Aerothermodynamics of Hypersonic Shock Wave Boundary Layer Interactions

N. R. Deepak, S. L. Gai and A. J. Neely

School of Engineering & IT
High Speed Flow Group
UNSW, Australian Defence Force Academy
Canberra 2600, Australia

Abstract

Hypersonic laminar flow past a 24° compression corner has been numerically investigated using computational fluid dynamics. The flow conditions were: specific total enthalpy of $h_0 \approx 19\text{MJ/kg}$, $R_e \approx 3.1 \times 10^5 \text{1/m}$ and a Mach number of $M_{\infty} \approx 7.5$. A grid independence study has been carried out to estimate the sensitivity of heat flux and pressure in the strong interaction regions of the flow. The CFD investigation of perfect gas has also been extended to study the influence of real-gas effects such as thermal and chemical nonequilibrium using Park’s two-temperature model with finite-rate chemistry. The CFD results are compared with the available experimental data.

Introduction

Hypersonic flow past a compression corner is a typical problem of shock wave/boundary layer interaction (SWBLI). The importance of SWBLI on aerothermodynamic characteristics has led to its extensive investigation due to its relevance in the design of hypersonic space vehicles. While the compression corner problem appears to be a simplified geometric configuration, it is analogous to a number of generic regions of separated flow on a space vehicle. In high enthalpy hypersonic flow, the total energy content is sufficiently high enough to cause changes in the internal distribution of energy levels within the gas. This leads to molecular excitation that eventually induces chemical reactions via dissociation and ionisation. This is the thermo-chemical, non-equilibrium behaviour of the flow. Understanding SWBLI under such a scenario is therefore necessary for the reliable design of space vehicles.

The compression waves thus generated coalesce to form a reattachment shock. Downstream of reattachment the shear-layer grows and the flow becomes parallel to the surface. The separated shock may interact with the recompression shock forming a shock/shock interaction with the emergence of a slip line.

Geometric configuration & Flow conditions

A sharp leading edge flat-plate of length ($L_e = 85 \text{mm}$) followed by a compression corner with a ramp angle ($\theta_e$) of 24° is considered here under laminar flow high enthalpy conditions. The down stream ramp has a length of 95 mm. The flow is assumed to be two-dimensional. Experimental investigation [7] for this comparison was initially carried out at the T3 shock-tunnel facility with a specific total enthalpy of 19 MJ/kg and with a unit Reynolds number of $3.10 \times 10^7 \text{ 1/m}$ and $M_{\infty} \approx 7.5$. Air was used as the test gas. The CFD calculations have therefore been performed at these flow conditions. Table 1 summarises the freestream conditions used for this investigation.

### Table 1: Freestream conditions (Condition B [7]).

<table>
<thead>
<tr>
<th>(U_{\infty})</th>
<th>(T_{\infty})</th>
<th>(p_{\infty})</th>
<th>(\rho_{\infty})</th>
<th>(T_e/T_{\infty})</th>
<th>(\gamma)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5470</td>
<td>1160</td>
<td>0.99</td>
<td>2.60 \times 10^5</td>
<td>0.035</td>
<td>1.45</td>
</tr>
</tbody>
</table>

The test gas composition evaluated [7] for condition B through non-equilibrium analysis is given in Table 2.

### Table 2: Freestream species concentration

<table>
<thead>
<tr>
<th>Species</th>
<th>Mass fraction</th>
</tr>
</thead>
<tbody>
<tr>
<td>$N_2$</td>
<td>0.750</td>
</tr>
<tr>
<td>$O_2$</td>
<td>0.06</td>
</tr>
<tr>
<td>$N$</td>
<td>0.001</td>
</tr>
<tr>
<td>$O$</td>
<td>0.159</td>
</tr>
<tr>
<td>$N/O$</td>
<td>0.03</td>
</tr>
</tbody>
</table>

Computational Modelling

The two-dimensional flow field of interest was modelled using a multi-block Navier-Stokes solver, Eilmer-3 [6]. Eilmer-3 is a time-dependent, compressible, viscous, three-dimensional solver capable of solving laminar, turbulent and chemically reactive flows. Eilmer-3 operates on finite-volume approach, where the discretization occurs at the center of each control volume. A modified van Albada limiter and MUSCL (Monotone Upstream-centered Schemes for Conservation Laws) reconstruction scheme [1] are used to obtain second-order spatial accuracy. To calculate the mass, momentum and energy flux across the cells, the advection upwind splitting method combining difference and vector splitting (AUSMDV) [11] method is used. AUSMDV has robust shock-capturing capabilities, where the discontinuities are captured with high resolution and is particularly suitable for solving finite-rate chemical reactions. The carbuncle phenomena typically encountered during multi-dimensional shock capturing is treated here with shock-fix technique [11]. Eilmer has capability to solve the multi-block grids utilising multiple CPU through Message Passing

Figure 1: Flow schematic over a 2D compression corner

The laminar flow over a compression corner as shown in Figure 1 is characterised by boundary layer over the flat plate with a leading edge shock. With the presence of the corner, a shock wave forms and interacts with the boundary layer. If the corner angle ($\theta_e$) is large enough, then the boundary layer separates forming a separation bubble with recirculating flow. A separation shock in turn forms at the location where the boundary layer separates. Depending upon the Mach number, the leading edge shock may interact with the separation or reattachment shock. Downstream from the corner, the separated shear-layer reattaches to the surface. When this occurs, the thickness of the shear-layer is reduced and the flow turns towards the surface forming a neck, which is also shown in Figure 1.
Interface (MPI) library. Therefore, the CFD calculations were accordingly performed on multiple CPUs.

The grid independence study was performed using the assumption of perfect gas [3, 4, 10]. The grid independence study using the perfect gas assumption is justified because with the real gas effects, the only difference will be that it takes longer time for grid convergence. This is because with the real gas, the gas is treated as a mixture of thermally perfect gases. Air was used as the test gas with single species assumption and the thermodynamic behaviour was modelled as colorically perfect with a constant ratio of specific heats. In this case, the transport properties were evaluated using the Sutherland formulation. Real gas effects such as chemical and thermal nonequilibrium phenomena in the flow field were additionally investigated using the Park's [9] kinetic mechanism. The finite-rate chemistry for air with five species was computed for the dissociation and recombination reactions, ignoring the effects of ionisation. The thermal nonequilibrium effects were computed using the two-temperature model of Park [9]. Details of finite rate reactions are given in Table 3 and reaction rates were calculated from the Arrhenius coefficients given in Park [9].

Table 3: Finite-rate chemical reactions for high temperature air

<table>
<thead>
<tr>
<th>Reaction</th>
</tr>
</thead>
<tbody>
<tr>
<td>$N_2 + M \rightarrow N + N + M$</td>
</tr>
<tr>
<td>$O_2 + M \rightarrow O + O + M$</td>
</tr>
<tr>
<td>$NO + M \rightarrow N + O + M$</td>
</tr>
<tr>
<td>$NO + O \rightarrow O_2 + N$</td>
</tr>
<tr>
<td>$N_2 + O \rightarrow NO + N$</td>
</tr>
</tbody>
</table>

Figure 2: Stanton number distribution for various grids

Table 3: Finite-rate chemical reactions for high temperature air

For the real-gas computations, the gas is treated as a mixture of thermally perfect gases, where $C_p$ and $C_v$ are functions of temperature. The energy exchange between vibrational and translational modes of the molecules were evaluated using the Landau-Teller equation and vibrational relaxation time was then estimated using the Millikan and White empirical correlation [9]. The vibrational energy was finally calculated using the harmonic oscillator model in a Boltzmann distribution given at the vibrational temperature ($T_v$). Transport properties of the species mixture were then calculated using the method adopted by NASA's CEA program [5] that utilises interaction potential between each of the species for estimation. The finite-rate chemical reactions were modelled as a set of ordinary differential equations using a alpha-QSS method [6]. Diffusion velocities were modelled using a modified a Ficks Law [6].

A CGNS multi-block structured grid was developed for this problem using the commercial package, ICEM-CFD. The upstream flat-plate and the wedge surfaces are modelled as cold wall with a fixed wall temperature of $T_w=300K$. The inlet boundaries are specified with uniform supersonic velocities and the supersonic outlet is specified at the trailing edge of the model. Initialisation of the CFD simulation was performed with zero velocity and a pressure of 50 Pa, similar to the experimental conditions at $t=0s$. The flow was simulated in a time-accurate manner, until the flow variables converged to steady conditions. The grid independence study is carried out for this configuration and follows the approach adapted by Deepak et al. [2] for high enthalpy separated flows. Non-dimensional heat flux (Stanton number) variation is known to provide a sensitive test [2] for CFD solution and hence is used here as a criteria along with surface pressure and skin-friction. However, only Stanton number sensitivity is presented in Figure 2. Here, the wetted surface ($s$), is normalised with the upstream flat-plate length ($L_p$), such that $s/L_p = 1$ represents the corner location. The Stanton number number was evaluated as

$$S_l = q_{hu}/[\rho_e u_e (h_l - h_u)],$$

where, $q_{hu}$ is the surface heat flux and $h_l$ is the recovery enthalpy evaluated as $h_l = h_e + \sqrt{P_r} \rho_e u_e^2 / 2$. The subscript $e$ refers to the condition that exists at the edge of the boundary layer.

As can be noted from Figure 2, in the region $0 \leq s/L_p \leq 0.625$ not much variation in heat flux is noted. However, as the heat flux reduces due to separation $(0.625 \leq s/L_p \leq 0.75)$, changes are evident. Grid-1 severely under estimates the separation location. With further grid refinement, separation size appears to increase. Only after Grid-5, no change in the value of the Stanton number at separation is seen. Immediately after separation, within the recirculation region, not much differences between various grids can be noticed, except Grid-1. This suggests that, although heat flux may be insensitive with sufficiently refined grids in the separated region, adequate grid resolution is still required to predict proper scale of separation and its location. Downstream of the reattachment ($s/L_p \approx 1.2$), significant variations can be noted in the close vicinity of peak heat flux location. While, Grid-1 can be seen to over estimate the heat flux until it reaches a peak value, further downstream it under estimates in comparison to other grids. With more refinement, a gradual reduction in heat flux occurs as well as the location of the peak on the wedge surface. This indicates that both reattachment and peak heat flux location are highly grid dependent.

Beyond $s/L_p \approx 1.75$ in Figure 2, the region where heat flux gradually reduces, not much grid sensitivity is seen similar to what is seen upstream of separation. Nevertheless, only beyond Grid-5, heat flux does not vary with refinement in regions of separation/reattachment and up to the trailing edge. Hence, all other computations were carried out using Grid-5. Grid-5 comprises of 570 nodes in $x$ and 102 nodes in $y$ directions respectively, contributing to 58140 nodes in total. The first node from the wall surfaces ($\Delta y$) was at a distance of $20 \times 10^{-6} \mu m$ (20 $\mu m$). It has been shown by Deepak et al. [2] that $\Delta y$ of this order is required for proper heat flux estimation in high enthalpy separated flows. Since these numerical experiments were time-accurate, it was found that 280 $\mu s$ was required for the flow quantities to become completely established. For a perfect gas analysis, this was $\sim 130$ hours of computational time on 8 core Xeon CPUs. Figure 3 shows schematically the grid distribution for Grid-2.

### Results & Discussions

In Figure 4, the Stanton number ($S_l$) distribution obtained through CFD is compared with the experiments as well as the CFD data of Olejniczak and Candler [8]. Olejniczak and Candler [8] carried out two-dimensional CFD simulations using Park’s [9] two-temperature model including chemistry for the
same geometric configuration and flow condition as presently
investigated. The data of Olejniczak and Candler [8] taken here
for comparison was obtained on their finest grid comprised of
(1024 × 1024). With respect to the present simulation results,
we present two sets of CFD data, one under the assumption
of perfect gas (PG) and the other assuming thermo-chemical non-
equilibrium real-gas (RG). The reason for comparing the experi-
mental data with CFD simulations assuming both perfect gas
and real gas was because Mallinson et al. [7] found that for
the experimental data considered here, the real gas effects were
negligible.

As can be see from Figure 4 the distribution of heat-flux over
the initial length of the flat-plate is a result of attached bound-
ary layer growth and the leading edge shock. In this region, not
much difference can be seen between the present CFD data, al-
though it slightly over predicts the experimental data. However,
Olejniczak and Candler [8] results closely match the experimen-
tal values. After $s/L_c \approx 0.625$, where the heat flux reduction is
seen, separation occurs. Here, number of differences can be
seen. The perfect-gas data seem to over predict the separation
size, while the real-gas prediction appears to be closer to the
experimental value. This is also in close agreement with the
Olejniczak and Candler [8] CFD data. Within the recirculation
region upstream of the corner, all the CFD data compare well
with experiments.

Downstream from the corner after the separated shear layer re-
attaches to the plate, some interesting trends are seen. As the neck
region forms (see Figure 1), the thickness of the shear layer is
reduced increasing the heat flux to a peak value. Close to the
vicinity of this peak, reattachment occurs. Due to the reduc-
tion in shear-layer thickness, the temperature gradients become
steep causing high surface heat flux. This is quite clearly seen in
both CFD data and experiments. However, some significant dif-
fences are seen. Firstly, none of the CFD data matches the ex-
perimental peak heat flux location which is at $s/L_c \approx 1.37$. They
all seem to under predict the peak heat flux by a considerable
margin. Nevertheless, the present CFD real-gas result matches
most of the other experimental points over the wedge surface. In
contrast, the data of Olejniczak and Candler [8] severely under
predicts, except in the immediate region past the corner loca-
tion. A possible explanation for this may be that Olejniczak and
Candler [8] found that their computations with finer grid (1024
× 1024) were much lower than the experimental data than the
solution with the grid (512 × 512). They do not offer any ex-
planation for this apparent anomaly. Suffice to say at this stage
that the present solution with Grid-5 and real gas assumption
does predict results closer to the experimental data.

The under prediction of the peak heat flux with respect to ex-
periments suggests that a transition in the separated shear layer
might have occurred or that it is a three-dimensional effect.
Since all the CFD simulations have been performed assuming
laminar flow and two-dimensional conditions, these effects
could not be investigated. Downstream of reattachment, the
shear-layer grows and the flow becomes parallel to the surface
causing a decrease in heat-flux to the surface.

Figure 5 shows the normalised pressure distribution in compari-
son with experiments and also Olejniczak and Candler [8] data. On
the flat plate far upstream of the corner, the CFD results from all
the three sets are in close agreement and indicate a typical distri-
bution of pressure due the boundary layer growth. The real-gas
CFD results cannot be distinguished from the perfect gas data.
It should however be noted that there are no experimental data
to compare in this far upstream region. A pressure increase and
plateau can be seen to occur which is a consequence of adverse
pressure gradient which leads to separation. Based on the lo-
cation of increase in surface pressure and decrease in heat flux
from the real gas model, the separation location may be consid-
ered to occur at $s/L_c \approx 0.75$. This value also seems to match
with Olejniczak and Candler [8] data. The pressure inflection
in perfect gas prediction occurs slightly ahead of that of the
real-gas predictions. A similar trend was also noted with the
reduction in heat flux over this region. This illustrates that the
scale of separated region is over predicted with the perfect-gas
model. The CFD data from Olejniczak and Candler [8] almost
matches the present CFD real-gas data in the separated region.
The two sets of data also compare quite well with experiments.
of peak pressure indicating the shear layer reattachment to the plate matches the peak heat flux location. Beyond this location, the present real-gas computation shows a much reduced pressure in comparison to both perfect-gas and Olejniczak and Candler [8] data, but no where close to the experiments. The reduced pressures in the case of experiments may, as noted earlier, indicate transition or three-dimensional effects. However, three-dimensional effects would seem unlikely because, in the experiments, the model was fitted with side skirts.

Figure 6 shows an infinite fringe flow visualisation obtained from the experiment compared with the CFD real-gas data. The flow features from the fringe diagram have been delineated from the photographic images so that some inaccuracies in the measurements are possible. Here, the Mach contour in grey scale is shown to compare the flow features. As can be seen, the boundary layer thickness predicted over the flat plate region matches the fringe pattern reasonably well. The location of separation and the recirculation region in general are also in close agreement with the visualised image. The other features that are in close agreement are the separation shock, reattachment shock and the shear layer downstream. This indicates that the separation size and the reattachment locations are correctly modelled. In addition, the shock/shock interaction effect from separation and the reattachment shocks is also also correctly modelled and matches the experimental image.

Conclusions

CFD simulations have been carried out for the flow past a two-dimensional compression corner in high enthalpy hypersonic flow. Detailed grid independence study indicates the sensitive areas in the flow field where adequate grid resolution is required. Effects of real-gas phenomena have been considered through modelling thermo-chemical nonequilibrium simulation. The simulated data and experimental comparison appear to show that CFD can reasonably predict the laminar SWBLI in the separated region close to the corner. Nevertheless, some significant differences were noted over the downstream wedge surface, particularly in the location and magnitude of the peak heat flux and pressure and the subsequent distribution of heat flux and pressure. The study has shown that separation, reattachment and recirculation regions are affected by real gas effects.

Acknowledgements

The authors would like to express their sincere thanks to the following people. Dr. Peter Jacobs of the University of Queensland for allowing us to use his Eilmer-3 solver. Daniel Potter of University of Queensland and Dr. Rowan Gollan of NASA Langley Research Center for their support in explaining various intricacies in Eilmer-3 and its usability.

References