Passive Flow Modification Over the Supersonic and the Hypersonic Air-Intake System Using Bleed

The air intake should be operated at design conditions to achieve a high total pressure recovery and optimum mass capture ratio. The current research focuses on the numerical simulation of the supersonic and hypersonic air inlet and its starting and unstarting characteristics. 2D RANS equation for supersonic and hypersonic intake has been solved using the k-ωSST turbulence model. The in-house code and the algorithm based on the RANS equation have also been validated in due process and used for subsequent simulations. The sudden drop in mass capture ratio indicates the unstart condition of the intake. The presence of a bleed section has a commendable effect on the performance parameter of the air intake. A separation bubble was observed at the intake's entrance during the off-design conditions, resulting in performance losses. Four different bleed sections ranging in size from 1.6mm to 8.6mm were used, and simulations with bleed were run for different Mach numbers ranging from 3 to 8. The optimum bleed size of 3mm has been found quite effective in modifying Total pressure recovery within the optimum mass flow rate over the wide range of Mach numbers.

Keywords: Hypersonic inlet, Pressure recovery, Mass capture ratio, shockwave, starting and unstarting of inlet, Bleed geometry, Total pressure ratio

1. INTRODUCTION

Hypersonic inlet efficiency determines the overall engine performance. The performance of the hypersonic inlet is generally determined by two significant factors: total pressure recovery and mass capture ratio [1-4]. The inlet should provide better performance for its all-operating condition. Because of the high operating range of Mach numbers, the mixed compression type of inlet is the most convenient [5]. Flow field for the mixed compression type of inlet is shown in Figure 1.

![Figure 1. A mixed compression type of supersonic inlet [2]](image)

Performance losses of intake affect the overall engine efficiency. Some of the causes of low inlet efficiency are flow spillage, different operating flight conditions, improper combustion, and multiple issues related to backpressure [6, 7]. A high rate of spillage and discharging of shocks leads to a reduction in total pressure recovery and higher rates of flow distortion at the intake outlet [8, 9]. Shockwave interaction with thin boundary layers increases the thickness of the local boundary layers at the region of internal compression, which is the prime cause of shock discharge, leading to an unstart [10, 11]. The prior solution to prevent the unstart condition is to provide the bleed flow section inside the intake [12]. The hypersonic flow contains high total temperature and enthalpy, which leads to structural complexity and cooling issues [13]. Predicting intake unstart reduces the occurrence, which is critical for hypersonic intake design. The researchers have performed numerical and experimental analyses to improve the unstart characteristics.

Reinartz et al. [14] reported the effect of different isolator geometry on overall inlet efficiency. Eight different isolator geometries of various lengths were tested experimentally to find that the increase in the isolator length reduces the pressure sensitivity of the intake. Schmitz and Bissinger [15] investigated two fixed geometry hypersonic inlets to find the starting and unstarting operations. To study the inlet starting characteristics, Yi Wang et al. [16] investigated and compared three inlet designs with varied internal contraction ratios.

Reinartz & Behr [17] performed extensive tests and analyses to improve a hypersonic intake's performance with sidewall compression, resulting in refinements in the MFR and starting characteristics. K. Raja Sekar et al. [18] investigated the two-dimensional rectangular
hypersonic mixed compression inlet at Mach 5. In their experiment, the throttling is simulated by inserting a wedge plug of variable heights at the exit of the isolator section and simulating different throttling ratios ranging from 0 to 0.7 in stages of 0.1. Lee et al. [19] provide much-needed insight into the boundary layer instability of the hypersonic cases and the experimental methods to detect them. These modes have also been accounted for in the current study. Unsteadiness in the hypersonic flow studied over the double wedge structure is of three types: vibration mode, oscillation mode, and pulsation mode [20]. Their separation depends on the separation zone's size and the shock wave's location. Janarthanam and Babu [21] carried out the 3D analysis of the mixed compression type of hypersonic inlet, and our selection of the turbulence model is largely based on this. To control the hypersonic boundary layer transition, active techniques like Plasma actuated flow control [22] and passive techniques like carbon fiber reinforced ceramic surfaces [23] have been utilized. Starting characteristics of hypersonic intake have been carried out by Barber et al. [24]. They studied starting characteristics and causes of unstarting related to the viscous effects and different turbulence models. A lot of motivation has been taken from this paper to develop the methodology for the CFD simulation of the present paper.

Many research papers have been published on air-intake optimization, and many have also used bleeds. Though most researchers have worked on buzz and other instabilities, only a few are there on the sizing and positioning of the bleed. It has also been found that limited work has been carried out on the effect of the bleed geometry on the supersonic and hypersonic flow modification. So a gap exists where none of the research could assert the exact bleed geometry and its location on the air intake for a range of Mach numbers ranging from supersonic to hypersonic, where it could provide optimum total pressure ratio and mass flow rate.

Moreover, the selection of bleed geometry depends on the intake design and the operating Mach. So, in this research paper, we have tried to bridge that gap, and through numerical analysis, we have provided the flow physics of the intake at start and unstart conditions at different Mach numbers.

This paper mainly focuses on the passive methods to improve their operating efficiency. The main application of this paper and the documented research work is that we have narrowed down the bleed geometry options, and we also identified the optimum dimension among them that can be used for both the Supersonic and the hypersonic flow regime with the help of bleed geometry.

The novelty of this paper lies in the fact that any high-speed air vehicle (like fighter jets) can operate over both flow regimes as per the mission profile, with minor modifications in the bleed geometry. With the advances in research in Ramjet and Scramjet propulsion systems, this kind of study provides great insight into developing the air-intake system of fighter jets, guided missiles, or other high-speed air-breathing engines.

We have modified the intake geometry given by Emami and Trexler [25]. We simulated the same air-intake model, firstly, without bleed and then with the bleed, to obtain the modification in the optimum performance parameter as the function of the bleed geometry. The unique contribution of this research is that it presents the internal flow physics, performance parameter characteristics, and optimum bleed geometry selection over a wide range of Mach numbers ranging from 2.5 to 8. It forms the basis of further research and in-house code development to evaluate the different back pressure dynamics.

The sole purpose of this paper is to obtain the effect of bleed on a particular intake and then optimize the bleed size for the given intake geometry. Over fifty bleed sizes have been analyzed, after which a set of 4 slotted bleeds have been considered in the present research paper. Unsteady phenomena and buzz have yet to be covered in this paper as it will make this paper extremely lengthy.

2. GEOMETRY AND GRID

The geometry of the inlet used for experimental investigation by Emami and Trexler [25] is shown in Figure 2. The geometry has a mixed compression type of inlet with one compression ramp. The bleed case's geometry has been modified (see Fig. 3).

The four different bleed cases have been chosen as 1.3mm, 3mm, 7.1mm, and 8.6mm. The length of the model is 338 mm.
chosen for the isolator. The height of the far field from
the base is two times the length of the upstream sym–
metry line of length ‘L’. The Span of the computational
domain is also 2L. Such a square computational domain
provides better control of the grid generation.

Figure 4. Named selections for the simulation

Figure 5. Grid system

The grid system is selected after extensive grid
refinement exercises (see Fig. 6). We chose a fine mesh
over a superfine mesh to reduce the computational time
for the parametric studies. Figure 7 compares the lower
ramp wall pressure variation for experimental [25] and
computational reference data [2]. A number of cells
rounded off to the order of 10^5 in different mesh types
are shown in Table 1.

Table 1. Grid Types for grid independence study

<table>
<thead>
<tr>
<th>Mesh types</th>
<th>No. of cells (Rounded off to nearest lacs)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>200000</td>
</tr>
<tr>
<td>Medium</td>
<td>300000</td>
</tr>
<tr>
<td>Fine</td>
<td>400000</td>
</tr>
<tr>
<td>Superfine</td>
<td>500000</td>
</tr>
</tbody>
</table>

Figure 6. Grid independence check

Figure 7. Numerical validation check with reference data [2]

The simulation was performed based on the data
reported by Saha et al.[2], and different Mach numbers
have been imposed by varying the total pressure. Two air
outlets were created, one for the inlet and another for the
ambient outlet. To implement the bleed, an additional
outlet was created (Fig.4). In the current solution; the
simulations were performed in a steady-state, density-
based implicit Reynolds-averaged Navier-Stokes equa–
tions (RANS) solver with a K-ω SST turbulence model.
2nd order spatially accurate roe-flux difference splitting
scheme for spatial discretization and a 2nd order implicit
Euler Scheme for temporal discretization have been used.
The solver is verified using benchmark data generated
from a closed-form analytical model [26-27]. Further–
more, numerical results are validated using the
experimental data given by Emami and Trexlar [25] and
computational data given by Saha and Chakraborty [2].

Since no experimental work has been undertaken
supporting the present research work, we have validated
the work using the published paper by Milicev [28] and
Damljanović et al. [29]. The off-design performance of
the model in the wind tunnel has been studied based on
the work of Damljanović et al. [30].

Table 2. Solver set-up

<table>
<thead>
<tr>
<th>Scheme/parameters</th>
<th>Type/quantities</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solver precision</td>
<td>Double precision</td>
</tr>
<tr>
<td>Solver type</td>
<td>Density-based implicit, steady state</td>
</tr>
<tr>
<td>RANS</td>
<td>k-ω SST (2 PDE model)</td>
</tr>
<tr>
<td>Inviscid flux scheme</td>
<td>Roe flux-difference splitting scheme</td>
</tr>
<tr>
<td>Spatial discretization</td>
<td>Second-order upwind scheme</td>
</tr>
<tr>
<td>Gradient evaluation</td>
<td>Least squares cell-based</td>
</tr>
</tbody>
</table>

3. GOVERNING EQUATIONS

For the present simulation, the k-ω shear stress model
has been used. ‘k’ represents the turbulent kinetic
energy, and ‘ω’ represents the specific dissipation rate.
Turbulence viscosity is obtained as a function of
both ‘k’ and ‘ω’.
\[ \mu_t = \frac{(\rho u k)}{\varepsilon} \]  

Turbulent kinetic energy (k) equation:

\[ \frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \frac{\mu + \mu_t}{\rho} \left( \frac{\partial k}{\partial x_j} + \frac{\partial k}{\partial x_i} \right) \right] + \mu_t \nabla^2 k + G_k - \rho e \]  

Specific dissipation rate (\( \omega \)) equation:

\[ \frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial \omega}{\partial x_j} \right) \right] - \beta \rho \omega^2 \]  

4. NUMERICAL VALIDATION STUDY

4.1 Surface pressure distribution

The wall Pressure distribution across a lower ramp compared with the experimental work done by Emami and Trexler [25] and the 3D computational work done by Saha and Chakraborty [2], as shown in Figure 7. The pressure and distance are shown in non-dimensional terms. In the P/P_inf, P_inf is the free stream pressure, and in x/h, h is the intake throat height. The results show differences near the cowl shock impingement point, and overall good agreements are demonstrated with the experimental. However, certain variation at x/h = 26 and x/h = 28 can be attributed to the fact that this is a 2D simulation, whereas the others are experimental and the 3D simulation. Still, we have managed to obtain a good match with both the papers when we consider the Total pressure ratio and the Mass capture ratio.

4.2 Mass capture ratio & total pressure recovery

The inlet should perform optimally in all operating conditions. The mass capture ratio (MCR) and total pressure ratio (TPR) are essential factors in the overall efficiency of the inlet [1,2]. Figure 8 compares the mass capture ratio (MCR) for various free-stream Mach numbers. The findings are compared with the same set of reference data [2, 25]. From the graph, figure 8, we can observe that as the Mach number increases, the MCR also increases, indicating low spillage for higher values of Mach numbers (4 < M < 8). A good estimation of the numerical data can be seen here with the experimental data.

![Figure 8. Comparison of mass capture ratio for different free-stream Mach numbers](image-url)
This demonstrates the significance of the oblique shock wave caused by the first ramp. The first oblique shock must be attached to the cowl to achieve the highest MCR. The maximum MCR value was obtained at Mach 8, closely corresponding to experimental data [25].

TPR given in figure 9 also shows quite a good agreement between the reference data [2, 25] and the present 2D simulation.

![Figure 9. Comparison of total pressure ratio for different free-stream Mach numbers](image)

Hence, from figure 7,8,9 we can say that the in-house algorithm adopted to solve the supersonic and hypersonic case has well captured the flow physics.

5. RESULTS AND DISCUSSION

5.1 Air-intake start & unstart issues with bleed

The throat of the Air-intake must be inside the geometry, and the normal shock must be present just downstream of the throat for the starting condition. The contraction ratio is a critical parameter that effectively controls the starting behavior of an intake [31]. A low contraction ratio increases the chance of starting as the flight Mach number increases. However, this leads to a large area of the throat that may degrade the intake performance. Due to the flow spillage during the unstart condition, the spillage drag of the intake increases. The main reason for the intake unstart for the Hypersonic inlet is the boundary layer separation caused by the shock wave-boundary layer interaction at the intake entrance. Bleed has the capability to solve both the supersonic and the hypersonic unstart problem, and the bleed increases the actual flow area at the throat, which improves the amount of the intake mass flow.

5.2 Simulation for different free-stream Mach number

The starting and unstarting criteria of intake affect the overall engine performance, so analysing the intake with different free-stream Mach numbers is necessary. To determine the start and unstart conditions of the inlet, simulations were run for the supersonic and the hypersonic Mach numbers. The converged solution, corresponds to a higher Mach number, is fed as the initial flow field values. The simulation is carried out with free-stream Mach values of 2.5, 3, 3.2, 3.3, 3.5, 4, 5, 6, 7, and 8.

Figures 8 and 9 depict the variation in TPR for various free-stream Mach numbers without bleed. The total pressure ratio (TPR) is the ratio of the total pressure at the outlet to the total pressure at the inlet. For the optimal inlet performance, TPR and MCR both should be at their optimal condition. Here we can see that the TPR (fig 9) drops from the Mach number 2.5 to 3 and then reaches its peak near the Mach number range of 3.2-3.3. But if we look at Figure 8, we can see that MCR rises from 2.5 to 3, and then a sharp rise in MCR occurs near Mach 3-3.3. So, a critical point occurs near Mach 3-3.3. After this limit, TPR starts falling down and MCR further shoots up, and the pattern formed by them is quite smooth without much change in trend. So, understanding the flow physics at this particular Mach number is quite significant. Therefore, for this research we have undertaken a detailed effect of Bleeding only after Mach 3. The graph shows a sudden drop in TPR around Mach 3 due to shockwave boundary layer interaction, indicating the inlet unstart.

5.3 flow physics and effect of bleed size on TPR and MCR

Development of critical intake conditions near the mach 3 has been captured by analyzing multiple cases at Mach 3, 3.2, 3.3, and 3.5. The Mach number distribution for the free-stream Mach number 3.2 is shown in Figure 10. The figure also includes an enlarged view. The interaction of Mach 3.3 with the first ramp produces a weak oblique shock that is not attached to the cowl and originates above it, indicating spillage. The reflected shock interacts with the lower ramp once more, causing the boundary layer to interact with the shockwave, resulting in poor performance. In addition, the cowl shock is repeatedly reflected by both the ramp and the top wall, resulting in a train of oblique shocks inside the intake that compresses the flow even more.

![Figure 10. Mach number contour for M=3.3](image)
Separation bubbles exist after the bleeding in all cases. The 3 mm and 7.1 mm bleed sections exhibit the same trend until Mach 6. At around Mach 8, the TPR of bleed size 7.1mm almost coincides with TPR without bleed, and the bleed size of 3mm coincides with the bleed size of 8.6mm. The 8.6mm bleed indicates that it provides the best TPR while operating between Mach 4 and 6; it provides almost 20% increase in TPR over no-bleed configurations but gradually decreases for Mach 8, where the increase in TPR over no bleed is around 15%. 3mm bleed consistently shows a linear pattern throughout the Mach number range of 4 to 8. The bleed has a consistently higher TPR of around 15%.

As per Figure 17, the TPR for almost all bleed geometry is almost the same at or near Mach 3-4, but their effect becomes prominent as we increase the operating Mach number. TPR for 1.6mm bleed is almost the same as for no bleed configuration. TPR keeps on increasing with increased bleed size. At Mach 6, TPR for 8.6mm bleed is the highest. The shockwave is also modified near the bleed, as shown in Figure 13-16.

The impact of the MCR is almost similar to that of TPR. However, the overall mass flow rate of the air-intake system is a significant factor, and it has been provided in Tables 2-5.
Table 2 and 5 below shows the percentage drop in the mass flow rate when different bleed geometries are implemented in the air-intake model.

Table 2. Percentage drop in mass flow rate in the air-intake at bleed size= 1.6175mm

<table>
<thead>
<tr>
<th>Sl. No.</th>
<th>Mach no.</th>
<th>% drop in MFR (appx)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3</td>
<td>1.53955</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>1.314169</td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>0.6138</td>
</tr>
<tr>
<td>4</td>
<td>8</td>
<td>0.77629</td>
</tr>
</tbody>
</table>

Table 3. Percentage drop in mass flow rate in the air-intake at bleed size= 3 mm

<table>
<thead>
<tr>
<th>Sl. No.</th>
<th>Mach no.</th>
<th>% loss in MFR (appx)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3</td>
<td>0.175241</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>2.936982</td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>2.018282</td>
</tr>
<tr>
<td>4</td>
<td>8</td>
<td>0.057107</td>
</tr>
</tbody>
</table>

Table 4. Percentage drop in mass flow rate in the air-intake at bleed size= 7.131mm

<table>
<thead>
<tr>
<th>Sl. No.</th>
<th>Mach no.</th>
<th>% drop in MFR (appx)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3</td>
<td>3.352975</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>3.893315</td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>1.597731</td>
</tr>
<tr>
<td>4</td>
<td>8</td>
<td>0</td>
</tr>
</tbody>
</table>

Table 5. Percentage drop in mass flow rate in the air-intake at bleed size= 8.6 mm

<table>
<thead>
<tr>
<th>Sl. No.</th>
<th>Mach no.</th>
<th>% drop in MFR (appx)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3</td>
<td>8.507176</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>3.893315</td>
</tr>
<tr>
<td>3</td>
<td>6</td>
<td>9.310046</td>
</tr>
<tr>
<td>4</td>
<td>8</td>
<td>5.0122798</td>
</tr>
</tbody>
</table>

As per the table, if we compare the percentage drop in MFR for the bleed size of 3mm and 8.6mm, we can see the loss is significant in the latter. As we know, the thrust generated by the engine is a function of MFR, so there will be a huge thrust loss in the 8.6mm bleed compared to the 3mm bleed, making it unfit for implementation. On the contrary, the TPR of 3mm and 8.6mm are in close agreement.

6. CONCLUSION

Air intake is an essential part of a high-speed engine, and its design impacts the engine's overall performance. Computational studies of the 2D supersonic and hypersonic air intake with and without bleed are presented in this paper. In-house algorithm and code are fed on commercial CFD software, coupled with density-based RANS equations, and further solved with a k-ω/SST turbulence model. The computed wall pressure distribution and intake performance parameters (TPR and MCR) reasonably match the experimental results. The steady-state simulations were performed for various free-stream Mach numbers ranging from 2.5 to 8. The sudden drop in the mass capture and total pressure ratio determines the free-stream Mach number at which intake unstarting occurs. The bleed geometry in this paper has been shown to delay the flow separation that reduces the contraction ratio, hence, the effective throat area. It also increases engine performance through increased TPR. Sincethe bleed geometry also reduces the mass flow rate, in the current paper, we have also tried to maintain the trade-off between the MFR and TPR. The air intake has been studied with four different bleed sections ranging in size from 1.6mm to 8.6mm. Critical Mach number where the abrupt transition in the value of TPR and MCR occurs is obtained in the range of 3-3.3.

Further simulations with bleed geometry were run from Mach numbers 3 to 8. Low Mach number like 3-3.3 hardly shows any improvement with the bleed, but their effect becomes more dominant when we cross Mach number 4. Among different bleed geometries, the 3mm bleed diameter is the most consistent among its peers as its performance remains almost constant from Mach 4 to 8. In Tables 2-5, we can see that as the size of the bleed increases, the drop in MFR also increases. 8.6mm bleed geometry has undoubtedly increased the TPR over the non-bleed geometry, but it’s only for a small range of Mach 4-6 at the cost of thrust loss. At Mach 8, the TPR improvement achieved with 3mm and 8.6mm bleed is almost the same. Taking into account the loss in the mass flow rate due to the 8.6mm bleed, we recommend that a 3mm bleed is an ideal trade-off while operating at different Mach numbers of 4-8.

The unsteady analysis will provide the amplitude and frequency of the buzz phenomena for the same set of bleed geometries. It will be the scope of further research.

ACKNOWLEDGMENT

The authors would like to thank the Amity Institute of Aerospace Engineering faculty and staff members, who provided valuable feedback during this research.

REFERENCES


[20] Kumar, G. and De, A.: Modes of unsteadiness in shock wave and separation region interaction in hypersonic flow over a double wedge geometry, Physics of Fluids, Vol. 33, Article No. 076107 2021


NOMENCLATURE

2D RANS Two-Dimensional Reynolds-averaged Navier–Stokes

k – ω shear stress transport model

k - ωSST Turbulent kinetic energy

ω Specific dissipation rate
$\rho$ Density

$x_i, x_j$ Spatial coordinates

$\alpha$ the coefficient that depends on the specific implementation of the turbulence model

$u_i, u_j$ Velocity tensor

$G_k$ It represents the generation of turbulent kinetic energy due to mean velocity gradients

$\varepsilon$ Turbulence dissipation rate

$\mu$ Dynamic viscosity

$\mu_t$ Turbulent viscosity

$Pr_k$ Turbulent Prandtl number for turbulent kinetic energy

$Pr_{\omega}$ Turbulent Prandtl number for the specific dissipation rate

$Pr_\gamma$ Turbulent Prandtl number for thermal conductivity

$\beta^*$ A constant coefficient

$\tau_{ij}$ Stress tensor

$\gamma$ Thermal conductivity of the fluid

$\gamma_t$ Turbulent thermal conductivity

$T$ Temperature

$T_{\text{ref}}$ Reference temperature specified (273K)

$\mu_{\text{eff}}$ Dynamic viscosity of the fluid at $T_{\text{ref}}$

$S$ Sutherland constant

$Q$ Heat source or sink terms

$E$ Total Energy per unit mass

$c_v$ Turbulent viscosity constant

PDE Partial Differential equation

$P$ Pressure

$P_{\text{inf}}$ Free stream pressure

$\text{TPR}$ Total pressure ratio

$\text{MCR}$ Mass capture ratio

$\text{MFR}$ Mass flow rate

$\text{CFD}$ Computational Fluid dynamics

$\text{M/M}_\infty$ Mach number/Free stream Mach number

$L$ Length of the upstream symmetry section

$x/h$ The ratio of wetted length and the intake throat height

**ПАСИВНА МОДИФИКАЦИЈА ПРОТОКА ПРЕКО СУПЕРСОНИЧНОГ И ХИПЕРСОНИЧНОГ СИСТЕМА ЗА УСИСАВАЊЕ ВАЗДУХА ПОМОЋУ ОДВОДА**

Ц. Синя, С. Синг, О. Пракаш, Д. Панчал

Усисник ваздуха треба да ради у пројектованим условима да би се постигло висок укупни опоравак притиска и оптималан однос масе. Актуелно исуча–живљење се фокусира на нумеричку симулацију надзвучног и хиперсоничног улаза ваздуха у његових почетних и непокренутим карактеристикама. 2D RANS једначина за суперсонични и хиперсонични унос је решена коришћењем $k$–$\omega$SST модела турбулентности. Ин-хаус код и алгоритам заснован на RANS једначини су такође валидирани у одговарајућем поступку и коришћени за накнадне симулације. Нагли пад у односу масе указује на непо–кренуто стање уноса. Присуство одводног дела има похвалан ефекат на параметар перформанси усисника ваздуха. Уочен је мехур одвајања на улазу у усис током ванпројектованих услова, што је резултирало губицима перформансе. Коришћена су четири различита подела одељака за испуштање у распону величине од 1,6 мм до 8,6 мм, а симулације са блидом су вођени за различите Махове бројеве у распону од 3 до 8. Оптимална величина испуштања од 3 мм се показала привидно ефикасном у модификацији укупног опоравка притиска унутар оптимални масен проток у широком опсегу Махових бројева.