle</td>
</tr>
<tr>
<td>2</td>
<td>$\theta_2$</td>
<td>final cone angle</td>
</tr>
<tr>
<td>3</td>
<td>$x_d$</td>
<td>axial location of throat</td>
</tr>
<tr>
<td>4</td>
<td>$r_d$</td>
<td>radial location of throat</td>
</tr>
<tr>
<td>5</td>
<td>$x_e$</td>
<td>axial location of end of “constant” cross section</td>
</tr>
<tr>
<td>6</td>
<td>$\theta_3$</td>
<td>internal cowl lip angle</td>
</tr>
<tr>
<td>7</td>
<td>$H_{ej}$</td>
<td>height at end of constant cross section</td>
</tr>
<tr>
<td>8</td>
<td>$H_{fk}$</td>
<td>height at beginning of constant internal cross section</td>
</tr>
</tbody>
</table>

Note: All angles measured positive in counter clock wise direction
required to have continuous slopes at the connections. The duct is composed of entirely straight lines downstream of the throat. The external cowl is formed by an ellipse which is considered to have low cowl wave drag [11], and is not modified.

The external shock wave and isentropic compression waves are required to focus at the cowl lip leading edge. The inlet is required to start at Mach 2.6. In the geometry model, the throat area is therefore computed to be able to swallow the normal shock at the entrance at Mach 2.6 according to 1D aerodynamics model [12, 11]. With the initial cone angle and final external turning, the isentropic compression turning is known and the curve shape (a-b) is computed using the method of characteristics. The curve c-d is made by an ellipse if possible. If no ellipse exists to fit c-d, a cubic line is used. The advantage of an ellipse curve over a cubic line is that the ellipse is monotonic for both coordinates and slope. The cubic line can have a deflection. The curve at the inner cowl lip h-i is a cubic. To match the slopes at the point h and i, a deflection often occurs and the actual throat location therefore will be moved to somewhere before point i with smaller throat area. Because of the importance of the throat size and location which must be ensured, a slope discontinuity is allowed at the point i. Usually, such slope discontinuity is small (a few degrees) and the results show that we benefit from this treatment.

If the radius of point e, r_e, is allowed to vary, we can get the designed geometry with very sharp angle at location e-j. Since the analysis tool NIDA is based on 1D aerodynamics and smooth duct empirical correlation, the results reported from NIDA would not be reliable for such geometries and may mislead the high level accuracy analysis. We therefore only allow the z_e freely vary and require r_e to fall on the straight line between point d and f. Because NIDA computes the duct performance according to the cross section area and wall divergent angle, this treatment will not reduce the design search space for the optimization. Because we only choose the total pressure of the inlet as the object of the optimization at the current stage, the external cowl shape and therefore the cowl wave drag is kept unchanged. The external cowl shape was specified by UTRC [13]. The length of the inlet was also specified by UTRC.

3.1 Constraints

All the constraints are nonlinear inequality constraints and are continuous as suggested in [14]. Except the simple geometry constraints such as x_e > x_d, r_e > r_b, there are the following constraints required by aerodynamics and other disciplines:

1) The cross section area at the throat must be the minimum within the whole duct from the entrance to the exit for the inlet to be able to start,
2) The angle between the inner and external cowl at the cowl lip should not be less than 3 degrees for manufacturing purposes,
3) The minimum cowl wall thickness should not be less than 0.1 inch, so that the cowl wall has sufficient strength.

4 Design Space Search

Fig. 2 and Fig. 3 are two trade study results for initial cone angle and throat axial location. They show that the design space determined by NIDA and the geometry variables is non-smooth. Therefore, if we were only to rely on the gradient based optimum search procedure, it would likely to converge to a non-global local optimum. We therefore combine the non-gradient and gradient methods as the search procedure. When the non-gradient methods are not able to significantly improve the value of the objective function or are considered as being close to the possible global optimum, we switch the optimizer to CFSQP which is a gradient based procedure to obtain the optimum. CFSQP implements a quasi-Newton method to solve a nonlinear constrained optimization problem by fitting a sequence of quadratic programs.

We implement both the Random Probes and Genetic Algorithms as the non-gradient design space search procedure to initiate the optimization. As the first step, the results reported in this paper are from Random Probes. We found that the application of the Random Probes is straight-
forward and works fairly well. The GA is being tested and looks promising to efficiently find the possible global optimum. The GA results will be reported in the final paper.

5 Results and Discussion

We applied the design methodology using Random Probes with the CFSQP optimizer, starting from the original inlet design which had been obtained using NIDA but without any automated optimization. The design methodology was able to increase the total pressure recovery from the original value of 0.309 to 0.378. This is a 22% increase, and represents a very significant improvement. The optimal inlet design was also computed using GASP, which obtained a total pressure recovery of 0.375, thus confirming the improvement. The contraction ratio was increased from 3.08 to 3.628 while maintaining the capability to start the inlet at Mach 2.6.

This optimum was achieved using CFSQP with a starting point from the Random Probes with a total pressure recovery value equal to 0.357. CFSQP required 30 minutes to reach this optimum using a ALPHA DEC 2100 work station. The CFSQP optimizer with 34 random starting points and 22003 NIDA runs reached an optimum with almost the same value of the total pressure recovery. The CPU time for this multi-start CFSQP optimization was about 42.8 hours. Fig. 4 is a random probes optimization history with 20000 NIDA runs and about 42 hours CPU time. The rate of feasible solutions is 1.5% and is fairly low. A geometry re-parameterization to improve the rate of the feasible solutions will be implemented. The optimum from the random probes is 0.37 which is about 2% below the CFSQP optimum.

The basis for the multi-level design strategy is that the simple and sophisticated analysis should have similar trends. That is, when the simple analysis says one design is better than the other, the sophisticated analysis generally agrees. Such a trend is confirmed in our multi-level design by a limited number of the arbitrarily selected cases and is shown in Fig. 5.

For those inlets verified by GASP, Table 3 lists some important parameters representing the features of each inlet. The first three columns are plotted in Fig. 5. The Relative Weight Flow (RWF) is defined as the ratio of the actual captured mass flow to the inviscid captured mass flow (i.e., the mass flow in the freestream corresponding to the streamtube which intersects the cowl leading edge). The Distortion is

\[ \text{Distortion} = \frac{P_{\text{max}} - P_{\text{min}}}{P_{\text{mean}}} \]

The Contraction Ratio (CR) is

\[ \text{CR} = \frac{(\text{Captured Area})}{(\text{Throat Area})} \]

<table>
<thead>
<tr>
<th>No.</th>
<th>( P_{\text{NIDA}} )</th>
<th>( P_{\text{GASP}} )</th>
<th>RWF</th>
<th>CR</th>
<th>Distortion</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.272</td>
<td>0.311</td>
<td>0.959</td>
<td>2.979</td>
<td>0.160</td>
</tr>
<tr>
<td>2</td>
<td>0.313</td>
<td>0.330</td>
<td>0.962</td>
<td>3.187</td>
<td>0.116</td>
</tr>
<tr>
<td>3</td>
<td>0.341</td>
<td>0.338</td>
<td>0.946</td>
<td>3.295</td>
<td>0.135</td>
</tr>
<tr>
<td>4</td>
<td>0.357</td>
<td>0.364</td>
<td>0.964</td>
<td>3.409</td>
<td>0.156</td>
</tr>
<tr>
<td>5</td>
<td>0.367</td>
<td>0.374</td>
<td>0.950</td>
<td>3.570</td>
<td>0.108</td>
</tr>
<tr>
<td>6</td>
<td>0.370</td>
<td>0.369</td>
<td>0.952</td>
<td>3.581</td>
<td>0.0794</td>
</tr>
<tr>
<td>7</td>
<td>0.378</td>
<td>0.375</td>
<td>0.941</td>
<td>3.628</td>
<td>0.142</td>
</tr>
</tbody>
</table>

Table 3 indicates that the design with higher contraction ratio generally has the higher total pressure recovery. This agrees with inlet design principles. All the designs in Table 3 with total pressure recovery higher than 0.341 have some slope discontinuities (less than 3 deg) at the throat locations on the inner cowl surface. If we did not relax the slope condition at the throat, we would lose an important part of the design space. It is seen that the exit distortion for all the inlets are quite low. NIDA and GASP prediction of the total pressure recovery trend for case 5 and 6 have a difference about 1%. We are currently investigating whether this difference is due to the truncation error of the GASP computation or represents an uncertainty range between the two analysis tools.

The following is the Navier-Stokes flow field analysis for the optimum design, case 7. The computational domain is axi-symmetric and is composed of two zones, inner and outer zone.
Table 4: Geometry Parameters of Case 7

<table>
<thead>
<tr>
<th>No.</th>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fixed</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>$D$</td>
<td>15.00 in</td>
</tr>
<tr>
<td>2</td>
<td>$	au_f$</td>
<td>9.10 in</td>
</tr>
<tr>
<td>3</td>
<td>$x_f$</td>
<td>45.60 in</td>
</tr>
<tr>
<td>4</td>
<td>$x_m$</td>
<td>45.60 in</td>
</tr>
<tr>
<td>5</td>
<td>$x_n$</td>
<td>45.60 in</td>
</tr>
<tr>
<td>6</td>
<td>$r_m$</td>
<td>10.60 in</td>
</tr>
<tr>
<td>Variable</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>$\theta_1$</td>
<td>12.00 deg</td>
</tr>
<tr>
<td>2</td>
<td>$\theta_2$</td>
<td>24.00 deg</td>
</tr>
<tr>
<td>3</td>
<td>$x_d$</td>
<td>24.48 in</td>
</tr>
<tr>
<td>4</td>
<td>$r_d$</td>
<td>6.78 in</td>
</tr>
<tr>
<td>5</td>
<td>$x_c$</td>
<td>41.17 in</td>
</tr>
<tr>
<td>6</td>
<td>$\theta_3$</td>
<td>1.16 deg</td>
</tr>
<tr>
<td>7</td>
<td>$H_{ej}$</td>
<td>1.07 in</td>
</tr>
<tr>
<td>8</td>
<td>$H_{fk}$</td>
<td>1.02 in</td>
</tr>
</tbody>
</table>

Table 5: Flow Conditions

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M_{\infty}$</td>
<td>4.0</td>
</tr>
<tr>
<td>$P_{\infty}$</td>
<td>$6.817 \times 10^5$</td>
</tr>
<tr>
<td>$T_{\infty}$</td>
<td>914.76$^0$K</td>
</tr>
<tr>
<td>Attack Angle</td>
<td>0.0$^0$</td>
</tr>
<tr>
<td>Reynolds No.</td>
<td>$5.959 \times 10^6$ /m</td>
</tr>
<tr>
<td>$\delta_{in}$</td>
<td>0.013 m</td>
</tr>
<tr>
<td>$\delta_{out}$</td>
<td>0.0054 m</td>
</tr>
</tbody>
</table>

Table 6: Inner Grid Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>grid point number in X</td>
<td>133</td>
</tr>
<tr>
<td>grid point number in R</td>
<td>121</td>
</tr>
<tr>
<td>$\Delta x/\delta_{in}$</td>
<td>0.77</td>
</tr>
<tr>
<td>$\Delta r_f^{+}_{\max,centerbody}$</td>
<td>7.405818</td>
</tr>
<tr>
<td>$\Delta r_f^{+}_{ave,centerbody}$</td>
<td>2.366</td>
</tr>
<tr>
<td>$\Delta r_f^{+}_{\max,cowl}$</td>
<td>3.451</td>
</tr>
<tr>
<td>$\Delta r_f^{+}_{ave,cowl}$</td>
<td>1.838</td>
</tr>
</tbody>
</table>

Table 7: Outer Grid Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>grid point number in X</td>
<td>133</td>
</tr>
<tr>
<td>grid point number in R</td>
<td>81</td>
</tr>
<tr>
<td>$\Delta x/\delta_{out}$</td>
<td>1.85</td>
</tr>
<tr>
<td>$\Delta r_f^{+}$</td>
<td>0.065</td>
</tr>
<tr>
<td>$\Delta r_f^{+}_{ave}$</td>
<td>0.0167</td>
</tr>
</tbody>
</table>

Fig. 6 is the center plane of the mesh used for this computation. The inner zone is from the center body leading edge through the internal channel to the exit of the inlet. The outer zone is the flow field from the incoming free stream through the external surface of the cowl. Tables 4-8 summarize the geometry, flow and mesh parameters for the computation. In the tables, $\delta_{in}$ is the boundary layer thickness computed at the entrance of the inlet on the center body. $\delta_{out}$ is the boundary layer thickness computed at the streamwise midway location of the external cowl.

At the critical condition of the inlet, the exit of the inlet is subsonic and therefore the exit back pressure is a necessary boundary condi-

Table 8: Grid points within the boundary layers

<table>
<thead>
<tr>
<th>wall boundary layer</th>
<th>number of grid points</th>
</tr>
</thead>
<tbody>
<tr>
<td>wall of the center body</td>
<td>57</td>
</tr>
<tr>
<td>inner wall of the cowl</td>
<td>50</td>
</tr>
<tr>
<td>inner wall of the cowl</td>
<td>40</td>
</tr>
</tbody>
</table>
tion for the computation. A series of computations are needed to search the back pressure
which yields the critical flow for the inlet. The critical back pressure tolerance is less than 1%
(i.e., a 1% increase in the converged value for the back pressure will cause the inlet to unstart).
The total pressure recovery coefficients based on mass weighted average, area weighted average
and stream thrust weighted average are the following:

\[ P_{t \text{ mass average}} = \frac{1}{P_{t \infty}} \frac{\int \rho u P \, ds}{\int \rho u ds} = 0.3748 \]

\[ P_{t \text{ area average}} = \frac{1}{P_{t \infty}} \frac{\int P \, ds}{\int ds} = 0.3741 \]

\[ P_{t \text{ stream thrust}} = \frac{1}{P_{t \infty}} \frac{\int (\rho u^2 + p) P \, ds}{\int (\rho u^2 + p) \, ds} = 0.3740 \]

where \( s \) is the area in radial direction of the axisymmetric computational domain.

Fig. 7 shows the Mach number contours of the flow field. The oblique shock wave and isentropic compression waves generally focus around the leading edge of the cowl as intended. The
details of the Mach number contours in the throat area are shown in Fig. 8. There is a shock
generated by the cowl leading edge which intersects the centerbody. The cowl shock does not
cause separation on the centerbody. The reflecting shock from the centerbody intersects with an
other oblique shock wave from the cowl surface and forms a wave system which looks like a normal shock. It is actually not a normal shock and the flow after the intersection of the two oblique shocks is still supersonic. The light blue sonic line shows that flow after the oblique shock wave intersection gradually becomes subsonic. This
maintains a high total pressure for the flow in that region. Fig. 9 is the total pressure contours and demonstrates that the gradual change of the supersonic flow to subsonic benefits the
total pressure loss with a region protruding in green color while the flow separations induced by
the shock wave/turbulent boundary layer interaction cause serious total pressure loss. The low
energy flow from the separation strongly mixes with the main flow due to the turbulence and make the downstream total pressure field quite uniform. This is desirable to reduce the distortion of the flow going into the combustor. The
turbulent intensity is also enhanced by the terminal shock wave/turbulent boundary layer interaction and reaches the maximum in the separation region (not shown).

Fig. 10 displays the flow separations induced by the shock wave/turbulent boundary layer interaction for different inlets with total pressure recovery values from low to high. They are case 2 (a), case 4 (b), case 6 (c) and the optimum case 7 (d) respectively in Table 3. Because the channels have mild divergence, these separation regions are necessary to form the second throat and support the strong compression wave (the intersection of the two oblique shocks) compressing the supersonic flow to subsonic. Fig. 10 clearly shows the geometry evolution with the optimization. In most of the cases, there is a large separation on the inner cowl surface and a small separation on the centerbody surface. The optimum case has the smallest separation region and throat area with a larger separation on the centerbody than on the inner cowl surface. The flow separation is mainly determined by the shock wave intensity. Fig. 11 is the pressure contours of the flow field in the throat region and presents the shock wave systems inside the inlets. The optimum case has the weakest intensity of the terminal shock train due to the highest compression efficiency upstream of the throat.

In Table 3, it is evident that the relative weight flow is typically 0.95. Theoretically, the relative weight flow should be equal to one if the shock wave focuses on the cowl lip as the design requires. The reasons for this difference are:
1) NIDA uses a 1D aerodynamics model to determine the wave focus point without taking into account the boundary layer effect. When the flow field is computed using the GASP Navier-Stokes solver, the pressure after the oblique shock from the centerbody leading will be slightly increased due to the centerbody boundary layer displacement. The shock and isentropic compression waves are hence slightly pushed away and some flow is spilt.
2) Even if the oblique shock wave from the central body leading edge can reach the cowl lip, the numerical diffusion broadens the shock wave at the cowl lip and makes the shock wave look like a strong compression wave which will also make
some flow to be split. The diffused shock wave at the cowl lip can be seen from Fig. 7, 8 and 11. Such shock wave diffusion may be reduced by aligning the mesh with the shock wave and mesh adaptation.

It is interesting to compare the case 5 and 7 which have nearly the same total pressure recovery evaluated by GASP, but quite different geometries. The significant difference is the inner cowl lip angles and the throat locations. The duct divergence after the throat is also quite different. Table 9 is the geometry variables for case 5. Fig. 12 is the total pressure recovery variation along the inlet axial axis for these two cases. It is seen that case 7 has the stronger cowl lip shock loss since it has smaller inner cowl angle (see Fig. 11). Because case 7 has stronger cowl lip shock and smaller throat area, the Mach number before the terminal shock is smaller than that of case 5, and thus the terminal shock intensity of case 7 is weaker (see Fig. 10 and 11). Therefore, case 7 has smaller total pressure loss due to the terminal shock wave/turbulent boundary layer interaction. However, after the terminal shock, case 7 has stronger pressure gradient and more rapidly turbulent boundary layer development, hence has larger total pressure drop. Even though these two cases have about the same total pressure recovery, case 5 has much lower distortion. This comparison means that, for a supersonic inlet, the same total pressure recovery may be obtained by controlling different geometry parameters. This would make the design more difficult if we did not have an automated optimization procedure.

From these two cases, we have found that the terminal shock wave/turbulent boundary layer interactions cause the sharp local total pressure drops and also enhance the total pressure loss after it. Therefore, there may be large potential to improve the total pressure recovery if we can further weaken the terminal shock wave/turbulent boundary layer interaction.

Fig. 13 is the velocity profiles of case 7 in the region of the exit. The flow is shown to be quite uniform.

6 Flow Field Computation

The flow field computation using GASP starts with the supercritical flow which is supersonic from the entrance to the exit. When a back pressure is imposed at the exit, a terminal shock appears with this separation region. The process to reach the steady state is the motion process of this shock-separation (SS) system. When the back pressure is substantially higher than the critical back pressure, the propagation speed of the SS system is fast and the shock is quickly pushed out of the entrance as the subcritical flow [13]. When the back pressure is approaching the critical value, the propagation speed of the SS system is very slow. This makes the computation particularly CPU time intensive. When a critical flow is considered to be obtained, the solution will be run again for another substantially long time to check if the SS system is displaced. This will tell if the SS system is still slowly moving or finally stable. Using GASP code, typically 24 hours CPU time is needed to achieve a converged flowfield on a DEC ALPHA 2100 work station computer. It needs about 5 days to search the back pressure value for the critical flow with a tolerance 1%.

Finally as a reminder, the design quantities we have obtained so far is still at the theoretical stage. The final design needs to be confirmed by the experiment.

7 Conclusions

A multi-level design strategy for supersonic missile inlet design is developed. The multi-level de-
sign strategy combines an efficient simple physical model analysis tool and a sophisticated CFD Navier-Stokes analysis tool. The efficient simple analysis tool is incorporated into the optimization loop and the CFD Navier-Stokes analysis tool is used to verify, select and filter the final design. Because of the non-smooth design space, the optimization is composed of a non-gradient method and a gradient method. The non-gradient method being used is Random Probes, and a Genetic Algorithm is being tested. The gradient based optimizer is CFSQP, which uses quasi-Newton method to make the line search. A geometry model for the supersonic missile inlet is developed.

The simple and sophisticated analysis tools generally have the same trend to evaluate the measure of merit. More than 20% improvement of the inlet total pressure recovery has been obtained with the contraction ratio increased from 3.08 to 3.628. A detailed flow field analysis is presented and shows that the main total pressure loss occurs in the region of the cowl lip shock, terminal shock, and in the region downstream of the separation induced by the terminal shock wave/turbulent boundary layer interaction due to the spread of the low flow energy and turbulent boundary layer development.

8 Acknowledgments

This research is part of the HPCD (Hypercomputing and Design) project based at Rutgers University. The HPCD project is supported by the Advanced Research Projects Agency of the Department of Defense through contract ARPA-DABT 63-93-C-0064 monitored by Dr. Bob Lucas. We appreciate the helpful discussions with David Sobel at United Technologies Research Center and Madara Ogot in Dept. of Mechanical and Aerospace Engineering, Rutgers University. The contents of this paper do not necessarily reflect the position of the United States government and official endorsement should not be inferred.

References


Figure 1: Supersonic missile inlet geometry model

Figure 2: Trade study of total pressure recovery versus the initial cone angle
Figure 3: Trade study of total pressure recovery versus the throat axial location
20001 calls to runNida made during random probe
42.83 CPU hours required, 7.71 seconds used per call

<table>
<thead>
<tr>
<th>evaluable</th>
<th>feasible</th>
<th>percent evaluations</th>
<th>percent of time</th>
<th>seconds per evaluation</th>
</tr>
</thead>
<tbody>
<tr>
<td>No</td>
<td>XXX</td>
<td>91.8</td>
<td>93.7</td>
<td>7.87</td>
</tr>
<tr>
<td>Yes</td>
<td>No</td>
<td>6.7</td>
<td>5.1</td>
<td>5.90</td>
</tr>
<tr>
<td>Yes</td>
<td>Yes</td>
<td>1.5</td>
<td>1.1</td>
<td>5.92</td>
</tr>
</tbody>
</table>

Figure 4: Random probes optimization history
Figure 5: Verification of low and high level analysis

Figure 6: Mesh for the flow field computation of the missile inlet
Figure 7: Mach number contours of the inlet flow field

Figure 8: Mach number contours of the inlet flow field in the throat area
Figure 9: Total pressure contours of the inlet flow field in the throat area
Figure 10: Streamlines of the inlet flow fields in the throat area, a: case 2, b: case 4, c: case 6, d: case 7
Figure 11: Pressure contours of the inlet flow fields in the throat area, a: case 2, b: case 4, c: case 6, d: case 7
Figure 12: Total pressure recovery distribution along the axis

Figure 13: Velocity profiles in the region of the exit