COMPUTATIONAL INVESTIGATION OF FLOW THROUGH
A LOUVERED INLET CONFIGURATION

Ismail H. Tuncer
Middle East Technical University
Ankara, Turkey

Max F. Platzer†
Naval Postgraduate School
Monterey, California

ABSTRACT
Viscous, subsonic flows through a flush louvered inlet configuration without inlet vanes are investigated using the OVERFLOW Navier-Stokes solver with overset grids. The variation of the mass flow rate through the engine intake as a function of intake pressure is assessed. The computed results show that the flow distortion and the pressure loss in the plenum chamber are significant, yet a sufficient amount of mass flow rate may be obtained at the expense of total pressure recovery.

INTRODUCTION
The aerodynamic analysis of subsonic lifting body concepts for future unmanned air vehicles and missiles has been an active research field in recent years. A lifting body, as shown in Figure 1, combines three emerging technologies, namely lifting body, thrust vector control, and louvered inlet (Figure 2). The flush louvered inlet configuration requires the flow to turn 360 degrees in the plenum chamber before entering the engine. The pressure losses and flow distortions experienced in the plenum chamber are serious concerns. In the analysis of the flow the boundary layer build-up on the underside of the missile body prior to reaching the louvered inlet, and the three-dimensional flow in the plenum chamber are to be accounted for. Engine tests simulating this condition are currently not possible. It is, therefore, highly desirable to compute viscous, compressible flows for a typical flush louvered inlet configuration.

In the design of louvered inlets, inlet vanes are required to achieve good total pressure recovery. However, in this analysis the vanes are not yet incorporated to simplify the computations and to obtain a first estimate of the flow characteristics.

In this work, a numerical study of flow through a louvered inlet configuration is performed. Preliminary solutions of the highly complex flowfield are obtained using the OVERFLOW Navier-Stokes solver with overset grids. The computed results show that the flow distortion and the losses at the engine intake are significant, yet a sufficient amount of mass flow rate may be obtained.
NUMERICAL METHOD

In the computation of subsonic flows through a louvered inlet it is important to account for the boundary layer built-up on the underside of the missile body prior to reaching the louvered inlet as well as to compute the 3-D flow in the plenum chamber. Therefore, a compressible Navier-Stokes solver, OVERFLOW (version 1.7v), which has been developed at the NASA-Ames Research Center, is used to compute the flow through a louvered inlet configuration.

OVERFLOW is a compressible, thin-layer, Reynolds-averaged Navier-Stokes solver. It accommodates computational domains discretized with overset sub-grids, which are preprocessed with the PEGSUS code. Overset subgrids create holes in each other and the corresponding intergrid hole boundaries are formed. At the intergrid boundaries, the flow variables are interpolated from the neighboring grids. The PEGSUS code determines the hole boundaries created by the overset grids, and supplies the intergrid interpolation stencils and the corresponding interpolation weights. The intergrid interpolation stencils and weights are input to the OVERFLOW solver. In the past, we have successfully computed viscous flowfields over missile configurations using the OVERFLOW solver.

The OVERFLOW solver has several discretization and time integration schemes available. In this study, the 3-factor diagonal scheme with central differenced convective flux terms, which is the fastest and the most robust, is used. The central difference smoothing coefficient is gradually reduced to 0.10 from the starting value of 0.25. Local time stepping is used to obtain the steady state solution. The criterion for convergence to the steady state was based on the maximum and the L2 norm of the residuals.

The flowfield is assumed to be fully turbulent. Solutions with the Baldwin-Lomax algebraic eddy viscosity model with Degani-Schiff cutoff, and the one-equation Baldwin-Barth turbulence models are obtained, and compared with each other.

The computed flowfields are analyzed in terms of the mass flow rate through the engine intake, $\dot{m}$, the area averaged total pressure recovery ratio, $\mathcal{T}_{0\text{intake}} / P_{0\infty}$, and the area averaged Mach number, $M_{0\text{intake}}$, at the engine intake boundary. The area averaged total pressure at the engine intake is evaluated by

$$\mathcal{T}_{0\text{intake}} = \sum_{n} \frac{P_n [1 + \frac{\rho_{n}}{\rho_{\infty}} \frac{2 \gamma - 1}{2 \gamma} V_n^2]^{\frac{\gamma}{\gamma-1}} A_n}{\sum A_n}$$

where $n$ refers to a grid point on the engine intake surface and $A_n$ is the surface area associated with the grid point. Mass flow rate at the engine intake surface, which is normal to the $x$ axis, is similarly evaluated by

$$\dot{m} = \sum_{n} \rho_n u_x A_n$$

Boundary Conditions

The no-slip boundary condition and the one-dimensional Riemann invariant extrapolation are applied at solid surfaces and farfield boundaries, respectively. At the symmetry plane, symmetry in the cross-flow $y$ direction is imposed. A constant pressure based on the $P/P_{\infty}$ ratio is applied at the engine intake surface for the parametric study.

Computational Domain

The flowfield is assumed to be symmetric with respect to the mid-plane of the inlet configuration, and only half of the flow domain is discretized. The computational flow domain consists of the external flow region underside of the missile body, the inlet, the plenum chamber and the engine intake. The plenum chamber is about 15 in long, 15 in wide and 25 in deep. Since inlet vanes are not modeled in this preliminary study, the inlet is discretized as a rectangular opening. The computational domain is discretized using overset grids shown in Figure 3. In the cross-sectional views in Figure 3, the grid coordinates are projected onto the sliced planes. The overset grid system consists of 4 subgrids, which discretize the external flow region (66 x 31 x 51), the inlet passage (59 x 41 x 33), the plenum chamber (81 x 71 x 91) and the engine intake with the bellmouth (99 x 41 x 41). The total number of grid points is about 9 x 10^5.

In order to resolve the boundary layer flows at the wall boundaries the subgrids are designed to provide as much high resolution as the computing resources would permit. The first grid spacing on the wall boundaries varies from 1 x 10^-3 in on the outer wall to 1 x 10^-2 in on the plenum chamber and the bellmouth intake walls.

The subgrid for the inlet passage is overset onto the neighboring subgrids in the plenum chamber and the external flow regions. In the middle part of its outer boundaries which covers the wall boundaries of the inlet passage, the wall boundary conditions are imposed. In the remaining part of its outer boundaries which oversets onto the neighboring subgrids, the intergrid boundary conditions are successfully applied.
Fig. 3  Computational domain discretized with overset grids (Every other grid point is plotted)
Table 1: The effect of intake pressure and the turbulence models on the intake flow.

**RESULTS AND DISCUSSION**

The flowfields through the flush-louvered inlet were computed at three different engine intake pressure ratios using the Baldwin-Lomax and the Baldwin-Barth turbulence models. All the computations were carried out on a Cray-J916/4. A typical converged solution took about 6000 time steps in approximately 60 CPU hours.

All the viscous flowfields were computed at $M_\infty = 0.8$, $T_\infty = 467^\circ R$ and $Re = 4 \cdot 10^6$ per inch assuming the flow fully turbulent. The Baldwin-Lomax algebraic turbulence model was first applied. However, the model computes the normal distance to the wall boundaries separately for each wall surface, and fails at the corner regions of closed domains like the plenum chamber. Therefore, the flow is described to be turbulent for the front and the back surfaces of the plenum chamber in the $x$ direction. Thus, the flow is computed mostly laminar along the side walls. Having no such limitation, the Baldwin-Barth turbulence model computes the eddy viscosities based on the closest distance from any wall, and the flow in the plenum chamber is computed fully turbulent. The typical convergence rates for the solutions in the plenum chamber are shown in Figure 4. The solution with the Baldwin-Barth model converges significantly faster than that of the Baldwin-Lomax, which is attributed to the better performance of the turbulence model. The computed flow conditions and the results are summarized in Table 1.

The flow computed at $P_{\text{intake}}/P_\infty = 0.80$ with the Baldwin-Barth model is shown in Figures 5 and 6. The Mach number distribution and velocity vectors at the cross-sectional planes through the engine intake and the particle traces through the inlet reveal the highly complex nature of the flow in the plenum chamber. As the incoming flow hits the back end of the plenum chamber, vortical structures and recirculating flows are formed. Yet, a significant percentage of the flowfield in the chamber is covered with low-velocity, dead flow zones. The total pressure recovery distribution is shown in Figure 7. The flow which re-
Fig. 5 Computed flowfield by Baldwin-Barth
turbulence model, $P_{\text{intake}}/P_{\infty} = 0.80$

Fig. 6 Particle traces through the louvered inlet

Fig. 7 Total pressure recovery, $P_0/P_{0,\infty}$,
$P_{\text{intake}}/P_{\infty} = 0.80$
sults in an average 60% total pressure recovery at the engine intake produces a mass flow rate of 13.11 lbs/sec.

The same flowfield computed with the Baldwin-Lomax turbulent model predicts a higher total pressure recovery and significantly higher average Mach number at the engine intake (Table 1). Such a result is attributed to the mostly laminar flow with low eddy viscosity values near the side walls of the plenum chamber.

The flows computed at higher reduced pressure ratios at the engine intake, $P_{\text{intake}}/P_{\infty}$, produce higher mass flow rates at slightly increased total pressure losses (Table 1). Yet the intake Mach number increases significantly, and supersonic velocities are observed in the flow.

Figure 8 and 9 show the flowfield computed at $P_{\text{intake}}/P_{\infty} = 0.6$ using both the Baldwin-Barth and Baldwin-Lomax models. Except the higher Mach number values, the flowfield predicted by the Baldwin-Barth model is similar to that of $P_{\text{intake}}/P_{\infty} = 0.8$ case. As observed, at the inlet expansion, and at the intake blemouth the Mach number approaches sonic values. However, in the Baldwin-Lomax solution, the flowfield appears to be highly distorted with large flow gradients, and secondary recirculating flows at the inlet blocks the incoming flow. Significantly higher Mach number values are also observed at the engine intake. It is concluded that the Baldwin-Lomax model where the flow is computed mostly laminar introduces artificial instabilities, and does not perform well in the plenum chamber.
CONCLUSIONS

In this work, we have obtained preliminary solutions for a highly complex flow through a flush louvered inlet configuration using the OVERFLOW Navier-Stokes solver with overset grids. The preliminary solutions show that the flow distortion in the plenum chamber and the pressure losses are significant, yet a sufficient amount of mass flow rate at the engine intake may be obtained. The Baldwin-Lomax turbulence model is found to be unsuitable for the solution of such flows. However, considerably more work is required in terms of various flow parameters and finer grid resolutions before the flowfield is assessed with adequate accuracy. It is also necessary to model the inlet vanes and look for modifications of the inlet configuration in order to alleviate the losses and increase the efficiency of a louvered inlet.

ACKNOWLEDGEMENT

This investigation was supported by the Naval Air Warfare Center, China Lake, CA.

REFERENCES


