DOTTORATO DI RICERCA IN
Ingegneria Industriale e dell’Innovazione

Ciclo XXI

Settore scientifico disciplinare di afferenza: _____Ing–ind/08_______________

TITOLO TESI

Advanced Models for Prediction of High Altitude Aero-Thermal Loads of a Space Re-entry Vehicle

Relatore:
Ch.mo Prof. Aldo Bonfiglioli

Candidato:
Raffaele Votta

Correlatore:
Ch.mo Ing. Antonio Schettino

Coordinatore:
Ch.mo Prof. V. Magi

Triennio 2005-2008
Ad Antonella
## Nomenclature

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
</tr>
</tbody>
</table>

## Subscripts and Superscripts

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
</tr>
</tbody>
</table>

## Chapter 1 Introduction

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
</tr>
</tbody>
</table>

## Chapter 2 Space Re-entry Vehicles

### 2.1 Reusable Launch Vehicle (RLV): Current Status and On-going Activities

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
</tr>
</tbody>
</table>

#### 2.1.1 United States

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
</tr>
</tbody>
</table>

#### 2.1.2 Europe

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
</tr>
</tbody>
</table>

#### 2.1.3 Other Countries

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
</tr>
</tbody>
</table>

#### 2.2 Space Shuttle

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
</tr>
</tbody>
</table>

#### 2.3 ORION Crew Exploration Vehicle (CEV)

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
</tr>
</tbody>
</table>

#### 2.4 Unmanned Space Vehicle (USV)

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
</tr>
</tbody>
</table>

## Chapter 3 Hypersonic Rarefied Flows

### 3.1 Hypersonic Flow Regime

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>35</td>
</tr>
</tbody>
</table>

### 3.2 Rarefied Flow and Navier-Stokes Breakdown

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>39</td>
</tr>
</tbody>
</table>

### 3.3 Basic kinetic Theory and Boltzmann Equation

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>46</td>
</tr>
</tbody>
</table>

#### 3.3.1 The Boltzmann Equation

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>51</td>
</tr>
</tbody>
</table>

#### 3.3.2 The Moment and Conservation Equations

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>57</td>
</tr>
</tbody>
</table>

#### 3.3.3 Accommodation Coefficients

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>60</td>
</tr>
</tbody>
</table>

## Chapter 4 Numerical Solutions

### 4.1 Numerical Solution of Transitional Regime - Direct Simulation Monte Carlo (DSMC)

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>62</td>
</tr>
</tbody>
</table>

### 4.2 Solution of Free Molecular Flow

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>68</td>
</tr>
</tbody>
</table>

### 4.3 Solution of Continuum Regime – Computational Fluid Dynamics (CFD)

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>70</td>
</tr>
</tbody>
</table>

## Chapter 5 Effects of Rarefaction on a Winged Hypersonic Re-entry Vehicle

### 5.1 FTB-X High Altitude Aerodynamic

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>71</td>
</tr>
</tbody>
</table>

#### 5.1.1 Bridging Formulae

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>72</td>
</tr>
</tbody>
</table>

#### 5.1.2 Test Conditions

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>73</td>
</tr>
</tbody>
</table>

#### 5.1.3 DSMC Results for the Whole Vehicle

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>76</td>
</tr>
</tbody>
</table>

#### 5.1.4 Comparison of the Results from DSMC, CFD and Engineering Methods

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>78</td>
</tr>
</tbody>
</table>

#### 5.2 Nose Thermal Analysis

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>81</td>
</tr>
</tbody>
</table>

#### 5.2.1 Test Conditions

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>81</td>
</tr>
</tbody>
</table>

#### 5.2.2 Results

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>82</td>
</tr>
</tbody>
</table>

#### 5.3 Concluding Remarks

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>84</td>
</tr>
</tbody>
</table>

## Chapter 6 Local Effects of Rarefraction in Shock Wave Boundary Layer Interactions (SWBLI)

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>86</td>
</tr>
</tbody>
</table>

### 6.1 Shock Wave Boundary Layer Interaction

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>88</td>
</tr>
</tbody>
</table>

### 6.2 Results

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>92</td>
</tr>
</tbody>
</table>

#### 6.2.1 Hollow Cylinder Flare Test Case

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>92</td>
</tr>
</tbody>
</table>

#### 6.2.2 CIRA Plasma Wind Tunnel Test

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>99</td>
</tr>
</tbody>
</table>

#### 6.3 Concluding Remarks

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>104</td>
</tr>
</tbody>
</table>

## Chapter 7 Orion CEV (Crew Exploration Vehicle): High Altitude Aerothermodynamics

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>106</td>
</tr>
</tbody>
</table>

### 7.1 RTO-RTG043 Working Group

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>106</td>
</tr>
</tbody>
</table>

### 7.2 Test Conditions and Results

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>108</td>
</tr>
</tbody>
</table>

#### 7.2.1 Grid and Molecular Independence

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>109</td>
</tr>
</tbody>
</table>

#### 7.2.2 Slip Flow Boundary Conditions Validation

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>110</td>
</tr>
</tbody>
</table>

#### 7.2.3 Results

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>112</td>
</tr>
</tbody>
</table>

#### 7.3 Concluding Remarks

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>119</td>
</tr>
</tbody>
</table>

## Chapter 8 Conclusions

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>121</td>
</tr>
</tbody>
</table>

## References

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>125</td>
</tr>
</tbody>
</table>
Nomenclature

\( a \) = generic extensive variable
\( a_c \) = accommodation coefficient
\( B \) = magnetic field
\( CD, CL, CS \) = drag, lift and lateral force coefficients
\( CMx, CMy, CMz \) = rolling, pitching and yawing moment coefficients
\( c \) = molecular velocity vector
\( Cf \) = skin friction coefficient
\( Cp \) = pressure coefficient; specific heat at constant pressure
\( Cv \) = specific heat at constant volume
\( D \) = diameter of the nose
\( D_{ij} \) = diffusion tensor
\( e \) = specific energy
\( E \) = aerodynamic efficiency \((E=CL/CD)\), Electric field
\( f \) = normalized distribution function in velocity space
\( F^m \) = Magnetic force
\( F^{(N)} \) = N particle distribution function
\( F^{(R)} \) = reduced distribution function
\( F \) = cumulative distribution function
\( F_N \) = number or real molecules represented by a single DSMC one
\( H \) = total enthalpy
\( h \) = altitude, specific enthalpy
\( HTHL \) = Horizontal Take off and Horizontal Landing
\( K \) = Thermal conductivity
\( Kn \) = Knudsen number
\( L \) = reference length
\( L_x, L_y, L_z \) = dimensions of FTB-X along the rolling (x), pitching (y) and yawing (z) axes
\( M \) = Mach number
\( mcs \) = mean collision separation
\( n \) = number density
\( n_r \) = number flux
N = sampling molecules
P = probability, pressure
PFA = Projected Frontal Area
PND, PNL = parameters defined by Equation 5-3 and Equation 5-4
PPA = Projected Planform Area
q (0) = heat flux at the stagnation point
Q = molecular quantity
r = relative
r = position vector
RLV = Reusable Launch Vehicle
R = gas constant
Re = Reynolds number
s = stream wise wetted length
S = speed ratio
SSTO = Single Stage to Orbit
TPS = Thermal Protection System
St = Stanton number
t = time
T = temperature
u = velocity component in x direction
v = velocity component in y direction
V = velocity
w = velocity component in z direction
WA = Wetted Area
X, Y, Z = dimensions of the computational domain
V = flow velocity, m/s
α = angle of attack
β = angle of side-slip, reciprocal of most probable speed in an equilibrium gas
δ = boundary layer thickness, mean molecular spacing
ε = fraction of specular reflection
ε₀ = dielectric constant of vacuum
γ = specific heat ratio
λ = free molecule path
\(\lambda_e\) = electrical conductivity tensor
\(\psi\) = single particle distribution function in phase space, inverse power law
\(\rho\) = density
\(\sigma_T\) = collision cross section
\(\tau\) = viscous stress tensor
\(\nu\) = kinematic viscosity, mean collision frequency
\(\omega\) = molecular weight
\(\Omega\) = solid angle, production of specie
* = post collision values

**Subscripts and Superscripts**

c = charge
fm = free molecule
i = inviscid, incident, specie
n = component normal to solid wall,
m = moved
r = reflected
reat = reattachment
s = slip
sep = separation
vibr = vibrational
w = wall
\(\infty\) = free stream
0 = stagnation
2 = downstream a normal shock wave
– = normalized
Chapter 1  Introduction

The main purpose of the present Ph. D. thesis is the study of the capabilities in prediction of aero-thermal loads acting on a space re-entry vehicle at high altitude and improves understanding and modelling of relevant physics in rarefied flows.

The design of a space re-entry vehicle is one of the most challenging problems in aerospace engineering, since the high energy occurring during the re-entry trajectory causes large thermal and mechanical stresses.

The next figure shows the re-entry trajectory of Space Shuttle and the most challenging technological problems to be solved during the descent phase: A) transonic stability; B) maximum hinge moment; C) lateral controllability; D) black out and maximum heating; E) air rarefaction. The present work relies on the problem “E” showed in Figure 1-1

Figure 1-1 Design trajectory for the Re-entry of Space Shuttle Orbiter.

In the frame of the Italian National Space Program (PRORA), the Italian Aerospace Research Centre (CIRA) is carrying on, in collaboration with Alcatel Alenia Space Italia, the design of an experimental Unmanned Space Vehicle (USV) to be used as a test bed for the re-entry technologies. USV program, following a technical and programmatic revision in 2004, has been addressed to the development of a second type of flying test bed (FTB-X) devoted to in-flight testing and validation of system design and technology issues related to atmospheric re-entry and hypersonic flight regime.
One of the most challenging requirements of the FTB-X re-entry is a high aerodynamic efficiency, necessary for realizing a trajectory significantly longer than the one of Space Shuttle that performs a ballistic re-entry at high altitude. A long re-entry trajectory, in fact, will lead to a big down-range and will allow performing several manoeuvres.

To this aim the study and good prediction of the high altitude aerothermodynamics the evaluation of effects of rarefaction of gas in prediction of the main aero-thermal loads of a Space re-entry vehicle (in particular of the CIRA FTB-X) is presented and analyzed.

The present Ph. D. research activity is based on the technological program (CLAE=Configuration and Local Aerothermodynamics-reference 1) being developed by CIRA to study the main problems occurring during FTB-X re-entry, since one of these investigations relies on the analysis of rarefaction effects at high altitude.

The study of such flow regime presents theoretical and numerical difficulties.

It is well known that the Navier-Stokes equations fail in rarefied regimes and a molecular approach such as the Direct Simulation Monte Carlo method (DSMC) is necessary. Moreover, slip flow boundary conditions have been implemented in CIRA CFD code H3NS in order to extend the validity of a continuum method in the transition flow regime and bridging-formula models for the high altitude aerodynamic of a winged bodies, have been developed.

The focus of this investigation is to provide information on the most important problems to be solved in design of a space re-entry vehicle: Aerodynamic efficiency, blunt body heating and efficiency of the control surfaces.

These problems will be analyzed in low density conditions. It is worth to underline that such kind of studies are currently under investigation by the hypersonic community, since the old space re-entry vehicle pass through the zone E of Figure 1-1 rapidly performing a ballistic re-entry (with an high angle of attack) differently from FTB-X.

The improvement of the present research activity is the understanding of challenging problem in hypersonic in conditions (i.e. low density) not yet in depth analyzed.

The present analysis has been conducted on the CIRA Unmanned Space Vehicle (FTB-X), the NASA Crew Exploration Vehicle and on an axisymmetric Hollow Cylinder Flare in wind tunnel conditions.
Chapter 2  Space Re-entry Vehicles

2.1. Reusable Launch Vehicle (RLV): Current Status and On-going Activities

The access to space is one of the most challenging problems in engineering. Its utilization is important for different reasons; first of all from an economical point of view due to the importance of telecommunications and navigation satellite systems. The use of space satellites is also essential for some scientific investigations, for example for earth environmental studies and geological researches. Other sciences are strictly connected to the use of space by means of space platforms (International Space Station, Space Lab., etc.) for microgravity experimental activities. The access of space is also crucial for political and military reasons.

A Space mission can be divided in three segments: 1-Launch Segment; 2-Space Segment; 3- Ground Segment. The present paragraph relies on the next generation of future launch vehicles, related to the first segment of a space mission.

The costs of access to Space are very high; to take 1 kg of payload to the International Space Station about 40 K$ are needed. The cost reduction is one of the key points in design of the next generation of future launch vehicle.

Nowadays, the cheapest conventional non-reusable launchers are the U.S. Delta, the Russian Proton. The European Ariane and Vega are more expensive as, of course, the use of the manned Spacecraft such as the Space Shuttle.

![Conventional non-reusable launchers.](image)
To reduce the costs, some space agencies and industries are planning the development of a next generation of launchers completely or partially reusable, called Reusable Launch Vehicles (RLV).

**2.1.1. United States**

United States of America was the first country that developed a partially reusable launch vehicle (first generation of Reusable Launch Vehicle); with the NASA Space Shuttle since 1981. However, this spacecraft is more expensive to be used than the conventional launchers showed in the Figure 2-1 because is a manned vehicle and because it needs complex operations to prepare the spacecraft to launch.

The main objective of the RLV research activity is to design an aerospace plane able to perform an horizontal take off and horizontal landing (HTHL) with a single stage (SSTO). These characteristics should allow a drastic reduction of costs and time between two consecutive launches (note that the time needed to prepare the Space Shuttle for a flight is about three months).

The conventional re-entry vehicle are characterized by a blunt body need to dissipate the high energies occurring during the re-entry; this feature reduces the manoeuvrability of the vehicle during the descent phase. In fact the Space Shuttle orbiter can be manoeuvred only the last part of the re-entry trajectory, while at high altitude it flies with a constant angle of attack of about 40 degrees.

The use of the sharp leading edges on a winged vehicle could allow to perform a manoeuvred re-entry since the beginning of the descent phase, in order to increase the down range and allow the vehicle to land in more sites in the world, differently from all the current space re-entry vehicles.

The Space vehicles SSTO-HTHL will constitute the so called third generation of RLV. This generation needs an air-breathing propulsion system to perform the horizontal take off, that is an engine that allows the vehicle to reach super-hypersonic velocities by means of air/H\(_2\) combustion. These engines are still under development, so a second generation of RLV is being developed and they will be constitute by a Two Stage to Orbit and a Vertical Take off and Horizontal Landing (TSTO-VTHL).

Key technologies for design of the second and third generation of RLV are being developed by NASA with experimental vehicles, such as X-34 and X-43. X-34 is a winged hypersonic vehicle able to reach M=8 at the altitude of 90 km, its configuration
is an upgrade of Space Shuttle; this program has been cancelled for economical reasons, but the scientific databases are a reference for all the future projects. X-43 first flight has been performed the 27th of March 2004. X-43 is an hypersonic vehicle released from a B-52 airplane; it is equipped with a conventional solid rocket Pegasus that allows to reach to the target speed and altitude (i.e. M=7 and h=30 km); at this velocity, for the first time in aerospace history, a scramjet rocket has been ignited in flight conditions for 10 seconds, followed by a 10 minute glide and intentional crash into the ocean.

![Figure 2-2 X-34 and X-43.](image)

The space capsule eclipsed for decades by the more complex and costly shuttle now appears likely to emerge as successor of Space Shuttle in the middle term. Realization of such a change is demonstrated by China’s recent manned spaceflight successes and NASA announced vision of a new space craft for human space exploration the ORION Crew Exploration Vehicle (CEV). China conducted a second successful manned spaceflight in October 2005 with the Shenzhou 6, a configuration that is an adaption of Russian veteran Soyuz design. On September 19, 2005, NASA announced the findings of the Exploration Systems Architecture Study that recommended the use of an Apollo-like capsule for the CEV design. The CEV is much larger than Apollo, almost twice the mass of the Apollo Command Module along with a much larger volume, and like Apollo, would be attached to a service module for life support and propulsion. Mission applications of the CEV include a low Earth orbit (LEO) version with a crew of six to the International Space Station, a lunar version that would carry a crew of four and a Mars version that would carry a crew of six.
Figure 2-3 Manned Spacecrafts.

Definitely, the next generations of future launch vehicle can be divided in two categories: Reusable Launch Vehicle (second e third generation) and the revisiting of the “classic” re-entry capsule. United States has planned the following programs in the ISTP (Integrated Space Transportation Plan):

- Space Shuttle Safety Upgrades.
- Space Launch Initiative (ORION-CEV capsule will substitute the Space Shuttle).
- Third generation of RLV (derived from X-34, X-38 and X-43 concepts).

2.1.2. Europe

In Europe, European Space Agency (ESA) is funding the FLPP program (Future Launchers Preparatory Program) with the goal to select concepts for the future European TSTO launcher. Moreover, ESA is carrying on the EXPERT project, aimed (as NASA with X vehicles) to the development of re-entry capsule equipped with scientific payloads to gather flight data on particular aerothermodynamic phenomena, and.

EXPERT (European Experimental Re-entry Testbed) aim is to design and instrument a generic configuration for in flight measurement of critical aerothermodynamic phenomena using state-of-the art instrumentation. The Russian Volna launcher will be used for in-flight testing.

From experience with ambitious experimental vehicles around the world, such as NASA’s series of X-vehicles, there is general agreement within Europe’s space community that a step-by-step flight programme is the best approach. It limits the risks,
allows stepped costs and ensures that progressively more sophisticated developments benefit from the results of relatively low cost missions, so FLPP program has been launched by ESA with different flight demonstrator. To this aim the development of an Intermediate eXperimental Vehicle (IXV) dedicated to validating re-entry technologies. Studies for this are already underway and the IXV is scheduled to make its first orbiting flight, launched by Vega in 2010 and It will be released from an altitude of 120 km.

Moreover, ESA is studying the third generation of RLV in the so called LAPCAT (Long-Term Advanced Propulsion Concepts and Technologies) program, whose main purpose is the preliminary design of an hypersonic vehicle equipped with air breathing propulsion. Next Figure 2-5 shows the concepts being analyzed.
2.1.3. Other Countries

Japan Aerospace Exploration Agency has already launched different experimental vehicle:

- **OREX (Orbital Re-entry EXperiment)**
  A re-entry capsule launched in the 1994.

- **HYFLEX (Hypersonic FLight EXperiment)**
  An Hypersonic vehicle launched in the 1995 to perform investigations on the hypersonic flow regime.

- **ALFLEX (Automatic Landing FLight EXperiment)**
  A flight test campaign conducted in the 1996 to perform experiment on guide and navigation systems and automatic landing.

![OREX HYFLEX ALFLEX](image)

*Figure 2-6 Japanese Spacecrafts.*

China was the third nation in world to perform a manned mission with Shenzhou spacecraft launched on a Long March 2F rocket booster. As already told before, the re-entry has been performed with a capsule vehicle.

In the frame of the Italian national space program, the Italian Aerospace Research Centre (CIRA) is carrying on, the design of an experimental Unmanned Space Vehicle (USV) to be used as a test bed for the re-entry technologies. The present Ph. D. activity mainly relies on USV program, so a more exhaustive description will be treated in the next paragraph 2.4.

Moreover, the present research activity deals with Space Shuttle and CEV vehicles, so a brief in-depth description will be presented in the next paragraphs 2.2 and 2.3.
2.2. **Space Shuttle**

NASA’s Space Shuttle is the spacecraft currently used by the United States government for its human spaceflight missions. At launch, it consists of a rust-colored external tank (ET), two white, slender Solid Rocket Boosters (SRBs), and the orbiter, a winged spaceplane which is the space shuttle in the narrow sense. The main features are showed in Figure 2-7.

The orbiter carries astronauts and payload such as satellites or space station parts into low earth orbit, into the Earth's upper atmosphere or thermosphere. Usually, five to seven crew members ride in the orbiter. The payload capacity is 22,700 kg. When the orbiter's mission is complete it fires its Orbital Maneuvering System (OMS) thrusters to drop out of orbit and re-enters the lower atmosphere. During the descent and landing, the shuttle orbiter acts as a glider, and makes a completely unpowered ("dead stick") landing.

![Figure 2-7 Space Shuttle main features](image)

The Shuttle is the first orbital spacecraft designed for partial reusability. It carries payloads to low Earth orbit, provides crew rotation for the International Space Station (ISS), and performs servicing missions. The orbiter can also recover satellites and other payloads from orbit and return them to Earth, but this capacity has not been used often.
Six air-worthy Shuttles have been built; the first orbiter, Enterprise, was not built for space flight, and was used only for testing purposes. Five space-worthy orbiters were built: Columbia, Challenger, Discovery, Atlantis, and Endeavour. Challenger disintegrated 73 seconds after launch in 1986, and Endeavour was built as a replacement. Columbia broke apart during re-entry in 2003.

The orbiter resembles an aircraft with double-delta wings, swept 81° at the inner leading edge, and 45° at the outer leading edge. Its vertical stabilizer's leading edge is swept back at a 50° angle. The four elevons, mounted at the trailing edge of the wings, and the rudder/speed brake, attached at the trailing edge of the stabilizer, with the body flap, control the orbiter during descent and landing. The orbiter has a large payload bay measuring 4.6 m by 18.3 m comprising most of the fuselage.

Three Space Shuttle Main Engines (SSMEs) are mounted on the orbiter's aft fuselage in a triangular pattern. The three engines can swivel 10.5 degrees up and down, and 8.5 degrees from side to side during ascent to change the direction of their thrust and steer the Shuttle as well as push. The orbiter structure is made primarily from aluminum alloy, moreover a Thermal Protection System (TPS) covers the outside of the Orbiter, protecting it from the cold soak of -121 °C (-250 °F) in space to the 1649 °C heat of re-entry, although the engine thrust structure is made from titanium (alloy).

All Space Shuttle missions are launched from Kennedy Space Center.

The vehicle starts encountering more significant air density in the lower thermosphere at about 120 km), at around Mach 25 (8.2 km/s). The vehicle is controlled by a combination of RCS thrusters and control surfaces, to fly at a 40 degree nose-up attitude, producing high drag, not only to slow it down to landing speed, but also to reduce re-entry heating. In addition, the vehicle needs to bleed off extra speed before reaching the landing site. This is achieved by performing s-curves at up to a 70 degree roll angle.

The orbiter's maximum glide ratio/lift-to-drag ratio varies considerably with speed, ranging from 1:1 at hypersonic speeds, 2:1 at supersonic speeds and reaching 4.5:1 at subsonic speeds during approach and landing. Actually, in the first part of the descent phase the orbiter works like a capsule, in fact, as already told before, the angle initial angle of attack is about 40 degrees. Later, in the lower atmosphere, the orbiter flies much like a conventional glider or airplane, so it can be guided and controlled.

The Space Shuttle mission design trajectory and the different phases on the re-entry has already been illustrated in the Figure 1-1.
More in details, next Figure 2-8, Figure 2-9 and Figure 2-10 show the main features of Space Shuttle re-entry trajectory; in particular the Reynolds number, the Angle of attack and Altitude are reported in function of the Mach number. Note that, as already told before, the AoA is about 40 deg. for the higher part of the trajectory till the Mach number is equal to about 10 (i.e. Altitude ≈ 50 km, see Figure 2-9 and Figure 2-10). In this part of the trajectory the re-entry can be considered ballistic (capsule-like), while in the lower part the vehicle is guided by means of the control surfaces.

**Figure 2-8** Space Shuttle’s trajectory: Reynolds number vs. Mach number.

**Figure 2-9** Space Shuttle’s trajectory: AoA vs. Mach number.
2.3. **ORION Crew Exploration Vehicle (CEV)**

Orion CEV is a spacecraft design currently under development by the United States space agency NASA. Each Orion spacecraft will carry a crew of four to six astronauts, and will be launched by the Ares I, a launch vehicle also currently under development. Both Orion and Ares I are elements of NASA’s Project Constellation, which plans to send human explorers back to the Moon by 2020, and then onward to Mars and other destinations in the solar system. On August 31, 2006, NASA awarded Lockheed Martin (LM) the contract to design, develop, and build Orion.

Orion will launch from Launch Complex 39 at Kennedy Space Center, the same launch complex that currently launches the Space Shuttle. NASA will use Orion spacecraft for its human spaceflight missions after the last Shuttle orbiter is retired in 2010. The first Orion flight is scheduled for September 2014 with future flights to the International Space Station. If commercial orbital transportation services are unavailable, Orion will handle logistic flights to the Station. After that, Orion is to become a key component of human missions to the Moon and Mars, see also reference 2.

The proposal to create the Orion spacecraft was partly a reaction to the Space Shuttle Columbia disaster, the subsequent findings and report by the Columbia Accident

![Figure 2-10 Space Shuttle’s trajectory: Altitude vs. Mach number.](image)
Investigation Board (CAIB), and the White House's review of the American space program. The Orion spacecraft effectively replaced the conceptual Orbital Space Plane (OSP), which itself was proposed after the failure of the Lockheed Martin X-33 program to produce a replacement for the Space Shuttle.

The Orion spacecraft described here should not be confused with theoretical spaceship designs from Project Orion in the 1950’s in which nuclear explosions were to be used for propulsion. The Orion spacecraft described here uses a conventional (non-nuclear) propulsion system.

The Orion Crew and Service Module (CSM) stack consists of two main parts: a conical Crew Module (CM), and a cylindrical Service Module (SM) holding the spacecraft’s propulsion system and expendable supplies. Both are based substantially on the Apollo Command and Service Modules (Apollo CSM) flown between 1967 and 1975, but include advances derived from the Space Shuttle program.

![Orion-CEV Expanded View](image)

**Figure 2-11 CEV-Spacecraft configuration.**

The Orion Crew Module will hold four to six crew members, compared to a maximum of three in the smaller Apollo. Despite its conceptual resemblance to the 1960s-era Apollo, Orion's CM will use several improved technologies, including, as example, the use materials derived from Space Shuttle experience and a trunc base for the docking with ISS.
An important feature that may be introduced in the Orion crew module is a new system employing a combination of parachutes and airbags for capsule recovery and the partial reusability.

The Orion Crew Module is a 57.5° frustum shape, similar to that of the Apollo Command Module. As projected, the crew module will be 5.03 m in diameter and 3.30 m in length, with a mass of about 8.5 metric tons. It is to be built by the Lockheed Martin Corporation. It will have more than 2.5 times the volume of an Apollo capsule, which had an interior volume of 5.9 m³ and will carry four to six astronauts.

**Figure 2-12 CEV-Crew Module.**

Like its Apollo predecessor, the Orion Service Module has a rough cylindrical shape, but unlike its Apollo predecessor, the new Orion Service Module will be larger in diameter, shorter, and lighter and it is called ARES I.

A staged approach to human exploration beyond low Earth orbit (LEO). It recommends that Project Constellation be carried out in three distinct phases, called "Stages". These are:

- Stage 1 – "Features the development of a new crew exploration vehicle (CEV), the completion of the International Space Station (ISS), and an early retirement of the Shuttle Orbiter. Orbiter retirement would be made as soon as the ISS U.S.
Core is completed (perhaps only 6 or 7 flights) and the smallest number of additional flights necessary to satisfy our international partners’ ISS requirements. Money saved by early Orbiter retirement would be used to accelerate the CEV development schedule to minimize or eliminate any hiatus in U.S. capability to reach and return from LEO."

- Stage 2 – "Requires the development of additional assets, including an uprated CEV capable of extended missions of many months in interplanetary space. Habitation, laboratory, consumables, and propulsion modules, to enable human flight to the vicinities of the Moon and Mars, the Lagrange points, and certain near-Earth asteroids."

- Stage 3 – "Development of human-rated planetary landers is completed in Stage 3, allowing human missions to the surface of the Moon and Mars beginning around 2020."

Figure 2-13 and Figure 2-14 show the CEV re-entry trajectory in terms of Altitude and angle of attack with respect to the velocity. Note that different trajectories are planned: from ISS (i.e. from LEO orbit), from interplanetary return (i.e. from Moon or Mars), ballistic or guided, 28 deg and 20 deg. of angle of attack, roughly bounding the range of the trim expected for the CEV.

![Figure 2-13 CEV’s re-entry trajectory: Altitude vs. velocity.](image)
### 2.4. Unmanned Space Vehicle (USV)

In the frame of the Italian Aerospace Research Program (PRORA), an experimental Unmanned Space Vehicle (USV) is being developed. The vehicle is a winged eXperimental Flying Test Bed (FTB), named FTB-X, that will be launched with the Vega rocket and re-enter the Earth’s atmosphere, thus allowing to perform a number of experiments on critical re-entry technologies. The vehicle will be used to demonstrate manoeuvrability and an improved stability in the upper atmosphere FTB-X, allowing performing long endurance re-entry patterns, beyond one hour, as compared to conventional lifting re-entry vehicles such as the Space Shuttle.

Moreover the test campaign will be used to test advanced thermo-structure concepts such as leading edges made of Ultra High Temperature Ceramics (UHTC), and to investigate the flowfield features during reentry in order to validate CIRA numerical (e.g. CFD) and experimental prediction capabilities.

In fact FTB-X will provide aerodynamic and aerothermodynamic flight data for correlation with ground test (e.g. Scirocco) results, thus providing new insight into the understanding of complex aerothermodynamic phenomena occurring in flight and improving predicting methodologies and extrapolation to flight theory.
A first version of FTB’s family, namely FTB-1, designed to perform a number of flights in transonic and low supersonic regime, was successfully launched on February 2007, by means of a stratospheric balloon. This FTB-1 configuration, together with its evolution FTB-2 (that differ only for the ventral fins that were adopted in FTB-1), represent the basis for the current design activities on the FTB-X vehicle.

The USV FTB-1 vehicle is a winged body, see Figure 2-15, with an overall length of 8 m, from the nose apex (without considering the air data boom) up to the base plate, and a weight of about 1300 kg (see reference 3). The front fuselage is characterized by a pointed nose constituted by a quasi-conical shape closed by a 1-cm radius hemisphere. The windside part of the forebody rapidly changes from a quasi-circular to a rounded-square section shape. The midfuselage is characterized by a quasi-constant section while the afterbody ends with a boat-tailed truncated base. The wing of the FTB-1 vehicle has a double delta shape with a main 45 deg sweepback leading edge and a strake with a 76 deg sweepback leading edge. The trailing edge has a sweepforward angle of 6 deg. To improve the vehicle’s lateral stability, the wing has a dihedral angle of 5 deg. Overall wing span is 3.562 m, while the strake root chord is 2.82 m. An elevon with both the functions of elevator and aileron is mounted on the wing.

![USV- FTB-1 Vehicle.](image)

To enhance directional stability and control two vertical tails have been adopted, with a dihedral angle of 40 deg, a sweepback angle of 45 deg and a span of 0.8 m. A pair of full-span movable rudders is also implemented for the control of directionality.

Moreover, in order to augment directional stability characteristics of the vehicle and to reduce possibilities of Dutch-Roll occurrence, a pair of full symmetric ventral fins has been added. These fixed fins are characterized by a 55 deg sweepback angle, a
root chord of 0.8 m with a taper ratio of 0.455, and a span of 0.418 m. They have been conceived in order to have the highest effectiveness with the lowest impact on structure.

The first USV FTB-1 mission has been aimed at experimenting the transonic flight of a re-entry vehicle, the aerodynamic experiment results are showed in the in the reference 4, while the aerodynamic database is presented in the reference 5. In the frame of a step-by-step approach, the FTB-1 will perform additional missions, see Figure 2-16, each of them with an increasing maximum Mach number (up to about 2), thus simulating the final portion of a typical re-entry trajectory.

The USV FTB-1 vehicle is basically composed by a Flying Test Bed (the FTB-1 vehicle) and a Carrier based on a stratospheric balloon. During the missions the balloon carries the FTB-1 up to the desired altitude (around 20 km for the first mission, see Figure 2-16) and then, after having established a cruise horizontal trajectory, releases it from the gondola. At this moment the FTB-1 vehicle starts its own flight following the designed trajectory.

The second DTFT mission (the orange line in Figure 2-16) will be flown aiming at obtaining, after the release from an altitude of about 25 km and the subsequent acceleration phase, an angle-of-attack sweep at a rather constant low supersonic Mach number of about 1.2, followed by a turn and a slow-down maneuver. The third and final DTFT mission of FTB-1 (the cyan line of Figure 2-16) will be an extension of the second mission to about Mach 2.

Figure 2-16 USV-FTB-1 mission and flight corridor.
The first DTFT mission was carried out on February 24th, 2007 from the airport of Tortoli, close to the city of Arbatax along the East coast of Sardinia (see reference 7). Figure 2-17 shows the FTB-1 vehicle and the gondola linked together and suspended from the launch machine on the launch pad during the pre-flight operations, and after the launch during the first moments of the ascent phase. The mission was performed with “Castore”, the first of the two twin spacecrafts developed and built within the USV Program. The second one was obviously named “Polluce”.

![Figure 2-17](image)

**Figure 2-17** FTB-1 and gondola suspended from the launch machine before flight (left) and during the ascent phase after launch (right).

Figure 2-18 reports the initial state of the vehicle at release and the actual mission profile in a Mach number-Altitude plane, while Figure 2-19 shows the temporal distribution of interesting flight parameters (Mach number, static pressure, angle of attack, angle of sideslip, left and right elevons deflection, and left and right rudders deflection) acquired in the sensor calibration window.

The mission started at 8:30 a.m. with the balloon lift-off bringing the FTB-1 vehicle up to an altitude of 20.1 km, before its release happened within the isolated sea polygon controlled by the Italian Air Force Fire Test Range in Salto di Quirra (PISQ). The mission ended at 10:30 a.m. with the splash-down of the FTB-1 vehicle.

As Figure 2-18 and Figure 2-19 show the flight controlled phases were performed without any significant off nominal conditions, up to 39 s. First, a vertical acceleration in nose-down attitude up to M=0.4 was performed reaching a stabilized flight condition; then, a wing levelled pull-up manoeuvre was initiated; finally, at M=0.7 the testing phase started at an angle-of-attack around 7.5 deg and at an angle-of-sideslip around zero. The sonic condition was reached at an altitude of 15 km, while the maximum
Mach number $M=1.08$ was attained at an altitude of 13.5 km, still in a controlled aerodynamic attitude.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitude</td>
<td>81.298483 deg</td>
</tr>
<tr>
<td>Latitude</td>
<td>39.951317 deg</td>
</tr>
<tr>
<td>Altitude</td>
<td>20117 m</td>
</tr>
<tr>
<td>North Velocity</td>
<td>$V_n$ -8.6 m/s</td>
</tr>
<tr>
<td>East Velocity</td>
<td>$V_e$ 16.1 m/s</td>
</tr>
<tr>
<td>Down Velocity</td>
<td>$V_d$ -2.65 m/s</td>
</tr>
<tr>
<td>Roll Angle</td>
<td>$\phi$ 2.32964 deg</td>
</tr>
<tr>
<td>Pitch Angle</td>
<td>$\theta$ -88.33727 deg</td>
</tr>
<tr>
<td>Yaw Angle</td>
<td>$\psi$ 36.36879 deg</td>
</tr>
<tr>
<td>Roll Rate</td>
<td>$\dot{\phi}$ -1.0597 deg/s</td>
</tr>
<tr>
<td>Pitch Rate</td>
<td>$\dot{\theta}$ 0.03009358 deg/s</td>
</tr>
<tr>
<td>Yaw Rate</td>
<td>$\dot{\psi}$ 0.0222762 deg/s</td>
</tr>
</tbody>
</table>

**Figure 2-18** Initial state of the FTB-1 vehicle at release (left) and actual mission profile (right).

**Figure 2-19** Temporal distribution of flight parameters.

Unfortunately, after these phases a failure of the Recovery System, with specific regard to its transonic-supersonic parachute first stage, caused an uncontrolled flight attitude, and the vehicle was seriously damaged during the off-design splash down. Of course, the lower part of the descent trajectory was not correctly performed, thus degrading the aerodynamic experiment since Reynolds number effect cannot be derived from the available flight data.
However, despite the loss of the vehicle “Castore”, the mission targets were partially achieved with the collection of about 2-million measurements related to flight data, housekeeping sensors, as well as aerodynamic experiment and structural experiment sensors.

Globally, the available DTFT mission results were enough satisfactory thus confirming the goodness of predictions and in-flight measurements, and are strongly contributing to the definition of the near-future second and third missions of FTB-1, which will be flown with “Polluce”.

This FTB-1 configuration represents the basis for the current design activities on the FTB-X vehicle, with the major constraint to allocate such a vehicle in the fairing of European Vega launcher.

Two types of missions have been identified within the re-entry flight mission envelope of the USV program.

**Orbital Re-entry Test (ORT)**

This mission shall be considered as the reference mission of the FTB_X vehicle. It shall consist of a complete re-entry flight from LEO orbit at 200 Km (to be confirmed by the launcher capability). An improved gliding re-entry and a high maneuvering capability, as compared to the reference re-entry vehicle Space Shuttle, characterized by moderate angle of attacks (down to and below than 20°) and a longer flight duration shall be developed in order to allow for more extended in-flight testing capabilities in high energy hypersonic flight conditions.

The LEO orbit inclination as well as the landing site and the re-entry trajectory footprint shall be suitably selected, in order to fulfill the safety requirements for ground population as per later requirement.

**Sub-orbital Re-entry Test (SRT)**

This type of mission of the FTB_X vehicle may be envisaged in the mission plan as an intermediate step for both design validation and risk mitigation purposes. Indeed, the mission plan shall be conceived in order to gradually achieve the ORT reference mission capability, as above defined.
### Design Requirements

<table>
<thead>
<tr>
<th>Aerodynamic Shape</th>
<th>Wing-body configuration with:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>- improved (L/D) ratio, up to a maximum value of [2-2.5] at Mach number in the range [8-10];</td>
</tr>
<tr>
<td></td>
<td>- reduced (W/Sr) down to 100 Kg/m²;</td>
</tr>
<tr>
<td></td>
<td>- as compared with reference lifting re-entry vehicles</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Primary Structure</th>
<th>Based on a lightweight design approach to comply with reduced (W/S) requirement.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Compliant with the most critical static and dynamic thermo-mechanical loads identified in the various mission phases.</td>
</tr>
<tr>
<td></td>
<td>Compatible with novel hot structures concepts based on advanced UHTC materials, either complete vehicle sub-assemblies (nose cap and wing leading edge) or embedded material samples.</td>
</tr>
<tr>
<td></td>
<td>Allowing integration of payloads according to the in-flight testing needs.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Thermal Protection System</th>
<th>Compliant with aero-thermal loads experienced during the atmospheric re-entry (up to 25 MJ/KG) and extended re-entry.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Design compatible with hot structures based on UHTC massive ceramics on the mostly exposed parts of the vehicle.</td>
</tr>
<tr>
<td></td>
<td>Design to minimize abrupt steps and gaps along the outer mould line within acceptable limits to avoid heat peaks.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Guidance, Navigation and Control</th>
<th>Provide autonomous on-board attitude and flight control capabilities in the overall re-entry phase, either using a reaction control system or aerodynamic control surfaces.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Handle large aerodynamic uncertainties and a wide Mach number range, from hypersonic to low subsonic as well as a defined set of off-nominal conditions (robust design).</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Avionic System</th>
<th>Main functions to be provided:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>i) on-board Data Handling, for both housekeeping and scientific data;</td>
</tr>
<tr>
<td></td>
<td>ii) Telemetry Tracking &amp; Communications;</td>
</tr>
<tr>
<td></td>
<td>iii) up-link and down-link functionalities to be properly defined;</td>
</tr>
<tr>
<td></td>
<td>iv) electrical Power provision for all subsystems for long mission.</td>
</tr>
<tr>
<td></td>
<td>Qualification defined according to applicable standards for the above defined space and re-entry missions.</td>
</tr>
<tr>
<td></td>
<td>Designed to be reconfigurable for different missions and experiments.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Instrumentation</th>
<th>Housekeeping instrumentation, for flight parameters acquisition and any system monitoring function;</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Scientific sensors, as resulting from in-flight experimentation requirements and related design activities.</td>
</tr>
<tr>
<td></td>
<td>Accommodation of a dedicated bay for self-standing passenger experiments (e.g. microgravity) to be investigated.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Launch System</th>
<th>ORT to be accomplished with the VEGA launcher, possibly using the AVUM fourth stage for orbit injection, orbiting and de-orbiting.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SRT to be accomplished either with the VEGA launcher or other available launcher or booster compatible with the vehicle design.</td>
</tr>
</tbody>
</table>

| Landing and Recovery | The vehicle shall be landed in the sea after a final parachuted descent phase. |
|----------------------| Location site to be selected according to the following factors: |
|                      | i) minimum time of recovery of the vehicle, in any case lower than 48 hours (TBC); |
|                      | ii) applicable aero-navigability regulations during the final descent and parachuted phase; |
|                      | iii) need of a logistic base to support the operations in the proximity of the recovery area. |

<table>
<thead>
<tr>
<th>Life cycle and turn-around</th>
<th>Each unit of the FTB_X vehicle shall be capable of performing more than one flight, either sub-orbital or orbital, with a target of 5 flights.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>After mission completion, the vehicle shall be made ready to operate in 12 months (TBC), after proper refurbishment and reconfiguration according to the new mission requirements.</td>
</tr>
</tbody>
</table>

The project requirements that drive the aerodynamic design are recalled, and the main impacts of each requirement on the aerodynamic configuration are emphasized.
Requirement 1: **High maneuverability**

The main consequence of this requirement is that the vehicle has to be design balancing the normal need to have a good stability with the possibility to perform the maneuvers that will be defined by the payloads and by the System itself. In fact it is known that if a vehicle is too stable, it is characterized by a lower capability to maneuver. This aspect must be taken into account in the definition of the wing position and of the control surface, in strong synergy with the flight mechanics analysis.

Requirement 2: **Capability to manage long missions, from 1 to 3 hours**

This requirement has a major impact on the sizing of the TPS; however, a long mission implies also the need to minimize as much as possible the heat loads all over a vehicle.

Requirement 3: **Moderate angles of attack with respect to Shuttle**

The requirement to have a moderate angle of attack is mainly related to the request to fly at high efficiency (see requirement 6). The heat flux distribution will obviously be strongly influenced by the angle of attack, being in general the heat load on the wind-side lower with respect to the one that would occur at 40 degrees of AoA (typical of the Shuttle); on the other hand, the heat flux on the lee-side will be higher due to the lower expansion.

Requirement 4: **Winged vehicle and similarity with FTB-1**

This requirement is very challenging, also considering the constraint in the maximum wing span due to the use of VEGA launcher (see requirement 8). The only way to realize a wing span at least equal to the one of FTB-1 would be to strongly reduce the dimension of USV; on the other hand, this would be in contrast with the need to have a sufficient volume to allocate both the payloads and the tanks for the stability control.

Requirement 6: **Higher efficiency with respect to Shuttle**

In order to realize the required high efficiency, it was necessary on one side to guarantee a good slenderness of the vehicle, and on the other side to fly at moderate angles of attack (see requirement 3).

Requirement 7: **Low wing load W/Sr**

In order to fulfil this requirement, the wing surface was increased, by increasing the chord of the wing, in order to balance the reduction of the wing span. However, it must be noted that for a hypersonic re-entry vehicle the most appropriate reference area
to be used is the plan form surface, rather than the wing, because a higher contribution to the lift coefficients comes from the fuselage.

**Requirement 8: Use of VEGA launcher**

This requirement has a strong impact on

- the overall dimension of the vehicle
- the maximum wing span
- the height of the fin

A preliminary FTB-X configuration, named FTB-X_1.1.1, was initially generated by scaling FTB-1 of about 65% and shortening the forward part of the fuselage, with the aim to make it compatible with the fairing of VEGA launcher, maintaining adequate available vehicle inner volume to allocate subsystems. Since with such scaling the wings tips resulted out of the fairing, they were cut; however, the wing mean chord has been increased to compensate the area reduction in order to reduce the wing loading; the normalized wing profiles have not been modified to maintain similarity with FTB-1.

![Figure 2-20 FTB-X/VEGA payload accommodation bay compatibility.](image)

Moreover, the FTB-X_1.1.2 configuration was obtained simply substituting the double fin with a single one.
In order to optimize the heat flux distribution over the nose and to increase the vehicle aerodynamic efficiency, the vehicle forebody has been moved downward, obtaining a flat bottomed configuration, typical of winged hypersonic vehicles (FTB-X_1.2.2).

A further modification was required in order to increase the available volume; to this aim, the fuselage was laterally enlarged by 5 cm for each side (FTB-X_1.3.2).

Starting from the FTB-X_1.3.2 configuration, a sensitivity analysis was made with the goal to improve the longitudinal and lateral-directional stability, as well as the aerodynamic efficiency of the vehicle.

Several changes have been made also for the wing, with respect to the starting one, in terms of planform shape and of position with respect to the vehicle nose. The strake has been removed and the sweep angle has been increased from 60° to 65° (deg). The strake was removed because the one resulting from the scaling of FTB-1 toward the 1.1.2 configuration is no longer effective, and a detailed redesign would be needed; a newly designed strake could be added in the future, depending on the confirmation of a specific landing requirement.

A higher sweep angle, instead, was needed in order to increase both the efficiency and the wing area and to assure best performance with respect to the supersonic drag and aerodynamic heating.

The final important change consisted of moving the wing forward by 50cm (FTB-X_3.9.2-FW50), in order to increase the Cm_y and to allow to pitch trim the vehicle with positive deflections of control surfaces, so improving the vehicle stability and controllability for major parts of the flight envelope; the final position of the wing will be however fixed after a further trade-off with the CoM position.
Definitely, the overall vehicle dimensions are summarized as follows:

- total length (tail included): 5.68 m;
- total height (tail included): 1.62 m;
- fuselage length: 5.31 m;
- maximum fuselage width: 0.90 m;
- maximum fuselage height: 0.90 m;
- wingspan: 2.14 m;
- wing area: 3.05 m$^2$;
- nose radius ($R_N$): 0.10 m;
- wing leading edge radius ($R_{WN}$): 0.04 m.

The fineness ratio of the fuselage is about 6 while the wing aspect ratio is 0.9.

The aerodynamic configuration features a compact body with rounded edge delta-like fuselage cross section and delta planform wing as basic shape.
Figure 2-22 Three view drawing of FTB-X-392-FW50 concept. Dimensions are in mm.

Figure 2-23 to Figure 2-26 summarize the FTB-X design trajectory in terms of the time histories of altitude, Mach number, AoA, while the Reynolds number (Re∞) was reported as function of Mach number in order to characterize the aerodynamics and aerothermodynamics of vehicle.

From Figure 2-24 it can be clearly seen how FTB-X guidance law significantly differs from that of Space Shuttle. In fact the Shuttle AoA is about 40 deg for most of the trajectory, while FTB-X modulates its attitude, during critical phase, taking the AoA profile as close as possible to that corresponding to the maximum aerodynamic efficiency, compatibly with the maximum heat flux that the vehicle thermal shield could sustain. Actually, FTB-X will be guided in the higher part of the trajectory, also in rarefied flow conditions. This is one of the most challenging technological problems to be solved in USV program. As a consequence, this reentry scenario results in an aero-heating environment that must be accurately predicted for a reliable TPS design and the aerodynamic features has to be deeply solved also in rarefied flow conditions, since, differently from the other re-entry vehicle, in high altitude a FTB-X should be controlled.
Figure 2-23 Preliminary reference FTB-X re-entry trajectory; Altitude vs Time.

Figure 2-24 Preliminary reference FTB-X re-entry trajectory; AoA vs Time.
Figure 2-25 Preliminary reference FTB-X re-entry trajectory; Mach vs Time.

Figure 2-26 Preliminary reference FTB-X re-entry trajectory; Reynolds number vs Mach number.

Finally, Figure 2-27 shows the Maximum heat flux versus the Mach number.
Figure 2-27 Preliminary reference FTB-X re-entry trajectory; Maximum heat flux vs mach number
Chapter 3  Hypersonic Rarefied Flows

3.1. Hypersonic Flow Regime

There is a conventional rule of thumb that defines hypersonic aerodynamics as those flows where the Mach number, $M$, is greater than 5. However, this is no more than just a mule of thumb. Hypersonic flow is defined as that regime where certain physical flow phenomena become progressively more important as the Mach number is increased to higher values.

First of all a the hypersonic flow is characterized by a thin shock layer. According to the oblique shock theory, for a given flow deflection angle, the density increases across the shock wave becomes progressively larger as the Mach number is increased. At higher density the mass flow behind the shock can more easily “squeeze through” smaller areas. Per flow over a hypersonic body, this means that the distance between the body and the shock wave can be small. The flowfield between the shock wave and the body is declined as the shock layer, and for hypersonic speeds this shock layer can be quite thin. If high-temperature, chemically reacting effects are included (the detail of this effect will be discussed below), the shock wave angle will be even smaller. It is a basic characteristic of hypersonic flows that shock waves lie close to the body, and that the shock layer is thin. In turn, this can create some physical compilations, such as the merging of the shock wave itself with a thick, viscous boundary layer growing from the body surface, a problem which becomes important at low Reynolds numbers, for example at high altitude of the re-entry trajectory, where the low values of Re are essentially caused by the low density.

Taking a look to Figure 3-1, at hypersonic Mach numbers, the shock layer over the blunt nose is also very thin, with a small shock-detachment distance $d$. 
In the nose region, the shock wave is highly curved. It is well known that the entropy of the flow increases across a shock wave, and the stronger the shock, the larger the entropy increase. A streamline passing through the strong, nearly normal portion of the curved shock near the centreline of the flow will experience a larger entropy increase than a neighboring streamline which passes through a weaker portion of the shock further away from the centerline. Hence, there are strong entropy gradients generated in the nose region; this ‘entropy layer’ flows downstream, and essentially wets the body for large distances from the fuse, as shown in Figure 3-1. The boundary layer along the surface grows inside this entropy layer, and is affected by it. Since the entropy layer is also a region of strong vorticity, (see Crocco’s theorem) this interaction is sometimes called a “vorticity interaction”.

Another important feature of the hypersonic regime is the so called “viscous interaction”. A high-velocity, hypersonic flow contains a large amount of kinetic energy, when the flow is slowed by viscous effects within the boundary layer, the lost of kinetic energy is transformed (in part) into internal energy (viscous dissipation). As a consequence the temperature increases within the boundary layer and the boundary layer thickness grows essentially as:

$$\delta \propto \frac{M^2}{\sqrt{Re}}$$

instead of,
\[
\delta \propto \frac{1}{\sqrt{Re}}
\]

as the law of “classic” aerodynamics. For a complete analysis see reference 8. The thick boundary layer in hypersonic flow can exert a major displacement effect on the inviscid flow outside the boundary layer, causing a given body shape to appear much thicker than it really is. Due to the extreme thickness of the boundary-layer flow, the outer inviscid flow is greatly changed: the changes in the inviscid flow in turn feed back to attest the growth of the boundary layer. This major interaction between the boundary layer and the outer inviscid flow is called *viscous interaction*. Viscous interactions have important effects on the surface pressure distribution hence lift, drag, and stability on hypersonic vehicles. Moreover, skin friction and heat transfer are increased by viscous interactions.

The high temperatures caused by dissipation of the kinetic energy in the boundary layer by means of viscous interactions (i.e., skin friction) excite vibrational energy internally within molecules, and dissociation and even ionization within the gas. The boundary layer is not the only region at high temperature of an hypersonic vehicle; in the shock layer in the nose region, or in the leading edge region for a winged vehicle the temperature raises and chemical reactions occur. The entire shock layer can be dominated by chemical reacting flows. This is the most important difference with classic aerodynamic: the composition of the gas is not constant in space and time and is characterized by variable specific heats. To summarize as the gas temperature increases:

- The vibrational energy of the molecule become exited, so \( c_p \) and \( c_v \) and, of course \( \gamma \equiv c_i / c_v \), become function of temperature. For air these effect is important above a temperature of 800 K.
- As the gas temperature increases, chemical reactions occur. For air at 1 atm of pressure, \( O_2 \) dissociation begins at about 2000 K \( (O_2 \rightarrow 2O) \) and the molecular oxygen is totally dissociated at 4000 K, at this temperature \( N_2 \) begins to dissociate \( (N_2 \rightarrow 2N) \) and is totally dissociated at 9000 K. As the temperature increases more than 9000 K, the gas becomes to ionize \( (N \rightarrow N^+ + e^- \text{ and } O \rightarrow O^+ + e^-) \) that can be considered as a plasma, so electro-magnetic field influence has to be taken into account.
The high temperature effects are also called “real gas effects”. These effects influence the global aerodynamic and a great aerodynamic heating occurs (convective, diffusive and radiative heat fluxes). The aerodynamic heating dominates the design of an hypersonic vehicle.

The Navier-Stokes equations for hypersonic flow regime must take into account all these effects, so the equations are more complex than the classic aerodynamic ones. In particular, Equation 3-1 (balance and conservation equations) and Equation 3-2 (phenomenological equations) show, in integral form, the modified terms of Navier-Stokes equations (red) and the additional ones (green for chemical reactions effects, purple for electro-magnetic effects). Moreover, the thermodynamic model needed to close the set of equations, varies for real gas effects (see reference 8).

\[ \frac{\partial}{\partial t} \int_{\Omega} \rho dV + \int_{\partial \Omega} n \cdot (\rho \vec{v}) dS = \int_{\Omega} \sum_j \Omega_j dV + \int_{\partial \Omega} n \cdot \vec{J}_m dS \]

\[ \frac{\partial}{\partial t} \int_{\Omega} \rho \vec{v} dV + \int_{\partial \Omega} n \cdot (\rho \vec{v} \vec{v}) dS = -\int_{\partial \Omega} (\rho \vec{v} \cdot \vec{\nabla} V + \vec{J}_B \times \vec{B}) dS - \int_{\Omega} \vec{Q}_v dV + \int_{\partial \Omega} n \cdot \vec{E} dS \]

\[ \frac{\partial}{\partial t} \int_{\Omega} \rho E dV + \int_{\partial \Omega} n \cdot (\rho E \vec{v}) dS = \int_{\Omega} \sum_j \Omega_j E dV + \int_{\partial \Omega} n \cdot \vec{Q}_m dS + \int_{\partial \Omega} (\vec{J} \cdot \vec{E}) dS \]

\[ \int_{\Omega} (\vec{\nabla} \times \vec{B}) dV = -\frac{\partial}{\partial t} \int_{\partial \Omega} \vec{E} dS \]

**Equation 3-1** Navier-Stokes equations.
The aerothermodynamics of a Space re-entry vehicle is also characterised at high altitude by the effects of rarefaction. This effect is the main object of the present work and the theoretical approach is introduced in the next paragraphs.

### 3.2. Rarefied Flow and Navier-Stokes Breakdown

A gas mixture is made up of individual molecules, for the air mainly Oxygen and Nitrogen, which are in random motion. Each molecule move a certain distance, and then collide one of the neighbouring molecules, after it will move again, and collide again with another molecule. This process continue indefinitely. The average distance between travelled by molecules between collisions is called **mean free path** $\lambda$.

More in details, following Bird (reference 10), we will introduce some basic definitions and a more accurate definition of $\lambda$.

The basic quantities associated with the molecular model are the number of molecules per unit volume and the mass, size, velocity, and internal state of each molecule. These quantities must be related to the mean free scales of the effects due to the collisional interactions among the molecules. Also, since the results from the molecular approach will generality be presented in terms of the macroscopic quantities, we must establish the formal relationship between the microscopic and macroscopic quantities. For reasons of simplicity and clarity the discussion in this section will be...
restricted to a gas consisting of a single chemical species in which all molecules are assumed to have the same structure. Such a gas may called a simple gas.

The number of molecules in one mole of a gas is a fundamental physical constant called Avogadro number’s \( N \). Avogadro’s law also states that the volume occupied by one mole of any gas at a particular temperature and pressure is the same for all gases. The number of molecules per unit volume, or number density \( n \), of a gas depends on the temperature and pressure, but is independent of the composition of the gas. The mass \( m \) of a single molecule is obtained by dividing the molecular weight \( \omega \)

\[
\text{Equation 3-3} \quad m = \frac{\omega}{N} = \omega m_u
\]

where \( m_u \) is the atomic mass constant. The average volume available to a molecule \( 1/n \), so the mean molecular spacing \( \delta \) is given by

\[
\text{Equation 3-4} \quad \delta = n^{-1/3}
\]

A hard elastic sphere of diameter \( d \) provides an over-simplified but useful model of a molecule. Then, as shown in Figure 3-2 two molecule collide if their trajectories are such that the distance between their centres decreases to \( d \). The total collision cross-section for these molecule is therefore

\[
\text{Equation 3-5} \quad \sigma_T = \pi d^2
\]

Figure 3-2 Collision between two hard spheres of diameter \( d \).
The proportion of the space occupied by a gas that actually contains a molecules is of the order of \((\delta/d)^3\). Equation 3-4 shows that, for sufficiently low densities, the molecular spacing \(\delta\) is large compared with the effective molecular diameter \(d\). Under these circumstances, only an extremely small proportion of space is occupied by molecules and each molecule will, for the most part, be moving outside the range of influence of other molecules. Moreover, when it does suffer a collision, it is overwhelmingly likely to be a \textbf{binary collision} involving only one other molecule. This situation may be characterized by the condition

\[
\delta \gg d
\]

and defines a \textit{dilute gas}. The time-scale of the macroscopic processes is set by the \textbf{mean collision time} which is, by definition, the mean time between the successive collisions suffered by any particular molecule. The reciprocal of this quantity in more common use and is called the \textbf{mean collision rate} or \textbf{collision frequency} \(\nu\) per molecule. In the derivation of an expression for this quantity we will fix our attention on a particular molecule which will be referred to as the test molecule. The velocities of the other, or field, molecules are distributed in some unspecified manner. Consider those field molecules with velocity between \(c\) and \(c + dc\). These will be referred to as molecules of class \(\mathcal{C}\) and their number density is denoted by \(n\). If the velocity of the test molecule is \(c_t\), the relative velocity between the test molecule and the field molecules of class \(\mathcal{C}\) is \(c_r = c_t - c\). Now choose a frame of reference in which the test molecule moves with velocity \(c_t\) while the field molecules of class \(\mathcal{C}\) are stationary. Then, over a time interval \(t\) much shorter than the mean collision time, the test molecule would collide with any field molecule which has its centre within the cylinder of volume \(\sigma c_r \Delta t\) as shown in Figure 3-3.
Figure 3-3 Effective volume swept out by moving test molecule among stationary field.

The probability of a collision between the test molecule and a molecule of class \( c \) in the time interval \( t \) is therefore \( \Delta n \sigma_T \Delta t \). When collision do occur, the cylinder swept out by the collision cross-section along the trajectory becomes distorted. However, for a dilute gas in which only a very small proportion of the trajectory is affected by collisions, the restriction on \( \Delta t \) can be removed and the number of collisions per unit time with a class \( c \) molecule is \( \Delta n \sigma_T c_r \). The mean collision rate is obtained by summing over all velocity classes and therefore over all values of \( cr \). That is,

\[
\text{Equation 3-7} \quad \nu = \sum (\Delta n \sigma_T c_r) = n \sum ((\Delta n/n) \sigma_T c_r)
\]

And, since \( \Delta n/n \) is the fraction of molecules with cross-section \( \sigma_T \) and velocity \( c_r \),

\[
\text{Equation 3-8} \quad \nu = n \sigma_T c_r
\]

A bar over a quantity or expression denotes the average value over all molecules in the sample. For hard sphere molecules, this becomes

\[
\text{Equation 3-9} \quad \nu = n \sigma_T c_r = n \sigma_T c_r
\]

The total number of collisions per unit time per unit volume of gas is therefore given by

\[
\text{Equation 3-10} \quad N_c = \frac{1}{2} n \nu = \frac{1}{2} n \sigma_T c_r
\]

The symmetry factor of one half is introduced because each collision involves two molecules.
As already told before, the mean free path is the average distance travelled by a molecule between collisions. It is defined in a frame of reference moving with the stream speed of the gas and is therefore equal to the mean thermal speed $c^\prime$ of the molecule divided by the collision frequency, i.e.

Equation 3-11

$$\lambda = c^\prime / \nu = \left[ n(\sigma_T c^\prime, l c^\prime) \right]^{-1}$$

Or, for the constant cross-section hard sphere case,

Equation 3-12

$$\lambda = \left[ (c^\prime / c^\prime) \pi d^2 n \right]^{-1}$$

At sea level for air $\lambda = 6.632 \cdot 10^{-8} m$, so there is a sufficient number of molecules within the smallest volume of flow and the gas can be considered continuum. In these case the balance and conservations equations can be written, since the theorems of mathematical analysis are valid as the phenomenological relations of Equation 3-2.

As the considered altitude raises, the set of equation 3-2 reaches the limit of validity before the breakdown of continuum hypothesis. Also continuum hypothesis is no longer valid, so a molecular description of the gas is needed; this is also called Navier-Stokes breakdown. As example, at 100 km of altitude $\lambda = 0.3 m$.

There are certain hypersonic applications which involve low density flow, generally involving flight at high altitudes. For example, the flow in the nose region of the Space Shuttle cannot be properly treated by purely continuum assumptions for altitudes above 92 km. For any given flight vehicle, as the altitude progressively increases (hence the density decreases and $\lambda$ increases), the assumption of a continuum flow becomes tenuous. An altitude can be reached where the conventional viscous flow no slip conditions begin to fail. Specifically, at low densities the flow velocity at the surface, which is normally assumed to be zero due to friction, takes on a finite value. This is called the velocity slip condition. In analogous fashion, the gas temperature at the surface, which is normally taken as equal to the surface temperature of the material, now becomes something different. This is called the temperature slip condition. At the onset of these slip effects, the governing equations of the flow are still assumed to be the familiar continuum flow equations, except with the proper velocity and temperature slip conditions utilized as boundary conditions. These boundary conditions will be treated later.

As already told before, as the altitude continues to increase, there comes a point where the continuum flow equations themselves are no longer valid, and methods from
kinetic theory must be used to predict the aerodynamic behaviour. Finally, the air density can become low enough that only a few molecules impact the surface per unit time, and after these molecules reflect from the surface, they do not interact with the incoming molecules. Definitively, the molecules don’t interact significantly in a statistical sense between themselves, but the only interaction with the wall has to be considered. This is the regime of free molecular flow. For the Space Shuttle, the free molecular regime begins about 150 km.

Therefore, in a simplified sense, we visualize that a hypersonic vehicle moving from a very rarefied atmosphere to a denser atmosphere will shift from the free molecular regime, to a transitional regime, where slip flow effects are to be taken into account and where the molecules cannot be considered infinite but (differently from free molecular regime) they interact between themselves. After these regime, continuum hypothesis take place.

The degree of rarefaction of a gas is generally expressed through the Knudsen number (Kn) which is the ratio of the mean free path \( \lambda \) to the characteristic dimension \( L \), i.e.

\[
\text{Equation 3-13} \quad Kn = \frac{\lambda}{L}
\]

The traditional requirement for the Navier-Stokes equations to be valid is that the Knudsen number should be less than 0.1. This can be misleading if \( L \) is chosen to be some overall dimension of the flow in order to define a single overall Knudsen number for the complete flow. The limit can be specified precisely if a local Knudsen number is defined with \( L \) as the scale length of the macroscopic gradient \( a \); e.g.

\[
\text{Equation 3-14} \quad L = \frac{a}{da/dx}
\]

\( \lambda \) according to the Kinetic theory can be written:

\[
\text{Equation 3-15} \quad \lambda = 1.25\sqrt{\gamma} \frac{V}{\sqrt{2RT}}
\]

The Knudsen numeric in terms of free stream condition can also be written as.
Equation 3-16 \[ Kn_{in} = 1.25 \sqrt{\frac{\gamma}{\gamma - 1}} \frac{M}{Re} \]

Note that this equation is an approximation since Mach and Reynolds number have sense in continuum regime.

Another way to write the Knudsen number “locally” is to use as characteristic length the local boundary layer thickness \( \delta \), this is also called Tsien number:

Equation 3-17 \[ Kn_{\delta} = \frac{\lambda}{\delta} \]

The error in the Navier-Stokes result is significant in the region of the flow where the appropriately defined local number exceeds 0.1, and the upper limit on the local Knudsen number at which the continuum model must be replaced by the molecular model may be taken to be 0.2.

However, slip effects can be included when \( Kn > 0.03 \). The effects of the free molecular flow being, according to Moss (see reference 9), around 50. Hence the transitional regime is essentially contained within \( 0.03 < Kn < 50 \).

The transport terms vanish in the limit of zero Knudsen number and the Navier-Stokes equation then reduce to the inviscid Euler equation. The flow is then isentropic from the continuum viewpoint, while the equivalent molecular viewpoint is that the velocity distribution function is everywhere of the local equilibrium or Maxwellian form. The opposite limit of infinite Knudsen number is the collisionless or free-molecule flow regime. These Knudsen number limit on the conventional mathematical formulations are shown schematically in Figure 3-4.

<table>
<thead>
<tr>
<th>Boltzmann equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Continuum</td>
</tr>
<tr>
<td>EULER</td>
</tr>
</tbody>
</table>

\[ \approx 80 Km \] \approx 150 Km

Figure 3-4 The Knudsen number limits on the mathematical models.
Because $Kn = \lambda / L$ is the governing parameter, that altitude below which we have continuum flow is greater or lesser as the characteristic length $L$ is larger or smaller. Hence, large vehicles experience continuum flow to higher altitudes than small vehicles. Moreover, if we let the characteristic length be a running distance $x$ from the nose or leading edge of the vehicle, then $Kn L/x$ becomes infinite when $x=0$. Hence, for any vehicle at any altitude, the flow immediately at the leading edge is governed by low-density effects. For most practical applications in aerodynamics, this leading edge region is very small, and is usually ignored. However, for high-altitude hypersonic vehicles, the proper treatment of the leading edge flow by low-density methods can be important. This problem will be treated in the Chapter 6.

### 3.3. Basic kinetic Theory and Boltzmann Equation

Kinetic theory and Boltzmann equation represent the way to describe for all the values of Knudsen number (see Figure 3-4) the mechanics of fluid and in this paragraph a brief introduction is presented.

A gas flow would be completely described, in a classical sense, by listings of the position, velocity and internal state of every molecule at a particular instant. The number of molecules in a real gas is so large that such a description is unthinkable, and we must resort to a statistical description in terms of probability distributions. A number of distinct velocity distribution functions are employed in kinetic theory and possible confusion may be avoided by a general review of the relationships between them.

We will commence by defining the single particle distribution function in velocity space. Consider a sample of gas that is homogeneous in physical space and contain $N$ identical molecules. A typical molecule has a velocity $\mathbf{e}$ with component $u$, $v$, and $w$ in the direction of the Cartesian axes $x$, $y$, and $z$. Just as $x$, $y$, and $z$ define a space called physical space, $u$, $v$, and $w$ define velocity space, as shown in Figure 3-5.
Figure 3-5 Typical molecule and element in velocity space.

Each molecule can be represented in this space by the point defined by its velocity vector. The velocity distribution function \( f(\mathbf{c}) \) is then defined by:

\[
\text{Equation 3-18} \quad dN = N f(\mathbf{u}, \mathbf{v}, \mathbf{w}) d\mathbf{u} d\mathbf{v} d\mathbf{w}
\]

where \( dN \) is the number of molecules in the sample with velocity components \( u \) to \( u+du \), \( v \) to \( v+dv \), and \( w \) to \( w+dw \). The product \( du \ dv \ dw \) may be identified as a volume element in velocity space and is denoted by \( dc \). An alternative form of \( dN \) is, therefore,

\[
\text{Equation 3-19} \quad dN = N f(\mathbf{u}, \mathbf{v}, \mathbf{w}) d\mathbf{u} d\mathbf{v} d\mathbf{w} = N f(\mathbf{c}) d\mathbf{c}
\]

and this need not be restricted to Cartesian coordinates. The functional statement is usually omitted so that \( f(\mathbf{c}) \) becomes simply \( f \). Also, since both \( dN \) and \( N \) refer to the molecules in the same volume of physical space, the number density may be used in place of the number. Therefore, the fraction of molecules within the velocity space element \( dc \) is

\[
\text{Equation 3-20} \quad \frac{dn}{n} = f d\mathbf{c}
\]

Since every molecule is represented by a point in velocity space:

\[
\text{Equation 3-21} \quad \int_{-\infty}^{\infty} f d\mathbf{c} = N/N = 1
\]
This distribution function is therefore a normalized function such that its integration over all velocity space yields unity. Note that $f$ can never be negative and must either have finite bounds in velocity space or tend to zero as $v$ tends to infinity.

The macroscopic quantities were defined in terms of averages over the molecular velocities. These averages may be established as instantaneous, time, or ensemble averages over the molecules in an element of physical space. These molecules may be regarded as constituting a homogeneous gas sample and the single particle distribution function in velocity space provides an appropriate description. In order to relate the macroscopic properties to this distribution function, we must determine the relationship between the function and the average value of any molecular quantity $Q$. This quantity is either a constant or a function of the molecular velocity. The mean value principle gives:

$$\text{Equation 3-22} \quad \bar{Q} = \frac{1}{N} \int Q dN$$

And substituting $dN$ from Equation 3-22, omitting the functional statements:

$$\text{Equation 3-23} \quad \bar{Q} = \frac{1}{N} \int Q f d\zeta$$

This process is often referred to as establishing a **moment of the distribution function** and the macroscopic properties are referred to as moments of the distribution function. For example, the stream velocity can be written as:

$$\text{Equation 3-24} \quad \bar{c} = \frac{1}{N} \int c f d\zeta$$

The macroscopic flow properties are generally functions of position and time, and it is sometimes desirable to express the explicit dependence of the distribution function on the position vector in velocity space $\mathbf{r}$, and the time $t$. Just as $d\mathbf{r}$ has been used to denote a volume element in velocity space, a volume element in physical space may be denoted by $d\mathbf{r}$. The product $d\mathbf{c}d\mathbf{r}$ then denotes a volume element in **phase space**, which is the multi-dimensional space formed by the combination of physical space and velocity space. The single particle distribution function in phase space $\phi(\zeta, \mathbf{r}, t)$ is defined by:
Equation 3-25
\[ dN = \phi(\mathbf{c}, \mathbf{r}, t) d\mathbf{c} d\mathbf{r} \]

with \( dN \) now representing the number of molecules in the phase space element \( d\mathbf{c} d\mathbf{r} \). In Cartesian coordinates, \( d\mathbf{c} d\mathbf{r} \) becomes \( du dv dw dx dy dz \) and \( dN \) is the number of molecules with velocity components ranging from \( u \) to \( u+du \), \( v \) to \( v+dv \), and \( w \) to \( w+dw \) and spatial coordinates ranging from \( x \) to \( x+dx \), \( y \) to \( y+dy \), and \( z \) to \( z+dz \). Note that \( \phi \) defines the number rather than the fraction of molecules in the phase space element. It has not been normalized and an integration over the whole phase space yields the total number of molecules in the system \( N \), rather than unity.

If the distribution function in velocity space \( f(\mathbf{c}) \) is applied to the physical space element \( dr \), the number of molecules \( N \) in Equation 3-19 represents the total number of molecules in \( dr \), while \( dN \) represents the number of molecules in the phase space element \( d\mathbf{c} d\mathbf{r} \). We can then write:

Equation 3-26
\[ dN = N f(\mathbf{c}) d\mathbf{c} = \phi(\mathbf{c}, \mathbf{r}, t) d\mathbf{c} d\mathbf{r} \]

and, since the number density in the phase space element is \( N/dr \),

Equation 3-27
\[ n(r) f(\mathbf{c}) = \phi(\mathbf{c}, \mathbf{r}, t) \]

Therefore, when \( f(\mathbf{c}) \) is used in a context in which it also depends on \( r \) and \( t \), we have

Equation 3-28
\[ nf \equiv \phi \]

We will take advantage of this identity to use \( f \) exclusively.

The most specific distribution function is that for all \( N \) molecules in the system. At any instant, a complete system of monatomic molecules can be represented by a point in \( 6N \) dimensional phase space. If we consider a large number or ensemble of such systems, the probability of finding a system in the volume element \( dc_1 dr_1 dc_2 dr_2 ... dc_N dr_N \) about the phase space point \( c_1, r_1, c_2, r_2, ..., c_N, r_N \) is

Equation 3-29
\[ F^{(N)}(c_1, r_1, c_2, r_2, ..., c_N, r_N, t) dc_1 dc_2 ... dc_N dr_1 dr_2 ... dr_N \]

thus defining the \( N \) particle distribution function \( F^{(N)} \). The subscript denotes the number of the molecule. A reduced distribution function \( F^{(R)} \) for \( R \) of the \( N \) molecules is defined by:
\[ F^{(R)}(\xi_1, r_1, \xi_2, r_2, \ldots, \xi_R, r_R, t) = \int \int F^{(N)}(c_1, d c_2, \ldots, d c_N, d r_1, d r_2, \ldots, d r_N) \]

In particular, the single particle distribution function \( F^{(1)}(\xi_1, t) \) is obtained by setting \( R=1 \). The probability of finding molecule number 1 in the phase space element \( d\xi_1 dr_1 \) at time \( t \) is \( F^{(1)}(\xi_1, t) \) irrespective of the positions of the other \( N-1 \) molecules. Since the molecules are indistinguishable, the number of molecules in the phase space element at time \( t \) is \( NF^{(1)} \). We therefore have

\[ NF^{(1)} \equiv \phi \]

And \( F^{(1)} \) can be regarded as the normalized version of \( \phi \).

The two particle distribution function \( F^{(2)}(\xi_1, r_1, \xi_2, r_2, t) \) is of particular importance when considering binary collisions. The definition of a dilute gas requires that only a very small fraction of the space occupied by the gas actually contains a molecule. Therefore, in such a gas, it is generally assumed that the probability of finding a pair of molecules in a particular two particle configuration is simply the product of the probabilities of finding the individual molecules in the two corresponding one particle configurations. This requires

\[ F^{(2)}(\xi_1, r_1, \xi_2, r_2, t) = F^{(1)}(\xi_1, r_1) F^{(1)}(\xi_2, r_2) \]

and expresses the principle of molecular chaos. While the higher-order distribution functions are required for the study of dense gases, the single particle distribution function provides an adequate description of dilute gases.

If the molecules are diatomic or polyatomic, the dimensions of phase space are increased by the number of internal degrees of freedom. Also, if the molecules are not spherically symmetric, their orientations must be specified. In general, the dimensions of phase space are equal to the least number of scalar variables that are required to specify the position, velocity, orientation, and internal state of a molecule. Separate distribution functions are required for each species of a gas mixture. It is hardly surprising that most presentations of kinetic theory deal almost exclusively with gases of a single consisting of a single species of monatomic molecules.
3.3.1. The Boltzmann Equation

We have seen that the velocity distribution functions provide a statistical description of a gas on the molecular level. The next step is to establish the relationships between the distribution functions and the variables on which they depend. Ideally, the resulting equations would allow analytical solutions of problems in molecular gas dynamics.

The basic statistical mechanics equation for a gas is the Liouville equation which expresses the conservation of the $N$ particle distribution function in $6N$ dimensional phase space. This equation is not directly useful, since the description of a real gas flow in terms of $F^{(N)}$ is completely out of the question. However, just as a hierarchy of reduced distribution functions $F^{(R)}$ was defined by Equation 3-20, a hierarchy of equations called the BBGKY equations may be obtained through the repeated integration of the Liouville equation. The final equation in the hierarchy is for the single particle distribution function $F^{(1)}$ and is the only one to hold out some hope of solution for flows of engineering interest. This equation also involves the two particle distribution function but becomes a closed equation for $F^{(2)}$ when molecular chaos Equation 3-32 is assumed. Then, through Equation 3-31, this becomes an equation for the single particle distribution in phase space, and is equivalent to the equation that was originally formulated by Boltzmann (1872). The mathematical limits that define the validity of the Boltzmann equation are most precisely established through the derivation from the Liouville equation. On the other hand, the physical significance of each term in the equation is more readily appreciated if the single particle distribution function is used throughout a derivation from first principles. The latter procedure will be followed here and, for clarity and simplicity, the derivation will be restricted to a simple gas.

At a particular instant, the number of molecule in the phase space element $d\zeta dr$ is given by Equation 3-31 as $\phi d\zeta dr$. The identity Equation 3-28 enables this to be written $nf d\zeta dr$. If the location and shape of the element does not vary with time, the rate of change of the number of molecules in the element is

\[
\text{Equation 3-33} \quad \frac{d}{dt} \left(nf \right) d\zeta dr
\]

The processes that contribute to the change in the number of molecules within $d\zeta dr$ are illustrated in Figure 3-6. They are
Figure 3-6 Molecular flux to and from phase space element $d\mathbf{c}d\mathbf{r}$.

(i) The convection of molecules across the face of $d\mathbf{r}$ by the molecular velocity $\mathbf{c}$. The representation of the phase space element as separate volume elements in physical and velocity space emphasizes the fact that $\mathbf{c}$ and $\mathbf{r}$ are treated as independent variables. $e$ is regarded as a constant within $d\mathbf{r}$, and $de$ is regarded as being located at the point defined by $\mathbf{r}$.

(ii) The ‘convection’ of molecules across the surface of $d\mathbf{c}$ as a result of the external force per unit mass $F$. The effect of the acceleration $F$ on the molecules in $d\mathbf{c}$ is analogous to the effect of the velocity $\mathbf{c}$ on the molecules in $d\mathbf{r}$.

(iii) The scattering of molecules into and out of $d\mathbf{c}d\mathbf{r}$ as a result of intermolecular collision. The gas is assumed to be dilute. One consequence of this assumption is that a collision may be assumed to be an instantaneous event at a fixed location in physical space. This means that a collision causes a molecule to jump from one point to another in velocity space, but it remains at the same point in physical space and time. Therefore, Figure 3-6, collisions are represented as affecting only the element $d\mathbf{c}$. A second major consequence of the dilute gas assumption is that all collisions may be assumed to be binary collisions.

First consider process (i) which is a conservative process across the surface $d\mathbf{r}$. The number of molecules in the phase space element is $n_{fd\mathbf{c}d\mathbf{r}}$, so the number density of class $e$ molecules within $d\mathbf{r}$ is $n_{fd\mathbf{c}}$. Then the net inflow of molecules of this class across the Surface of $d\mathbf{r}$ to be written as:
Equation 3-34  \[ -\int_{S_r} n f_{c} \cdot e_r dS_r d_c \]

Here, \( S_r \) is the total area of the surface of \( dr \), \( dS_r \) is an element of this surface, and \( e_r \) is the unit normal vector of this element. Gauss’ theorem enables the surface integral over \( S_r \) to be converted to a volume integral over \( dr \). The expression then becomes

Equation 3-35  \[ -\int_{d_r} \nabla (n f_{c}) d(\rho_r) d_c \]

Or, since \( cf \) and \( c \) are constants within \( dr \),

Equation 3-36  \[ -\nabla (n f_{c}) d(\rho_r) d_c \]

Note that the theorems of mathematical analysis has as hypothesis the continuity. Even though the fluid is not a continuum in a physical sense, in this formulation theorems can be applied since the distribution function is continuous.

Also, since we are considering only molecules of class \( c \), the velocity \( e \) may be taken outside the divergence in physical space. Therefore, the inflow of molecules of class \( e \) across the surface of \( dr \) due to the velocity \( e \) is

Equation 3-37  \[ -\rho \cdot \frac{\partial (n f)}{\partial t} d cd \rho \]

We may take advantage of the analogy between process (ii) in velocity space and process (i) in physical space to write the inflow of molecules across the surface of \( dr \), due to the external force per unit mass \( F \), as

Equation 3-38  \[ -F \cdot \frac{\partial (n f)}{\partial t} d cd \rho \]

The total number of molecules scattered out of the element \( dcdr \) as a result of collisions is readily obtained through an analysis similar to that which led to...inserire rif a equazioni... for the collision frequency and total number of collisions in the gas. However, in order to obtain a meaningful expression for the molecules scattered into the element, we must consider both the pre-collision and post-collision velocities of the molecules participating in the collisions. In particular, we are concerned with the collision of a molecule of class \( c \) with one of class \( c_1 \) such that their post-collision
velocities are $c$ and $c_1$, respectively. This is called a class $c$, $c_1 \rightarrow c^* c_1^*$, collisions, and we will now calculate the rate of scattering of molecules of class $c$ out of $dc_1 dr$ as a result of collisions of this class $c_1$. A molecule of class $c$ may be chosen as a test particle moving with speed $c_2$ among stationary field molecules of class $c_1$. The volume swept out in physical space by the cross-section for this class of collision is $c_2 dc_1$ and the number of class $c_1$ molecules per unit volume in physical space is $nf_1dc_1$. The number of collisions of this class suffered by the test molecule per unit time is, therefore,

\[ -nf_1 c_1 \sigma d\Omega dc_1 \]

Since the number of class $c$ molecules in the phase space element is $nf_1dc_1$, the number of class $c$, $c_1 \rightarrow c^* c_1^*$ collisions per unit time in the element is

\[ n^2 f f_1 c c_1 \sigma d\Omega dc_1 dc d\Omega \]

Just as $f$ denotes the value of the velocity distribution function $f$ at $c$, $f_1$ denotes the value of $f$ at $c_1$. Similarly, $f$ and $f_1^*$ may be used to denote the values of $f$ at $c^*$ and $c_1^*$, respectively. Note also that the expression of a binary collision probability in terms of the product of two single particle distribution functions has implicitly invoked the principle of molecular chaos.

The existence of inverse collisions means that an analysis, exactly similar to that leading to Equation 3-40 may be made for the collisions of class $c$, $c_1 \rightarrow c^* c_1^*$ that scatter molecules into class $c$. This yields

\[ n^2 f^* f_1^* c^* c_1 d\Omega \]

for the collision rate in the phase space element $dc^* dc$. Since binary elastic collision is assumed, $c_2$ is equal to $c^* r$, while the symmetry between the direct and inverse collisions is such that there is a unit Jacobian for the transformation between the pre-collision and post-collision values of the product of the differential cross-section and velocity space elements. That is,

\[ (\sigma d\Omega dc_1 dc_1) = (\sigma d\Omega)^* dc_1^* dc_1^* \]

The Equation 3-41 can be written:
Equation 3-43 \[ n^2 f^* f_1^* c_c \alpha d \Omega d c_1 d \xi d r \]

The rate of increase of molecules of class \( c \) in the phase space element \( d c \) \( d r \) as a result of the combined direct and inverse collisions of class \( c \), \( c_1 \leftrightarrow c^* c_1^* \) is obtained by subtracting the loss rate (Equation 3-40) from the rate of gain (Equation 3-43). This gives:

Equation 3-44 \[ n^2 (f^* f_1^* - f f_1) c_c \alpha d \Omega d c_1 d \xi d r \]

The total rate of increase of molecules of class \( c \) in the element as a result of collisions is given by the integration of this expression over the complete cross-section for its collision with class \( c \) molecules, followed by the integration of the class \( c_1 \) over all velocity space. The required expression for process (iii) is, therefore,

Equation 3-45 \[ \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} n^2 (f^* f_1^* - f f_1) c_c \alpha d \Omega d c_1 d \xi d r \]

The Equation 3-33 for the total rate of increase of molecules of class \( c \) due to all three processes may be equated to the aim of Equation 3-45 for process (iii) and Equation 3-37 and Equation 3-38 for processes (i) and (ii), respectively. If the latter terms are transferred to the left-hand side and the complete equation is divided by \( d c \) \( d r \), we have the Boltzmann equation for a simple dilute gas. This is

Equation 3-46 \[ \frac{\partial (n f)}{\partial t} + c \cdot \frac{\partial (n f)}{\partial r} + F \cdot \frac{\partial (n f)}{\partial c} = \int_{-\infty}^{\infty} n^2 (f^* f_1^* - f f_1) c_c \alpha d \Omega d c_1 d \xi d r \]

In a gas mixture consisting of a total of \( s \) separate species, a separate distribution function must be defined for each species. The Boltzmann equation then becomes a set of \( s \) simultaneous equations, in particular species may be represented by the subscripts \( p \) or \( q \). The Boltzmann equation for species \( p \) of the mixture can therefore be written

Equation 3-47 \[ \frac{\partial (n_p f_p)}{\partial t} + c_p \cdot \frac{\partial (n_p f_p)}{\partial r} + F_p \cdot \frac{\partial (n_p f_p)}{\partial c} = \sum_{q=1}^{s} \int_{-\infty}^{\infty} n_p n_q (f^* f_1^* - f f_1) c_q \sigma_{pq} d \Omega d c_1 d \xi d r \quad p = 1, \ldots, s \]
The presence of internal degrees of freedom requires the definition of extended distribution functions that allow for the additional dimensions of phase space. Also, the collision cross-sections of asymmetric molecules are a function of the molecular orientation and therefore change with time as the molecules rotate between collisions. Moreover inverse collisions do not exist for the classical model of polyatomic molecules. It is, however, possible to define cross-sections that are smoothed or averaged over the molecular rotations and vibrational. The Liouville theorem then leads to a unit Jacobean far the transformation corresponding the Equation 3-42 thus permitting a formulation similar to Equation 3-46. Chapman and Cowling (1970) have called the resulting equation the **generalized Boltzmann equation**.

The terms on the right-hand side of the Boltzmann equation is called the collision terms. Its integral form contrast with the partial differential form of the terms expressing space and time dependence of $n_f$, and is responsible for much of the mathematical difficulty associated with the Boltzmann equation. On the other hand, $n_f$ is the only dependent variable in the equation. This might be considered an advantage when comparing the Boltzmann equation with the Navier-Stokes equations of continuum gas dynamics, since these have the velocity components and, allowing for the equation of state, two of the thermodynamic properties as dependent variables. However, this advantage is far outweighed by the addition of the velocity space coordinates to the list of independent variables. A homogeneous gas problem becomes one-dimensional because the velocity distribution function is spherically symmetric in velocity space. This distribution becomes axially symmetric for a one-dimensional problem in physical space, so that a Boltzmann solution becomes a three-dimensional problem. For two and three dimensional flows there are no symmetries in velocity space and the problems become five or six dimensional. Time is an additional dimension if the problem is unsteady.

There is no possibility of obtaining analytical solutions of the Boltzmann equation for flows that involve complex geometries or large disturbances. Numerical solutions that require a computational mesh in phase space pose formidable problems because of the number of dimensions and the difficulties in setting the bounds of velocity space.
3.3.2. The Moment and Conservation Equations

The quantity \( Q \) related to a single molecule and is either a constant or function of the molecular velocity. We have previously seen that average value of this quantity may be obtained through the multiplication of the velocity distribution function by \( Q \), followed by the integration of product over all velocity space. These averages are referred to as moments of the distribution function. Similarly, a **moment of the Boltzmann equation** may be obtained by multiplying it by the quantity \( Q \) and then integrating the resulting equation over all velocity space. Since the moments of the distribution function include all the macroscopic properties of a monatomic gas, the moment equations may be expected to include the monatomic gas version of the conservation equations of continuum gas dynamics.

The multiplication of the Boltzmann Equation 3-46 by \( Q \) yields

\[
\text{Equation 3-48} \quad Q \frac{\partial (nf)}{\partial t} + Q \frac{\partial (nf)}{\partial r} + Q \frac{\partial (nf)}{\partial c} = Q \int \int n^2 \left( f^* f_1^* - f^* f_1^* \right) \sigma \Omega d\Omega d\epsilon_c
\]

Both \( f \) and \( Q \) refer to molecules of class \( c \) and the moment equation obtained by integrating over all classes of molecule. Since \( Q \) is either constant or a function of \( c \) only, it may be taken within the derivative the first term. The required integral of this term is, therefore,

\[
\text{Equation 3-49} \quad \int \frac{\partial}{\partial t} (n Q f) d\epsilon_c
\]

And using Equation 3-23

\[
\text{Equation 3-50} \quad \frac{\partial}{\partial t} (n Q)
\]

Both \( c \) and \( Q \) may be taken inside the derivative in the second term of Equation 3-48,

\[
\text{Equation 3-51} \quad \int \nabla \cdot (n c Q f) d\epsilon_c
\]

or

\[
\text{Equation 3-52} \quad \nabla \cdot (n c Q)
\]
through the distribution function \( f \) and must be treated as functions of \( r \) and \( t \). The integral in the third term in Equation 3-48 can be written:

**Equation 3-53**

\[
\int_{-\infty}^{\infty} F \frac{\partial}{\partial \zeta} \left( nQf \right) d\zeta - \int_{-\infty}^{\infty} F \frac{\partial Q}{\partial \zeta} (nf) d\zeta
\]

It is assumed that \( F \) is independent of \( c \) and, since \( f = 0 \) or \( f \to \infty \) as \( c \to \infty \), the first integral vanishes and the second becomes

**Equation 3-54**

\[-nF \frac{\partial Q}{\partial \zeta} \]

The integral of the term on the right-hand of Equation 3-48 is called the **collision integral** and is denoted by \( \Delta[Q] \), i.e.

**Equation 3-55**

\[
\Delta[Q] = \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} n^2 Q \left( f^+ f^+_1 - ff_1 \right) \frac{d\Omega}{4\pi} d\zeta d\zeta
\]

Summarizing the **moment equation** for \( Q \) is:

**Equation 3-56**

\[
\frac{\partial}{\partial t} \left( nQ \right) + \nabla \cdot \left( nQ \right) - nF \frac{\partial Q}{\partial \zeta} = \Delta[Q]
\]

This equation may also be called the transfer equation or the equation of change.

If the quantity \( Q \) is either the mass \( m \), momentum \( mc \), or energy \( \frac{1}{2}mc^2 \) of a molecule, the conservation of these quantities in collisions requires that \( Q + Q_1 - Q^* - Q^*_1 = 0 \). It can demonstrated that \( \Delta[Q]=0 \), as would be expected from physical meaning of the integral. The quantities \( m \), \( mc \), and \( \frac{1}{2}mc^2 \) are called **collisional invariants**, while any that satisfy the condition \( Q + Q_1 - Q^* - Q^*_1 = 0 \) are called **summational invariants**. It can be that the collisional invariants, or linear combinations of them, are the only summational invariants. Therefore, if \( Q \) is a summational invariant, the collision integral \( \Delta[Q]=0 \) and \( Q \) can be written

**Equation 3-57**

\[
Q = A \frac{1}{2} mc^2 + B \cdot mc + C
\]

where \( A \), \( B \) and \( C \) are constants.
The collision integral \( I_0 \) zero in the three equations for the collisional invariants and the average on the left-hand side of the equations can be expressed in terms of the macroscopic gas properties. The three equations are the **conservation equations of gas dynamics**.

First, the equation for the conservation of mass is obtained by setting \( Q = m \) in Equation 3-56 to give:

**Equation 3-58**

\[
\frac{\partial}{\partial t} (nm) + \nabla \cdot (nmc_0) = 0
\]

Since \( \rho = nm \) and it can be demonstrated that \( c_0 \) coincides with free stream velocity \( c_0 \):

**Equation 3-59**

\[
\frac{\partial}{\partial t} \rho + \nabla \cdot (\rho c_0) = 0
\]

Introducing the substantial derivative:

**Equation 3-60**

\[
\frac{D\rho}{Dt} = -\rho \nabla \cdot c_0
\]

By means of similar considerations the conservation of momentum (or equation of motion) can be written:

**Equation 3-61**

\[
\rho \frac{Dc_0}{Dt} = -\nabla \cdot p + \nabla \cdot \tau + \rho F
\]

And the conservation of energy:

**Equation 3-62**

\[
\rho \frac{De}{Dt} = -p \nabla \cdot c_0 + \psi - \nabla \cdot q
\]

Since the momentum Equation 3-61 is a vector equation, the equations for the conservation of mass, momentum, and energy constitute five equations. The dependent variables include the three velocity components and, if the equation of state is taken into account, two of the three thermodynamic properties \( p, p, \) and \( T \). The viscous stress tensor \( \tau \) contains nine components but, because of its symmetry and the relationship of the sum of the diagonal components to the pressure, it contributes only five dependent variables. Finally, the heat flux vector \( q \) contributes a further three dependent variables. The conservation equations therefore contain a total of thirteen dependent variables and
do not form a determinate set. The inviscid case with $\tau$ and $q$ both equal to zero does result in a determinate set, and these are called the **Euler equations**.

It can be demonstrated that when the distribution function $f$ is equal to the Maxwellian $f_0$:

\[
f_0 = \left(\frac{\beta^3}{\pi^{3/2}}\right) \exp\left(-\beta^2 c^2\right)
\]

Boltzmann equations coincides with Navier-Stokes equations.

### 3.3.3. Accommodation Coefficients

The majority of problems involve the interaction of gas molecules with solid surface. The gas generally has a stream velocity relative to the surface, and this means that the stagnation temperature of the gas differs from its static temperature. In addition, the surface temperature must differ from at least one of these, and the distribution function of the incident molecules will be different from that for the reflected molecules. Therefore, the distribution function of the gas near the surface will not be an equilibrium distribution. Also, the energy of a molecule relative to the surface before it strikes the surface will generally be different from the corresponding energy after it has been reflected from the surface, so that the process is inelastic. The current models for gas-surface interactions are largely phenomenological in nature and their adequacy varies with the nature of the surface and the magnitude of the impact energy of the molecule relative to the surface. The most widely used models are based on generalization of the diffuse and specular models.

The most common generalization of the diffuse model is to allow the incident and reflected molecules to have different temperature. For example, it may be assumed that the molecules that are incident on the surface are characteristic of those in an equilibrium gas at temperature $T_i$, and that those reflected from the surface are characteristic of those in an equilibrium gas at temperature $T_r$. Note that the temperature $T_r$ may differ from the temperature $T_w$, of the surface. An indication of the extent to which the reflected molecules have their temperature adjusted toward that of the surface is given by the thermal accommodation coefficient

\[
a_c = \frac{(q_i - q_r)}{(q_i - q_w)}
\]

Here, $q_i$ and $q_r$ are respectively the incident and reflected energy fluxes, while $q_w$ is the energy flux that would be carried away in diffuse reflection with $T_r = T_w$. The range
of $a_c$ is from zero for no accommodation to unity for complete thermal accommodation. Accommodation coefficients may also be defined for the normal and tangential components of momentum, mass fractions and the other variables. However, the accommodation coefficients may be written as functions of the macroscopic pressure, shear stress, and heat flux and, in general, it is preferable to describe the interaction directly in terms of these quantities.

Experiments with ‘engineering’ surface in contact with gases at normal temperatures indicate that the reflection process approximates diffuse reflection with complete thermal accommodation. This behaviour may be a consequence of such surfaces being microscopically rough with the incident molecules suffering multiple scattering, or of the molecules being momentarily trapped or adsorbed on the surface. Most analytical and numerical studies are based on the assumption of diffuse reflection, and it is fortunate that this appears to be adequate for the vast majority of practical gas flows. The diffuse assumption is supposed in the present research activity.
Chapter 4 Numerical Solutions

4.1. Numerical Solution of Transitional Regime - Direct Simulation Monte Carlo (DSMC)

Direct Simulation Monte Carlo (DSMC) method has been developed by Bird (reference 10) since 1963. Nowadays, this method is the only able to solve in depth transitional regime.

Differently from the “classic” fluidodynamics, DSMC doesn’t consider the fluid as a continuum but made of discrete molecules; each molecule is provided of translational, rotational and vibrational energy variable with time as result of collisions between molecule/molecule or molecule/wall.

Molecular movement and collisions are based on Kinetic theory.

The gas is considered composed by N molecules each representative of $F_N$ real molecules. Usually, the $F_N$ is equal to $10^{17}$ according to statistical considerations (see reference 10). Each simulating molecule is followed directly by means of a procedure that will be discussed later, so the words Direct Simulation are used.

The sampling N molecules and their evolution in space and time is obtained by means of sampling and evolution based on random numbers generation, so the words Monte Carlo are used.

The DSMC procedure for the simulation of a representative set of intermolecular collisions are best derived and demonstrated in the context of a homogeneous or ‘zero-dimensional’ gas. The homogeneous gas is divided into a one-dimensional structure of cells and sub-cells. The procedures for the establishment of the correct collisions rate were based on the cells, while individual collision pair were chosen from the sub-cells.

First of all, the control volume is divided in cell, at each time step microscopic properties are calculated. The macroscopic quantities, such as pressure temperature and so on, are calculated as an average of the microscopic ones.

Each cell is divided in subcelles, in which the collisions are computed according to the “nearest neighbour” logic.
The cell are automatically computed according to molecular motion, while the subcelles according to intermolecular collisions. This represents an essential approximation in DSMC procedure: uncoupling between molecular motion and intermolecular collisions.

Of course, the cell size must be less than the mean free path, because the choice of collisional partners inside has to be physically correct. The cell are computed automatically with these guidelines.

DSMC evolves his procedure in time; time step must be less than $1/\nu$, where $\nu$ is the mean collisional frequency.

The ratio of the local mean separation between collision partners to the local mean free path (mcs/l) should be less than unity every domain. Bird (references 11 and 12) suggests the value of 0.2 as a limit value, so an adequate number of simulated molecules and cells is required.

The next diagram shows briefly DSMC procedure.
More in detail, the first step, as already told before, is to fix cells and subcelles.

At the beginning the molecules are distributed uniformly in the calculation region with an initial state (velocity, thermodynamic parameters as inputs). Another choice is to set the initial state to vacuum and a mass flow in the inlet. The applications treated in the present work relies on first choice (first step in the flow chart in Figure 4-1).

Then each molecule \( m \) is positioned in a cell a homogenously according to the next formula:
Equation 4-1  \[ x_m^a = x_b^a + R\Delta x \]

Where, \( x_b^a \) is the beginning position inside the cell a, R is a random number included in the range \([0,\ldots,1]\) and \( x_m^a \) the new position (the step “Molecules distribution inside the cells” in the flow chart in Figure 4-1).

The initial velocity of each molecule is equal to the free stream one \( V_\infty \), then the new velocity is computed taking into account the thermal speed \( c \) according to a maxwellian distribution (\( c = \sqrt{\frac{2k}{m}T} \)).

The new velocity is obtained as follow:

Equation 4-2  \[ U = c\sqrt{-\ln(R)} + \text{sen}(2\pi R) \]

The molecules are moved simultaneously to the new position \( x_m \).

Equation 4-3  \[ x_m = x_m^c + U\Delta t \]

After the movement some molecules should have changed their cell, so after the ”move step” and before the “collision step” the molecules will be “addressed” to eventually new cell (the step “Index of molecules based on the new position” in the flow chart in Figure 4-1); for example, in the one-dimensional case:

Equation 4-4  \[ \text{New - cell} = \frac{x_m}{\Delta x} \]

At the end of this process, the number of molecules in each cell is known. Moreover for each cell the subcell in which each molecule is located and the value of velocity and position is identified.

The next step is to evaluate for each cell the collisional partners (the step “Collisions” in the flow chart in Figure 4-1).

The probability of a collision between two molecules in a homogeneous gas is proportional to the product of their relative speed \( c_r \) and total collision cross-section \( \sigma_T \).

It can be demonstrated (see reference 10) that the total number of collisions per unit time per unit volume of gas is given by.
Equation 4-5 \[ N_c = \frac{1}{2} n^2 \sigma_T c_r \]

This equation for the non-equilibrium collision rate in a homogeneous gas could be used to establish the number \( N_c \Delta t \) of collisions in each cell at each time step of duration \( \Delta t \), and this number could then be computed. The mean value of the product of \( c_r \) and \( \sigma_T \) is calculated for each cell, and the maximum value could also be recorded. The collision pairs could then be chosen by the acceptance-rejection method, with the probability of a particular pair being given by the ratio of their product of \( c_r \) and \( \sigma_T \) to the maximum product. However, this procedure would have a computation time proportional to the square of the total number of molecules in the cell.

Consider a DSMC cell of volume \( V \) in which each simulated molecule represents \( F_N \) real molecules. The probability \( P \) of collision between two simulated molecules over the time interval \( \Delta t \) is equal to the ratio of the volume swept out by their total cross-section moving at the relative speed between them to the volume of the cell, i.e.

Equation 4-6 \[ P = \frac{F_N \sigma_T c_r \Delta t}{V_c} \]

The relative speed varies with the choice of the collision pair and the total cross-section is generally a function of the relative speed, but the other quantities in Equation 4-5 are independent of this choice. The average number of real molecules in the cell \( nV_C \) and the average number of simulated molecule is \( N = nV_C/F_N \), where \( n \) is the number density in the real gas. The full set of collisions could be calculated by selecting, in turn, all \( N(N-1)/2 \) pair in the cell and by computing the collisions with probability \( P \). This method has been used in DSMC simulations, but is inefficient because \( P \) is generally a very small quantity and the number of choice is nearly proportional to the square of the number of molecules. Also, since \( F_N \) is an extremely large number, the number of choices should be \( N^2/2 \), and this error becomes significant for small values of \( N \). The procedure can be made more efficient and the second difficulty removed if only a fraction of the pair are included and the resultant probability increased by dividing Equation 4-5 by this fraction. Maximum efficiency is achieved if the fraction is such that the maximum probability becomes unity. The fraction is then given by:
and the number of pair selections per time step is obtained by multiplying this equation by $N^2/2$. However, in most cases, $N$ is a fluctuating quantity and, because the mean of the square differs from the square of the mean, $N^2$ should be replaced by the product of the instantaneous value and a time or ensemble averaged value. Therefore, the NTC method la that

$$P_{\text{max}} = \frac{F_N(\sigma_Tc_\ell)_{\text{max}} \Delta t}{V_c}$$

pairs are selected from the cell at the time step, and the collision la computed with probability

$$p = \frac{\sigma_Tc_\ell}{(\sigma_Tc_\ell)_{\text{max}}}$$

In particular, once fixed the cell, the first molecule “i” is randomly selected from the first subcell as well as the second particle “j”. If $j$ coincides with $i$ the selection will be repeated and, if the subcell contains only the particle $i$, $j$ will be selected from the nearest cell. This process follows the so called “nearest-neighbour” logic.

Once obtained the candidate collisional partners “i-j”, Equation 4-9 is evaluated and if $P$ is larger than $R$ (random number included in [0,…,1]) the collision occurs. This process continues till the completion of all the candidate collisional partners (Equation 4-8).

If the collision occurs, the post-collisional velocity of the particles $i$ and $j$ is evaluated by means of the binary collision dynamics (see reference 10). Different collision dynamics models can be used, for example the classic binary elastic collision model. The results that will discussed in the following chapters relies on the Variable Hard Sphere model (VHS).

It can be demonstrated that the post-collisional velocities in the three-dimensional case are:

$$U = c_r \cos \chi \quad V = c_r \sin \chi \cos \varepsilon \quad W = c_r \sin \chi \sin \varepsilon$$

$$\cos \chi = 2R - 1 \quad \varepsilon = 2\pi R$$
Known the post-collision velocity, the thermodynamic variables in the cell can be calculated as average on all the particles inside the cell by means of Kinetic theory (the step “Extrapolation for each cell of macroscopic variables from the microscopic ones” in the flow chart in Figure 4-1).

This process will continue till the stationary (the step “Stationary?” in the flow chart in Figure 4-1).

Of course, the described process can be generalized to a non-equilibrium chemical reacting gas.

The DSMC software used in this work is the DS2V and DS3V codes of Bird (references 11 and 12).

\[\text{4.2. Solution of Free Molecular Flow}\]

As already discussed, free-molecule or collisionless flows are the limiting case in which Knudsen number tends to infinity.

The interactions between particles can be neglected and the only interactions between particle wall can be taken into account.

In this case the solution of Boltzmann equation, in the hypothesis of maxwellian distribution of the distribution function \(f\), can be analytically obtained, and the number flux \(n_r\) is obtained (for details see reference 10):

\[\text{Equation 4-11} \quad n_r = n_w \left(\frac{T_w}{T_r}\right)^{\frac{1}{2}} \left[\exp\left(-s^2 \sin^2 \alpha\right) + \frac{1}{\sqrt{2}} s \sin \alpha \{1 + \text{erf}(s \sin \alpha)\}\right]\]

Where \(s = V_w / \sqrt{2RT}\) is the speed ratio and \(\alpha\) the local angle of incidence between flow and the wall.

If a fraction \(\varepsilon\) of the molecules is reflected specularly and the remaining fraction \(1-\varepsilon\) diffusely the pressure is:
Equation 4-12
\[
p / p_w = \left[ (1 + \varepsilon) \pi r^2 s \sin \alpha + 1/2 (1 - \varepsilon) (T_r / T_w)^{1/2} \right] \times \exp \left( -s^2 \sin^2 \alpha \right) + \\
+ \left[ (1 + \varepsilon) (1/2 + s^2 \sin^2 \alpha) + (1 - \varepsilon) (T_r / T_w)^{1/2} \pi r^2 s \sin \alpha \right] \left[ 1 + \text{erf} \left( s \sin \alpha \right) \right]
\]

The shear stress is:

Equation 4-13
\[
\tau / p_w = \pi r^2 s \cos \alpha \left[ \exp \left( -s^2 \sin^2 \alpha \right) + \pi r^2 s \sin \alpha \left[ 1 + \text{erf} \left( s \sin \alpha \right) \right] \right]
\]

The heat transfer to the surface:

Equation 4-14
\[
2 \beta^3 q / \rho_w = (1 - \varepsilon) \left( 2\pi r^2 \right) \left[ (s^2 + \gamma/(\gamma - 1) - \{1/2 (\gamma + 1)/(\gamma - 1)\}(T_r / T_w)) \times \right. \\
\left. \left[ \exp \left( -s^2 \sin^2 \alpha \right) + \pi r^2 s \sin \alpha \left[ 1 + \text{erf} \left( s \sin \alpha \right) \right] \right] - 1/2 \exp \left( -s^2 \sin^2 \alpha \right) \right]
\]

Where \( \beta \) is the reciprocal of the most probable molecular speed in an equilibrium gas \( \beta = (2RT)^{-1/2} \), and the subscript \( r \) indicates the reflecting properties that, in the case of fully accommodated conditions is coincident with the wall conditions.

For specular reflection and stationary gas:

\[
\begin{align*}
p &= 2p_i \\
\tau &= 0 \\
q &= 0
\end{align*}
\]

For diffuse reflection:

\[
\begin{align*}
p &= \frac{n_r m}{4 \beta_r^2} \\
\tau &= \tau_i \\
T &= T_r = T_w \\
q &= \left( \frac{\gamma + 1}{\gamma - 1} \right) \frac{n_r m}{8 \pi r^2 \beta_r^2}
\end{align*}
\]

The subscript \( i \) means “incident” i.e. the variable value multiplied for the sine of the local angle of incidence.
4.3. **Solution of Continuum Regime – Computational Fluid Dynamics (CFD)**

The continuum regime results have been obtained by using the CIRA Computational Fluid Dynamics (CFD) code H3NS developed at the Aerothermodynamics and Space Propulsion Laboratory (reference 13). The code solves full Reynolds Averaged Navier-Stokes (RANS) equations and considers the air flow in thermo-chemical non-equilibrium. Park model with five species (O, N, NO, O$_2$, N$_2$) and 17 chemical reactions (reference 14) is implemented and the energy exchange between vibrational and translational temperature is based on Landau-Teller non-equilibrium equation, with average relaxation times taken from the Millikan-White (reference 15) theory modified by Park (reference 16). The viscosity coefficients for the single species are computed by means of Yun and Mason collision integrals (reference 17), while the conductivity coefficient using Eucken’s law. These coefficients for the gas mixture are calculated using semi-empirical Wilke formulas. Diffusion coefficients are obtained by Yun and Mason tabulated collision integrals (reference 17).

From the numerical point of view, the code is based on a finite volume approach with a cell centered formulation. The inviscid fluxes are computed by Flux Difference Splitting scheme (reference 18), second order approximation is obtained with an Essentially Non Oscillatory (ENO) reconstruction of interface values. Time evolution is performed by an explicit multistage Runge-Kutta algorithm coupled with an implicit evaluation of the source terms.

To take into account the effects of rarefaction, **slip boundary conditions** have been employed. From the large number of available formulations of this kind of conditions, the one proposed by Kogan (reference 19) has been chosen. These boundary conditions have been obtained by matching the solution of Boltzmann equation in the Knudsen layer to the solution of the macroscopic Navier-Stokes equations, thus yielding:

**Equation 4-17** \[ V_s = 1.012 \left( \frac{\partial V_x}{\partial n} \right)_w \]

**Equation 4-18** \[ T_s - T_w = 1.73 \frac{\gamma \sqrt{\pi}}{\gamma - 1} \frac{1}{4} \lambda \left( \frac{\partial T}{\partial n} \right)_w \]
Chapter 5 Effects of Rarefaction on a Winged Hypersonic Re-entry Vehicle

As already discussed, an important aspect to be taken into account in the design of a re-entry vehicle is the evaluation of the effects of rarefaction on the aerodynamic coefficients and the heat flux. The evaluation of the lift and of the drag is important if a high aerodynamic efficiency re-entry is required (one of the main goal of FTB-X trajectory). At high altitudes (or in rarefied regimes) a decrease of the lift and an increase of the drag occur. Also the moments can be different from those in continuum regime, implying different stability behaviour. The availability of reliable aerodynamic coefficients during the re-entry at high altitudes is important also for defining the trajectory and for sizing the Reaction Control System.

To this aim DSMC simulations are necessary, but they are very time consuming. Therefore, some bridging formulae proposed by Potter (references 21, 22 and 23) are tested on a preliminary FTB-X configuration (1.1.2), in order to assess the methodology that can be applied both to the current baseline and to a future FTB-X configuration (as per 3.9.2-FW50), allowing a prompt evaluation of the aerodynamic database up to high altitudes. Than a goal is of the present chapter is to verify the applicability of engineering methods such as bridging formulae and panel methods in a preliminary design of FTB-X. since in this phase of the project, several configurations must be rapidly analyzed in order to make a trade-off study and to define the most promising geometry to be chosen for the following phases of the project.

Moreover, the sizing of the Thermal Protection System has been investigated. In particular, the thermal load at the stagnation point in axial-symmetric has been calculated with different catalytic assumption and methodologies in the higher part of the FTB-X re-entry trajectory.

The next paragraph 5.1 will illustrate the obtained results on the global FTB-X aerodynamic and the paragraph 5.2 the nose thermal analysis.
5.1. **FTB-X High Altitude Aerodynamic**

In this paragraph the analysis of rarefaction effects on the global aerodynamic of FTB-X vehicle is presented. As already told, Potter’s bridging formulae have been used, so a brief description is briefly conducted in the next paragraph 0.

### 5.1.1. Bridging Formulae

Potter described the correlation of normalized aerodynamic coefficients with a simulation parameter which is designed to account for the principal flow phenomena that cause the coefficients to vary.

These correlation were necessary to extrapolate and scale wind tunnel data, enable to duplicate all the full-scale Space Shuttle’s flight environment.

Since FTB-X is “similar” to Space Shuttle, as it is a winged body, these scaling factors will be applied.

Potter used the high number of experimental results and flight data to build his formulae that “bridge” free molecular regime to the well known inviscid hypersonic limit, i.e. from Re from zero \((K\xi \to \infty)\) to infinity. From a physical point of view, if we follow a typically re-entry trajectory free molecular flow regime takes place, of course, at very high altitude (about 150 km), while the hypersonic inviscid limit takes place in the *continuum* regime when the Mach number reaches the maximum value.

Potter defined the Drag and Lift normalized as follow:

**Equation 5-1**  
\[
\overline{C_D} = \left( C_D - C_{D_{Df}} \right) / \left( C_{D_{f}} - C_{D_{f}} \right)
\]

**Equation 5-2**  
\[
\overline{C_L} = \left( C_L - C_{L_{Df}} \right) / \left( C_{L_{f}} - C_{L_{f}} \right)
\]

The coefficients are correlated with simulations parameters:

**Equation 5-3**  
\[
P_{ND} = \left\lfloor \frac{V_s}{V_{\infty}} \left( \frac{PFA/WA}{V_{\infty}} \right)^{1/2} \left( \frac{H_{\infty}}{(0.2H_{\theta} + 0.5H_{w})} \right)^{0.63} \right\rfloor^{1/2}
\]

**Equation 5-4**  
\[
P_{NL} = P_{ND} \left( \frac{PPA/PFA}{PFA} \right)^{1/4}
\]
The classic behaviour of the coefficients in the Equation 5-1 and Equation 5-2 with respect to the parameters $P_{ND}$ and $P_{NL}$ is showed in the next figure:

![Figure 5-1](image)

**Figure 5-1** Potter Bridging Formula.

Figure 5-1 shows the typical “S” behaviour of aerodynamic coefficients in the transitional regime: as the altitude raises the Drag coefficient grows, while the lift decreases and, as a consequence, the aerodynamic efficiency decreases.

The loss of aerodynamic performance at high altitudes means that the aerodynamic shape of the spacecraft does not work and the spacecraft behaves as a flat plate. It is worth noting that high values of $P_N$ corresponds to the higher altitude conditions, while the lower $P_N$ values corresponds to the lower ones.

### 5.1.2. Test Conditions

The Figure 5-2 shows that the re-entry trajectory of FTB-X, in the altitude interval from 90 to 150 km, is in transitional regime, for higher altitude the flow regime can be considered free molecular, while for lower continuum conditions occur.
Three dimensional *continuum* regime simulations for the global aerodynamic evaluation have been obtained by means of CIRA CFD code H3NS. These simulations are very CPU consuming, so some Panel Method (PM) calculations have been performed.

The Panel Method (PM) solution is based on several well known methodologies such as Newtonian, Modified Newtonian, Prandtl-Meyer, Tangent Wedge Empirical and Tangent Cone applied to different parts of the vehicle (nose, fuselage, wing leading edge, upper wing and so on) with viscous corrections based on boundary layer models and Eckert method.

The comparison of the results obtained by PM with those by CFD indicated the best combination of the engineering methods to be used in evaluating aerodynamic coefficients of the vehicle. An example of such comparisons is reported in Figure 5-3 where the good match of the aerodynamic coefficients from CFD and PM is shown. The runs were made at conditions very close to the present ones: M=20, Re_LX=5×10^5, even though on a configuration (3.9.2-FW50) of FTB-X slightly different from the present one.
Figure 5-3 Comparison of drag (a) and lift (b) coefficients of FTB-X from CFD and Panel Method.

The present work relies on the aerodynamic analysis of the whole vehicle in the altitude interval 90-200 km and a free stream velocity of 7330 m/s, constant with altitude. The wall temperature was 300 K and free stream thermodynamic parameters were provided by the U.S. Standard Atmosphere 1976.

The aerodynamic forces were evaluated on the assumption of not-reactive surface. The surface was considered fully catalytic level. As the working gas is made up of 5 chemical species (O2, N2, O, N and NO), the recombination reactions, implemented in the present runs were: O+O->O2, N+N->N2 and N+O->NO.

Table 5-1 reports the DSMC test conditions and some aerodynamic parameters for the whole vehicle. The tests in side-slip flight were made at $\alpha$=30 deg.. Table 5-2 reports the conditions for CFD and PM; these runs were made in symmetric flight.

<table>
<thead>
<tr>
<th>h [km]</th>
<th>$M_\infty$</th>
<th>$Re_{st,\infty}$</th>
<th>$Kn_{4,\infty}$</th>
<th>$Kn_{5,\infty}$</th>
<th>$Kn_{4,\infty}$</th>
<th>$Kn_{5,\infty}$</th>
<th>$\alpha$ [deg.]</th>
<th>$\beta$ [deg.]</th>
</tr>
</thead>
<tbody>
<tr>
<td>90</td>
<td>26.6</td>
<td>1.1x10^4</td>
<td>3.4x10^7</td>
<td>9.5x10^7</td>
<td>1.3x10^7</td>
<td>0,10,20,30,40</td>
<td>0,5,10,15</td>
<td></td>
</tr>
<tr>
<td>95</td>
<td>26.4</td>
<td>4.5x10^3</td>
<td>8.4x10^4</td>
<td>2.4x10^4</td>
<td>3.1x10^4</td>
<td>0,10,20,30,40</td>
<td>0,5,10,15</td>
<td></td>
</tr>
<tr>
<td>100</td>
<td>25.7</td>
<td>1.8x10^3</td>
<td>2.1x10^4</td>
<td>5.9x10^4</td>
<td>7.7x10^4</td>
<td>0,10,20,30,40</td>
<td>0,5,10,15</td>
<td></td>
</tr>
<tr>
<td>105</td>
<td>24.6</td>
<td>6.8x10^2</td>
<td>5.3x10^3</td>
<td>1.5x10^3</td>
<td>1.9x10^3</td>
<td>30</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>110</td>
<td>22.5</td>
<td>2.5x10^2</td>
<td>1.3x10^3</td>
<td>3.6x10^3</td>
<td>4.8x10^3</td>
<td>0,10,20,30,40</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>115</td>
<td>20.1</td>
<td>94</td>
<td>3.1x10^3</td>
<td>8.5x10^3</td>
<td>1.1</td>
<td>30</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>120</td>
<td>18.1</td>
<td>42</td>
<td>6.2x10^3</td>
<td>1.7</td>
<td>2.3</td>
<td>30</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>130</td>
<td>15.6</td>
<td>12</td>
<td>1.8</td>
<td>5.1</td>
<td>6.8</td>
<td>30</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>140</td>
<td>14.0</td>
<td>5</td>
<td>4.2</td>
<td>11.6</td>
<td>15.3</td>
<td>30</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>150</td>
<td>13.0</td>
<td>2</td>
<td>8.0</td>
<td>22.3</td>
<td>29.5</td>
<td>30</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>170</td>
<td>11.6</td>
<td>1</td>
<td>22.7</td>
<td>63.1</td>
<td>83.5</td>
<td>30</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>200</td>
<td>10.3</td>
<td>0.2</td>
<td>74.1</td>
<td>206.0</td>
<td>272.5</td>
<td>30</td>
<td>0</td>
<td></td>
</tr>
</tbody>
</table>

Table 5-1 DSMC test conditions for FTB-X.
Table 5-2  CFD and PM test conditions for FTB-X.

<table>
<thead>
<tr>
<th>Method</th>
<th>α [deg.]</th>
<th>Re_in,∞</th>
<th>M_∞</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>0,10,20,30,40</td>
<td>Inviscid</td>
<td>26.36</td>
</tr>
<tr>
<td>PM</td>
<td>0,10,20,30,40</td>
<td>Inviscid</td>
<td>26.36</td>
</tr>
<tr>
<td>PM</td>
<td>0,2,5,7,10,12,15,17,20,22,25,27,30</td>
<td>2×10^4</td>
<td>26.36</td>
</tr>
<tr>
<td>PM</td>
<td>0,2,5,7,10,12,15,17,20,22,25,27,30</td>
<td>2×10^4</td>
<td>26.36</td>
</tr>
<tr>
<td>PM</td>
<td>0,10,20,30,40</td>
<td>1×10^4</td>
<td>26.36</td>
</tr>
<tr>
<td>PM</td>
<td>30</td>
<td>2×10^6</td>
<td>25,20,16</td>
</tr>
<tr>
<td>PM</td>
<td>30</td>
<td>2×10^4</td>
<td>25,20,16</td>
</tr>
<tr>
<td>PM</td>
<td>30</td>
<td>1×10^4</td>
<td>25,20,16</td>
</tr>
</tbody>
</table>

For what concerns DSMC computations, the computational domain for DS3V was a parallelepiped: X=7.5 m, Y=3.2 m, Z=2.8 m. The parallelepiped was divided into 69×13×25 cells, each one into 7×7×7 elements; the dimension of the elements in the three direction was about 1.6×10^-2 m. This dimension satisfies the requirement that it has to be smaller than the free molecular path. The simulation times were about 10 times longer than the related reference times. The reference time, considered as the time needed to cross the computational domains at the free stream velocity (L_{ref}/V_{∞}), are about 10^-3 s. The number of simulated molecules was between 1.5×10^7. For the tests, elated to the whole vehicle in the altitude intervals 90-110 km, msc/λ ranged from 0.25 to 7×10^-2.

For what concerns the CFD simulations the grid independence has been checked.

5.1.3. DSMC Results for the Whole Vehicle.

The high altitude, aerodynamic behaviour of FTB-X (1.1.2) in symmetric and side-slip flights in transitional regime (see also 20) is shown in Figure 5-4 a-d and Figure 5-5 a-c, respectively. The well known effects of rarefaction are clearly shown in Fig.4a and 4b; CL decreases and CD increases with altitude. In the altitude interval 90-110 km, the lift curve slope (dCL/dα) decreases from 1.40 to 0.88 rad^-1 (the percentage decrease is 37%) and CD increases, for instance at α=10 deg. from 0.37 to 0.86 (the percentage increase is 136%), at α=40 deg. from 1.70 to 2.55 (the percentage increase is 52%). At each altitude, the maximum value of the aerodynamic efficiency is met at α=30 deg. (Figure 5-4 a); it decreases from 0.66 to 0.25.
Figure 5-4 Aerodynamic coefficients of FTB-X (1.1.2) in symmetric flight: $\beta=0$.

As expected the effects of side-slip are detrimental for the aerodynamic qualities of FTB-X; the lift reduces, the drag increases and the pitching moment increases. For instance, at $h=100$ km, $C_L$, $C_D$, $E$ and $C_M$ at $\beta=0$ deg. are 0.64, 1.40, 0.46 and $-1.85$, at $\beta=15$ deg. are 0.61, 1.43, 0.43 and $-1.82$, respectively; the percentage variations with respect to the symmetrical flight are about $-5\%$, $2\%$, $-7\%$ and $2\%$. $dC_L/d\beta$, $dC_Mx/d\beta$ and $dCMz/d\beta$ increase respectively from $1.06$, $-9.0\times10^{-2}$, $0.293$ rad$^{-1}$ to $1.92$, $-6.24\times10^{-2}$, $0.24$ rad$^{-1}$ in the altitude interval 90-110 km.
Figure 5-5 Aerodynamic coefficients of FTB-X (1.1.2) as a function of the side-slip angle: $\alpha=30$ deg..

5.1.4. Comparison of the Results from DSMC, CFD and Engineering Methods

Figure 5-6 shows the global aerodynamic efficiency obtained with different methodologies (i.e. CFD, PM and DSMC computations) with respect the a angle of attack and different unit Reynolds number (Rem). In particular a parametric study has been performed to cover from low density (i.e. Rem=800) to inviscid conditions.

As already discussed in the pervious paragraph 5.1.2, PM calculations are in good agreement with CFD inviscid runs.

Note that for an equal Rem number (for example 2000) continuum overestimates DSMC ones of about 43% and the angle of attack where the maximum efficiency occurs is about 20 deg. for DSMC results, while the predicted by PM methods is about 30 deg. This is a very important result, since the good prediction of maximum E and the related angle of attack is crucial for guidance system.
Moreover, as the Reynolds number raises the efficiency grows and the related angle of attack decreases, as good as the flow field is closer to the *continuum* conditions.

![Figure 5-6](image)

**Figure 5-6** Global aerodynamic efficiency vs. AoA - comparison between methodologies.

Figure 5-7 reports the efficiency $E$ computed by DSMC, PM and the Potter bridging formulae, as functions of the $P_{ND}$ parameter for the angle of attack in which maximum efficiency in rarefied conditions occurs, i.e. $\alpha=30$ deg. The inviscid limit, as computed by CFD is also shown and a free molecular calculation has been performed, since in order to apply Potter’s formulae (see Equation 5-3 and Equation 5-4) inviscid limit and the free molecular one must be known.

The good match of the results from DSMC and the bridging formulae verifies the capability of using these formulae also for FTB-X even though, as said before, the formulae were obtained using the Space Shuttle data. It is worth noting that in this work DSMC data are the best data to verify the good prediction of Potter’s formula of FTB-X since flight information are, of course, not yet disposable.

Figure 5-7 shows that, for high values of $P_{ND}$ (i.e. at low altitude), the continuum results from PM are in good agreement with those from the bridging formulae. However, when altitude increases and $P_{ND}$ decreases, the PM results tend to significantly overestimate the efficiency. For example, when $P_{ND} = 7.8$ (corresponding to $h=90$ km) the value of $E$ from PM is nearly 0.91, while the one from DSMC is about 0.58. In this condition, PM overestimates $E$ of about 60%. It can be concluded that, for $P_{ND}$ approximately lower than 40 (i.e. $h>60$ km), it is important to take into account for
the FTB-X trajectory the rarefaction effects; this can be done by using a DSMC code or alternatively the bridging formulae, that allow having a more rapid and sufficiently reliable estimation of the aerodynamic coefficients; the results can be successively verified by means of DSMC computations only in selected conditions.

![Aerodynamic efficiency of FTB-X](image)

**Figure 5-7** Aerodynamic efficiency of FTB-X (1.1.2) as a function of PND: $\alpha=30$ deg..

Finally, in order to characterize the stability behaviour of FTB-X at high altitude, the moment coefficients computed by means of CFD code and PM is shown in Figure 5-8 as a function of angle of attack for an altitude of 90 km. Some discrepancies appear between the two methodologies and, it can be concluded that also for the prediction of the pitching moment $C_{my}$ for an altitude higher than 60 km, rarefaction effects has to be taken into account and continuum results are not applicable.
Figure 5-8 FTB-X (1.1.2) pitching moment coefficient versus the angle of attack: $\beta=0$ deg., $h=90$ km.

5.2. **Nose Thermal Analysis**

5.2.1. Test Conditions

Table 5-3 reports the test conditions for 2-D axis-symmetric simulation on FTB-X nose for the thermal analysis.

<table>
<thead>
<tr>
<th>$h$ [km]</th>
<th>$V_\infty$ [m/s]</th>
<th>$M_\infty$</th>
<th>$Re_D^*$</th>
<th>$Kn_{Dx}$</th>
<th>$Re_D^2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>110</td>
<td>7342</td>
<td>22.5</td>
<td>8</td>
<td>3.8</td>
<td>0.45</td>
</tr>
<tr>
<td>100</td>
<td>7353</td>
<td>25.8</td>
<td>62</td>
<td>6x10^{-1}</td>
<td>2.7</td>
</tr>
<tr>
<td>95</td>
<td>7357</td>
<td>26.5</td>
<td>159</td>
<td>2.4x10^{-1}</td>
<td>6.7</td>
</tr>
<tr>
<td>85</td>
<td>7346</td>
<td>26.6</td>
<td>917</td>
<td>4.2x10^{-2}</td>
<td>38.7</td>
</tr>
<tr>
<td>80</td>
<td>7296</td>
<td>25.7</td>
<td>1987</td>
<td>1.9x10^{-2}</td>
<td>88.2</td>
</tr>
<tr>
<td>77</td>
<td>6945</td>
<td>24.1</td>
<td>2916</td>
<td>1.2x10^{-2}</td>
<td>139.2</td>
</tr>
<tr>
<td>75</td>
<td>6589</td>
<td>22.7</td>
<td>3662</td>
<td>8.9x10^{-2}</td>
<td>187.0</td>
</tr>
<tr>
<td>70</td>
<td>5890</td>
<td>19.8</td>
<td>6492</td>
<td>4.3x10^{-3}</td>
<td>386.2</td>
</tr>
<tr>
<td>68</td>
<td>5455</td>
<td>18.0</td>
<td>8246</td>
<td>3.1x10^{-3}</td>
<td>542.5</td>
</tr>
<tr>
<td>64</td>
<td>5087</td>
<td>16.4</td>
<td>12288</td>
<td>1.9x10^{-4}</td>
<td>897.4</td>
</tr>
<tr>
<td>60</td>
<td>4624</td>
<td>14.6</td>
<td>17019</td>
<td>1.22x10^{-5}</td>
<td>1412.5</td>
</tr>
</tbody>
</table>

Table 5-3 DSMC and CFD test conditions for the nose of FTB-X
DSMC and CFD computations in real gas conditions have been performed in order to investigate the influence of methodology on prediction of thermal load at the stagnation point.

Heat flux was evaluated at the stagnation point and, in order to check the incidence of catalyticity. The surface was considered not-reactive and reactive with a fully catalytic level. As the working gas is made up of 5 chemical species (O2, N2, O, N and NO), the recombination reactions, implemented in the present runs were: O+O->O$_2$, N+N->N$_2$ and N+O->NO.

For the runs of DS2V, the computational domain was a rectangle in the meridian plane of the sphere-cone body: X=0.6 m and Z=0.3 m. The code is able to set automatically the numbers of cells and elements, according to the input number of megabytes to be used by the calculation, for the present runs 1350 Mb. Also for 2-D test the simulation time were set about 10 times longer than the related reference time. The number of simulated molecules was $2 \times 10^7$ and $\text{msc}/\lambda$ ranged from 0.85 to $10^{-3}$.

For what concerns the CFD simulations the grid independence has been checked.

5.2.2. Results

Figure 5-9 shows the profiles of the heat flux at the nose stagnation point versus altitude from DSMC and CFD for fully-catalytic and non-catalytic wall. These results are compared with the heat flux predicted by the Scott formula, already used in the trade-off design of FTB-X (reference 24). The profiles of the maximum heat flux from DSMC and CFD in fully catalytic wall conditions are similar; both computations agree in stating the maximum value of $q$ since high altitudes ($h \cong 80 \text{ km}$). The fact that DSMC and CFD codes give comparable results, even though based on different approaches, is due to the fully catalytic wall condition that “forces” the solution on the wall. The Scott formula instead underestimates the results from DSMC and CFD; the maximum value of $q$ predicted by Scott, is about 1100 kW/m$^2$, while for the DSMC and CFD codes the maximum value is about 1400 kW/m$^2$.

When the wall is non-catalytic, strong discrepancies appear between DSMC and CFD. In this case DSMC gives results comparable with those from the Scott formula, while CFD calculations underestimate these ones. At $h \cong 80 \text{ km}$, $q$ from DSMC is about 1230 kW/m$^2$, from CFD is about 821 kW/m$^2$. CFD, in noncatalytic wall condition, shows its limits in simulating rarefied flows, because of the failure of phenomenological
models, such as the Fourier and the Fick laws. As a conclusion, also in a trade-off design of TPS an accurate methodology is needed in the prediction of the heat flux. In fact, depending on the catalytic behaviour of the wall, the Scott formula could significantly underestimate the heat flux. Therefore it is important, on one side, to have good characterization of the wall catalicity also in rarefied conditions and, on the other side, to evaluate the heat flux by means of appropriate methods like DSMC.

Moreover, in the transitional zone of the re-entry trajectory where both methodologies can be used, DSMC calculations has to be preferred since thei results are more conservative.

![Figure 5-9 Heat flux at the stagnation point of the FTB-X nose.](image)

Next Figure 5-10 shows, as example of an altitude where both methodologies can be applied (i.e. 77 km), the Mach number contour map for both the used methodologies, for fully catalytic wall conditions. The flow field computed is fair different, since the shock wave is larger for DSMC computations with respect to the Navier-Stokes one. This can be clearly seen from Figure 5-11, where the mach number profile along the stagnation line is plotted. The shock wave thickness, generally expressed as a multiple of the upstream mean free path (see reference 10) is well predicted by DSMC code for all the analyzed cases.
Concluding Remarks

The aerodynamic behaviour of FTB-X (1.1.2) has been evaluated in the altitude interval 90-110 km, where the vehicle is in transitional regime. Computations have been performed both in symmetric and side-slip flights by a 3-D DSMC code. The angle of maximum aerodynamic efficiency, which is the re-entry angle of attack, has been verified to be, at each altitude, 30 deg..

Engineering methods, allowing the computation of the FTB-X aerodynamic parameters in a faster way, have been also tested. The panel methods verified to be
lacking at high altitudes, in particular for an altitude higher than 60 km for the FTB-X re-entry trajectory. On the opposite, the bridging formula by Potter verified to be suitable compared with DSMC data.

The computations of the heat flux at the nose stagnation point have been carried out by a 2-D DSMC code, in axial-symmetric flow, in the altitude interval 60-110 km and considering fully catalytic and non-catalytic wall.

When the most conservative hypothesis of catalytic wall is considered, the maximum thermal load on the nose is about 1400 kW/m$^2$ and the altitude where it is met is about 80 km from both DSMC and CFD codes. The Scott formula underestimates the heat flux; this is a critical point that must be taken into account in the design of the thermal protection system of FTB-X.
Chapter 6 Local Effects of Rarefaction in Shock Wave Boundary Layer Interactions (SWBLI)

In the frame CIRA CLAE technological program, two research activities are being performed in parallel, that are considered critical in the design of a re-entry vehicle: the evaluation of the Shock Wave Boundary Layer Interaction (SWBLI) over a control surface and the analysis of rarefaction effects at high altitude. It is well known that on one side the SWBLI phenomenon strongly affects the efficiency of control surfaces during the re-entry, and on the other side the behaviour of a hypersonic vehicle at high altitude is very different with respect to what happens in continuum regime, for what concerns both the aerodynamic coefficients and the heat loads.

Therefore, the correct evaluation of both phenomena is crucial in the design phase and the influence of the local effects of rarefaction on the prediction of the main parameters typical of Shock Wave Boundary Layer Interaction in depth analyzed.

The SWBLI phenomenon can be simulated in flight by means of advanced numerical codes, provided that a good accuracy of both the numerical scheme and the grid is attained; due to the high energy that typically characterizes the upper part of a re-entry path, the code must be also capable to solve the Navier-Stokes equations taking into account chemical and vibrational non-equilibrium. Since a few experimental data are available for dissociated flows, in the frame of PRORA program some specific tests have been designed to validate the CIRA code H3NS; moreover, the goal of such experiments is to improve the understanding of the phenomenon, in order to be able to correlate flight and wind tunnel conditions and to extrapolate from flight and to flight the experimental data (see references 25, 26 and 27).

However, since the stagnation pressure of a facility like the CIRA Plasma Wind Tunnel (PWT) Scirocco is very low with respect to a classic aerodynamic wind tunnel, a question arises about the possible rarefaction effects that can occur and that can influence the results of the tests; indeed the unit free-stream Knudsen number in Scirocco ranges from a value of 5.375*10^{-5} for the higher pressures, to a maximum of 1.467*10^{-3} in the lower pressure conditions (P₀=1 bar); in the variation of Kn is Figure
6-1 reported as function of the reservoir pressure and enthalpy for both the nozzle D and F; these values could justify local rarefaction effects, for example over the nose, even though much of the flow domain is in the continuum regime (as verified for similar flow conditions by Markelov in reference 28).

![Figure 6-1 Kn vs Reservoir pressure in Scirocco PWT.](image)

The SWBLI phenomenon itself can be affected by rarefaction effects (reference 28). Therefore, the same numerical tools that are typically used in flight to assess the rarefaction effects on the aerodynamic coefficients of a full vehicle have been used in wind tunnel conditions in order to verify if they occur in the experimental test.

First of all, an introduction to Shock Wave Boundary Layer Interaction phenomena is treated in the paragraph 0.

In order to validate the code with the slip conditions, a classic numerical test has been used, and the H3NS results have been compared with both numerical and experimental results available in references 28 and 29 (paragraph 6.2.1). Then the same tools have been applied to rebuild the Scirocco tests, in order to check if rarefaction effects locally occur (paragraphs 0).
6.1. Shock Wave Boundary Layer Interaction

The prediction of mechanical and thermal loads acting on the surfaces of hypersonic vehicles such as RLV is crucial for the design of their aerodynamic shapes and thermal protection. In this context, the interaction of shock waves with boundary layers on these surfaces at relatively high temperatures is an important aspect of high speed flows because of its frequent occurrence in applications and its impact on the behaviour of the boundary layer.

Control surfaces as elevons and wing- and body flaps are embedded in the flow field of the vehicle and are dominated by shock wave/boundary layer interactions that can cause extended separations, laminar-to-turbulence transition and localized intense heating. Moreover, the occurrence of such interactions produces an increase of drag, a variation of hinge moment and a loss of the aerodynamic efficiency of control surfaces, thus making critical the flight control and the structural integrity of the vehicle. Therefore, due to the complexity and the design implications of these phenomena, an understanding of the controlling mechanisms and their quantitative estimation is crucial for the design of the next generation of RLVs.

For two dimensional SWBLI typical of flows over compression ramps, the physics is rather well understood for flows with negligible unsteady effects. The shock induced by the ramp interacts with the boundary layer and, due to the upstream propagation of pressure disturbances across the subsonic portion of the boundary layer, flow separation may occur depending on the local conditions at the location of the interaction (see Fig. 2-1).
The upstream influence, that is the distance from the ramp to the location where the pressure disturbance induced by the ramp is felt through the subsonic part of the boundary layer, may be relatively small for attached flows or it may be dramatic for fully separated flows, its extent depending upon the thickness of the boundary layer and the strength of the shock wave produced by the turning of the flow around the deflected surface.

The presence of the separation bubble causes the formation of the separation and reattachment shocks, whose interaction generates a transmitted shock, a shear layer and, depending upon the Mach number, either a shock wave or an expansion fan that interact with the boundary layer on the ramp. The bubble of recirculation is a region of low speed flow at practically constant pressure, except near separation and reattachment, where the flow deflection increases the pressure levels.

In the separated region the skin friction is negative and in laminar flows the heat transfer reaches a minimum due to the greater boundary layer thickness than lowers temperature gradients. Skin friction and heat transfer rapidly increase downstream of the reattachment due to the recompression of the flow, and have a peak immediately past the reattachment in the proximity of the location where the boundary layer thickness is minimum; in that region, particles coming from the upper part of the boundary layer with very large kinetic energy arrive close to the wall, from which they are deviated and slowed down. This brings to a very large increase in pressure, temperature and heat.
flux, and then they decrease on the ramp due to the boundary layer thickening and flow reacceleration.

Extensive reviews describing the physical phenomena of shock-wave/boundary-layer interaction in hypersonic regime and some correlation laws for incipient separation conditions, characteristic pressures, separation extent, and peak heating can be found in the works of Needham and Stollery (reference 30) Holden (reference 31) and Delery (reference 32). Several experimental test were conducted by Holden in 70’s (references 31 and 33) considering laminar and turbulent interactions from supersonic through hypersonic regime and investigating Mach and Reynolds numbers, ramp angle and leading edge bluntness effects on the flow field in terms of the upstream influence, separation extent and peak heating. In reference 31 it was established that upstream influence increases with ramp deflection angle, decreases with Mach number and it is affected also by the Reynolds number; leading edge bluntness reduces pressure and thermal loads over the compression ramp because, in this case, the interaction occurs in a locally supersonic regime due to the presence of the bow shock. Delery has experimentally shown the dependence of the upstream influence and separation length from the ramp angle showing the they increase, for a given Mach and Reynolds number, when the deflection ramp increase and decrease with Mach number for a given ramp angle and Reynolds number. In reference 32 it was also shown that the main flow features remain similar in laminar and turbulent conditions, the differences being the extent of the interaction, that is, the characteristic scale, and the pressure and thermal loads. Grasso and Marini (references 34, 35 and 36) have studied hypersonic viscous flows dominated by strong shock-wave/laminar boundary-layer interactions over wing–flap and wing–fuselage junction configurations and have assessed the effects of the control surface deflection angle, leading-edge shape, and viscous interaction parameter. Scaling laws for the upstream influence, peak heating, and aerodynamic coefficients have been established by means of numerical simulations and theoretical considerations. Grasso has also characterized different controlling mechanisms of the shock-wave/boundary-layer interaction phenomena and have critically reviewed the various correlation formulas (skin friction, Stanton number, characteristic pressures, and peak heating) applicable in the different regions. Under high enthalpy re-entry conditions, the gas may not always be treated as an ideal gas, and real gas effects such as vibrational excitation and chemical reactions affect significantly such phenomena; their influence (coupled also with turbulent effects) must be necessarily accounted for. Grasso and
Leone (see reference 37) have studied the influence of chemical reactions under the assumption of thermal and chemical equilibrium for interactions over compression ramps. In reference 38 it is shown that due to dissociation reactions (in equilibrium) the temperature is lowered and the shock waves are weakened, as well as their interaction with the boundary layer. The results show a reduction of the separation extent due to real gas effects; however, the peak heating (on the ramp) still correlates with the inviscid pressure jump across the shock. Mallinson et al. (see references 39 and 40) have conducted high-enthalpy compression ramp flow experiments at various stagnation conditions and ramp angles, concluding that real gas effects on the pressure distribution, incipient separation angle, and peak heating are negligible, at least for the conditions they have examined. They concluded that real gas effects on the interaction were negligible under the conditions investigated, because dissociation rates downstream of oblique shocks in shock tunnel flows remain insignificant for moderate shock angles, and dissociation in the boundary layer was found to be negligible even at the highest-enthalpy conditions. The free stream at high enthalpy was partially dissociated, but effects due to recombination were not considered. Davis e Sturtevant (see references 41 and 42) investigated from an experimental and theoretical point of view the effects of thermochemical non-equilibrium in hypersonic region considering a two dimensional geometry; they observed that the recombination reactions within the boundary layer were responsible of the reduction of the separation length. As reported, many experimental and theoretical studies dealing with the shock wave boundary layer interaction over compression ramps were conducted in the past and extensive reviews describing the physical features of a shock wave boundary layer interaction in hypersonic regime and some correlation laws for incipient separation conditions, separation and plateau pressures, separation extent and peak heating can be found in the technical literature. As far as today, however, only few studies have been carried out in order to analyse the shock wave/boundary layer interaction phenomena in high enthalpy real gas conditions, in such a way to complete the theoretical and/or empirical knowledge on the argument (currently based almost exclusively on perfect gas environment) accounting for real gas effects and gas-surface interaction effects.

The evaluation of local effects of rarefaction in such phenomena has been investigated in reference 43.
6.2. Results

6.2.1. Hollow Cylinder Flare Test Case

As a first application and in order to validate the methodologies for the prediction of local effects of rarefaction in hypersonic regime and, in particular, concerning the shock wave boundary layer interaction, a typical experimental test case has been selected: the hollow cylinder flare. The hollow cylinder has a sharp leading edge with a bevel angle of 15 deg. The compression flare is inclined by 30 deg with respect to the cylinder and is ended by a hollow cylindrical section. The model total length is 0.17 m and the reference length $L$ is the distance between the leading edge and the beginning of the compression flare. The experiment was carried out in the R5Ch blowdown hypersonic wind tunnel of ONERA at Chalais-Meudon, France (see references 44, 45 and 46). A complete numerical classic Navier-Stokes investigation has been performed by Marini (reference 29), while a comparison between continuum and kinetic approach compared with experimental results is reported by Markelov et al. in reference 28. A description of the experiment is briefly reported hereinafter.

The flow conditions are for the validation test case obtained under the nominal stagnation conditions $p_0 = 2.5 \times 10^5$ Pa and $T_0 = 1050$ K, which yield an upstream flow characterized by the following properties: $M_\infty = 9.91$, $R_\infty / m = 1.86 \times 10^5$, $T_\infty = 51K$ and $p_\infty = 6.3 Pa$. The surface temperature of the model is assumed to be constant and equal to $T_w = 293$ K, the Reynolds number based on the reference length $L = 0.1017 m$ is equal to, the mean free path $\lambda_\infty = 9.5103 \times 10^{-4} m$ and the Knudsen number:

\textbf{Equation 6-1} \quad Kn_\infty = \frac{\lambda}{L} = 9.35 \times 10^{-4}

It is worth underline that much of the flow domain is in continuum regime, except, obviously the sharp leading edge, and along the surface where local effect of rarefaction are significant (see reference 28).

A sketch of the geometric configuration is reported in Figure 6-3 together with the computational grid. The following experimental data are available from references 44, 45 and 46: i) pressure coefficient surface distribution (accuracy of 2%), ii) Stanton number (from temperature variations at wall) surface distribution (accuracy of 7%), iii)
surface oil-flow visualization (the pattern of skin friction lines over the model indicates the separation and attachment lines, respectively at $X_{\text{sep}}/L = 0.76 \pm 0.01$ and $X_{\text{rea}}/L = 1.34 \pm 0.015$), iv) flow field visualization by the Electron Beam Fluorescence (EBF) technique and v) density profile measurements by X-rays detection at the streamwise sections $x/L = 0.3; 0.6$ (accuracy of 15%) and $x/L = 0.76$ (accuracy of 8%).

The two-dimensional grid, used for both classic wall boundary conditions and slip flow ones, is a structured grid composed by 14 blocks, 165888 cells and 169062 points. The grid convergence has been checked. With regard to DSMC simulation, the location of separation has been used as parameter to check the needed number of molecules, since it is the most sensitive property to the variation of number of molecules. Table 6-1 summarizes the performed simulations and shows that the third solution is the converged one.

<table>
<thead>
<tr>
<th>Solution</th>
<th>Molecules</th>
<th>$X_{\text{sep}}$ [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$8.116 \times 10^6$</td>
<td>787.98</td>
</tr>
<tr>
<td>2</td>
<td>$12.174 \times 10^6$</td>
<td>788.34</td>
</tr>
<tr>
<td>3</td>
<td>$18.258 \times 10^6$</td>
<td>816.64</td>
</tr>
</tbody>
</table>

Table 6-1 Influence of number of molecules on separation.

The next Figure 6-4 and Figure 6-5 show the predicted Mach number contour maps and streamlines for CFD with slip boundary conditions and DSMC computations: the strong viscous interaction at the cylinder leading edge appears as well as the evident
shock wave boundary layer interaction around the corner, and the subsequent recirculation bubble.

Figure 6-4 CFD Slip: Mach number contours and streamlines.

Figure 6-5 DSMC: Mach number contours and streamlines.

Figure 6-6 displays the normalized slip velocity wall distribution predicted by DSMC and CFD with the boundary conditions of Equation 4-17 and Equation 4-18. In the first part of the wall, CFD overestimates the results predicted by DSMC, while downstream of the separation the matching between the two methodologies is rather good. It is worth to underline that slip velocity along the wall, and in particular ahead of
separation location, reaches about 10% of freestream velocity, therefore not negligible rarefaction effects are expected.

Figure 6-6 Slip velocity distribution.

Figure 6-7 exhibits the pressure coefficient behaviour on the wall for the three calculations (CFD No Slip, CFD Slip and DSMC) compared to experimental results.

A low initial decrease of pressure coefficient is predicted due to the strong viscous interaction, followed by a first increase of Cp at separation and, then, a strongest one (forty times higher) due to the reattachment shock wave. All methods
(CFD No Slip, CFD Slip and DSMC) show a good agreement with measured values. All computations overvalue the $C_p$ peak at reattachment, and classic CFD is the nearest one to the experimental value since at this point the conditions are close to continuum ones, being the local Knudsen number based on boundary layer thickness $\delta$, evaluated as:

\begin{equation}
\text{Equation 6-2} \quad Kn_\delta = \frac{\lambda}{\delta} \approx 5.28 \times 10^{-3}
\end{equation}

The analysis of skin friction distribution (Figure 6-8 and Figure 6-9) shows a different prediction of the three used methodologies for the location of separation. In particular, the predicted separation length is larger as the “amount of rarefaction of the method” decreases. In fact, the value of $X_{sep}/L$ for CFD No Slip, CFD Slip and DSMC, is respectively 0.7245, 0.7555 and 0.7748. The experimental value (0.76) is included between CFD Slip and DSMC values as a proof of the local effects of rarefaction, being the local Knudsen number:

\begin{equation}
\text{Equation 6-3} \quad Kn_\delta = \frac{\lambda}{\delta} \approx 2.17 \times 10^{-2}
\end{equation}

![Figure 6-8 Skin friction coefficient distribution.](image)
Figure 6-9 Skin friction coefficient distribution- zoom.

Note that the reattachment point is not significantly affected by the computation methodology (Figure 6-8).

The same considerations made in the framework of the analysis of skin-friction and pressure coefficients apply to the Stanton number distribution (see Figure 6-10).

Figure 6-10 Stanton number distribution.

Note that on the leading edge Navier-Stokes with classic boundary conditions (CFD No Slip) overestimates the Stanton number, in fact in this region the boundary layer thickness $d$ is of the same order of magnitude of the mean free path $\lambda$, so

\[ Kn_\delta = \frac{\lambda}{\delta} \approx o(1) \]
and the rarefaction effects are maximum.

A comparison of the results obtained by means of numerical simulations with the measured normalized density profiles inside the boundary layer is reported in Figure 6-11 and Figure 6-12.

In the cross section X/L=0.3 the correlation between numerical and experimental data in the case of Navier-Stokes with slip flow boundary conditions and DSMC is good, while it is fair for CFD with the classic boundary condition (CFD No Slip). In particular, the shock wave inclination predicted by classic CFD is higher with respect to measured one (compare also the Mach number contours of Figure 6-4 and Figure 6-5).

At the section X/L=0.76 the three numerical methodologies are closer each other and a good agreement with experimental results has been obtained. Note that the predicted inclination of shock wave is nearly the same.

Figure 6-11 Density profile: X/L=0.3.
In conclusion, the analysis of the two density profiles shows that the rarefaction effects, as expected, are more significant in the region near the leading edge.

Definitively, the analysis of the present wind tunnel test case simulated with the three different methodologies (classic CFD, CFD with slip flow boundary conditions and DSMC) has shown that local rarefaction effects are significant for the prediction of important aspects of shock wave boundary layer interaction as the sizing of recirculation bubble. Moreover, it has been also shown that CFD with slip flow boundary conditions is, in this case, a good compromise between computational cost and accuracy.

### 6.2.2. CIRA Plasma Wind Tunnel Test

Within the ESA EXPERT program, a number of experiments to be performed in the CIRA Plasma Wind Tunnel “Scirocco”, representative of the capsule flight conditions with respect to the shock wave boundary layer interaction phenomenon occurring around the 20-deg open flap, has been designed: PWT driving conditions, model configuration and attitude and model instrumentation have been defined, by means of a massive CFD activity performed by using the CIRA code H3NS, in order to duplicate on a forebody full-scale flap model both pressure and heat flux levels estimated in critical flight conditions.

The model, to be used for PWT test campaigns foreseen within 2008, reproduces the EXPERT capsule full-scale flap (scale 1:1) characterized by 20 deg deflection angle;
a sketch is shown in Figure 6-13. In order to be consistent with the EXPERT capsule, the model will be built by using as much as possible the same materials: in particular the part ahead the flap is covered by a flat plate of PM1000 (or similar material) equipped with pressure taps, thermocouples and combined heat flux/pressure sensors, whereas the flap is covered by a 4mm thick plate of C-SiC equipped with pressure taps and thermocouples. The cylinder leading edge has a radius of 100 mm and a length of 400 mm, the flat plate is 400 mm wide and 200 mm long, the flap is 400 mm wide and 300 mm long. All the lateral edges are rounded with a radius of 50 mm in order to avoid localized over heating, whereas the flap C-Si plate has a radius of curvature at the lateral edges equal to 4mm (i.e. its thickness). Detailed 2D and 3D computations of the flow around the model with proper thermal and catalytic modeling of the surface have been carried out in different PWT operating conditions determined to duplicate either the SWBLI phenomenon around the body flap and the associated thermo-mechanical loads acting on it during the EXPERT re-entry flight (for details see references 25, 26 and 27).

![Figure 6-13 Model configuration.](image)

Since the goal of the present analysis is the verification of rarefaction effects, only 2D section has been simulated; moreover one of the conditions with the lower possible pressure is taken into account. In particular the conditions $H_0=35$ MJ/kg, $P_0=2$ bar have been analyzed (corresponding to the lower freestream Knudsen number: $1.47 \times 10^{-3}$) by comparing the results obtained with a classical Navier-Stokes approach and DSMC.
method. Specifically, this test case is characterized by the following flow properties: $M_\infty = 12.94$, $R_e / m = 9.03 \times 10^3$, $T_\infty = 240K$ and a model attitude of 12 deg. A grid-independence study for CFD simulations has been carried out (reference 27) as well as a study of DSMC solution sensitivity to the number of particles (not shown). Note that for this high enthalpy case it has been decided to not perform the CFD slip computation since more accurate DSMC calculations are not strongly CPU-time demanding due to the reduced number of needed particles (i.e. $10.910^6$ molecules).

A preliminary analysis has been carried out considering the wall at fixed temperature of 300 K, and the following Figure 6-14 and Figure 6-15 show the Mach number contours and the streamlines for the two performed computations. Figures show the strong bow shock wave ahead of the model, that is more inclined, as expected, in the case of DSMC simulation, the strong expansion on the bottom part of the model, and finally the shock wave boundary layer interaction around the corner and the subsequent recirculation bubble, that is in incipient conditions in the case of rarefied flow simulation.

![Figure 6-14 CFD: Mach number contours and streamlines.](image)
Figure 6-15 DSMC: Mach number contours and streamlines.

Figure 6-16 exhibits the slip velocity wall distribution predicted by DSMC calculation showing a peak value of about 1.3% of freestream velocity in correspondence of the beginning of the flat plate downstream of the model nose. It can be underlined that these low values of slip velocity were expected since, differently from the validation test case (i.e. the hollow cylinder flare), no sharp leading edge is present in this PWT model, therefore continuum regime flow conditions are predicted around the nose. Looking also at Figure 6-6, it can be observed that the same qualitative cuspid-like distribution has been predicted in correspondence of the corner, where a separation (or incipient separation like in this case) occurs.

Figure 6-16 Slip velocity distribution.
By carefully examining Figure 6-17 and Figure 6-18, and remembering the analysis performed for the validation test case, the same considerations apply to the present applicative case in high enthalpy conditions. In particular, a reduction of separation extent is observed with DSMC calculation (Figure 6-17 and Figure 6-18), as well as a slight reduction of the mechanical load acting on the flap (see Figure 6-17).

![Figure 6-17 Pressure coefficient distribution.](image)

![Figure 6-18 Skin friction coefficient distribution.](image)

Finally, also looking at Figure 6-16, in correspondence of the section where the maximum of slip velocity occurs, i.e. X=0.1 m, the local Knudsen number is:

**Equation 6-5** \[ Kn_δ = \frac{\lambda}{\delta} = 4.05 \times 10^{-2} \]
and this value justifies the occurrence of local effects of rarefaction on the prediction of important aspects of shock wave boundary layer interaction as well as the extent of separation region.

As a conclusion, it must be stressed the fact that local rarefaction effects must be taken into account when designing plasma wind tunnel tests at limit conditions of the facility envelope, in particular for very low pressures and high enthalpies as in the present case.

This is particularly true when plasma test requirements are represented by the reproduction on the test model (or on parts of it) of given values of mechanical and thermal loads, as well as of shock wave boundary layer interaction characteristics (i.e. separation length, peak of pressure, peak of heat flux, etc.).

6.3. Concluding Remarks

Local effects of rarefaction on Shock-Wave-Boundary-Layer-Interaction have been studied by using both the continuum approach with the slip flow boundary conditions and the kinetic one by means of a DSMC code.

The hollow cylinder flare test case for R5Ch wind tunnel conditions was numerically rebuilt in order to validate the methodologies. The free stream Knudsen number for the selected test case implies that much of the flow is in continuum conditions, even though local effects of rarefaction have been checked. In particular, the comparison with experimental data has shown that rarefactions effects are not negligible in prediction of the separation length. The CFD code with slip flow boundary conditions has shown good predicting capabilities of the size of the recirculation bubble and the analysis of the density profiles inside boundary layer has shown a good agreement between DSMC Figure 6-16 slip velocity and CFD with slip conditions in different sections along the body. Definitively, the present wind tunnel test case, simulated with the three different methodologies (classis CFD, CFD with slip flow boundary conditions and DSMC), has shown that local rarefaction effects are significant for the prediction of important aspects of shock wave boundary layer interaction as the sizing of recirculation bubble and it has been also shown that CFD with slip flow boundary conditions is, in this case, a good compromise between computational cost and accuracy.
The same considerations apply to a CIRA Plasma Wind Tunnel test case planned within 2008, where significant rarefactions effects were found on the SWBLI phenomenon; therefore they must be taken into account when designing plasma wind tunnel tests at limit conditions of the facility envelope, in particular for very low pressures and high enthalpies as in the present case.
In the previous chapters some important aspects to be taken into account in design of a space winged re-entry vehicle (FTB-X), such as the high altitude aerodynamic and nose heating. Moreover, the influence of rarefaction in control surfaces has been investigated. This aspect are strictly connected to the design of FTB-X vehicle.

FTB-X technological program CLAE is also related to RTO-RTG043 international working group in which the Italian Aerospace Centre (CIRA) is involved, as the “nose and leading edge” task is related to the high altitude simulation on NASA ORION CEV capsule. The performed results and the comparison with the other partners of the group will be very useful for the assessment of the technologies that will be used for FTB-X design.

Moreover, the present scientific investigation has to be completed with the application of the methodologies in evaluation of the rarefaction effects on a capsule, than CEV aerothermodynamic in transitional zone has been analyzed.

The first paragraph describes RTO-RTG043 working group, while the next shows the obtained results.

7.1. RTO-RTG043 Working Group

The main purpose of Research Task Group (RTG)-043 of the NATO Research and Technology Organization (RTO) is to identify and mitigate current inadequacies in present aerothermal prediction capabilities by advancing our understanding and modeling of the relevant physics and by incorporating these advancements in evolving aerothermal design tools. Different data are needed to this scope: assess learned from past (gaps, uncertainties), assess measurement techniques, conduct experiments
(improved measurement techniques) and conduct computational simulations of experiments.

In the framework of RTO-RTG043 six work topic are planned:

1. Nose and Leading Edges;
2. Shock Interactions and Control Surfaces;
3. Chemical Kinetics and Radiation;
4. Boundary Layer Transitions;
5. Gas/Surface Interactions and Base

In this work Nose and Leading Edges Work topic will be presented and analyzed. In this topic, six test cases have been proposed:

1. Mars Science Laboratory (MSL): numerical rebuilding of experimental test cases performed in different NASA ground facilities to be used to validate CFD tools;
2. FIRE II: numerical simulations for FIRE II flight conditions and assess the current capabilities to predict heating along the forebody;
4. Tethers: Leading edges simulations in rarefied flows with 3D interactions;
5. Leading Edge Heating with Shock Interactions: measurements and numerical simulations;
6. Sharp Hot Structures: coupled aero-thermal simulations on a CIRA SCIROCCO plasma wind tunnel test case;

In this framework, CIRA is involved in the Nose and leading Edge work topic, referring to the evaluation of High-Altitude Aerothermodynamics of Orion Crew Exploration Vehicle (CEV). In particular, the scope of the present investigation is to provide information in the transitional part of the trajectory, where overlap between continuum approach and particle one occurs; moreover a sensitivity analysis with respect to the surface boundary conditions was performed. Moreover and a comparison between different results, coming from all the involved contributors.
7.2. Test Conditions and Results

CEV reference geometry is shown in the next figure:

Figure 7-1 ORION Crew Exploration Vehicle: reference geometry.

The baseline flight conditions suggested in the framework of the Nose and Leading Edges activities for CEV vehicle are representative of conditions that take place during the re-entry trajectory from the International Space Station (ISS) and corresponding to altitudes of 125, 115, 105, 95, 85, and 75 km.

The baseline conditions are the following: fixed wall temperature, freestream velocity of 7.6 km/s for all altitudes, axis-symmetric geometry and a zero angle of attack. In addition, a sensitivity analysis was performed with respect to the surface catalycity.

Freestream baseline conditions as a function of altitude are listed in the following table:
An in-depth aerodynamic analysis has been conducted by Moss in references 47 and 48, while an aerothermodynamic analysis in transitional regime has been treated in reference 49.

### 7.2.1. Grid and Molecular Independence

A sketch of two-dimensional CFD grid is shown in Figure 7-2, where the shock fitting is clear.

![Figure 7-2 CFD computational Grid.](image)

The computational grid is composed by three blocks and 42400 cells, with three grid levels. In order to check the grid convergence an analysis on different grid levels has been conducted. In particular, grid level 3 and level 2 (10600 cells) results have been compared for the altitude of 95 km (see Table 7-1), for a fully catalytic wall conditions and a fixed wall temperature (i.e. $T_w=951$ K).

The heat flux (variable more critical in variation with grid size) computed by the two run with a different levels is very close each other (Figure 7-3), so a grid independence has been verified.
For what concerns the maximum value of the ratio of the local mean separation between collision partners to the local mean free path $\frac{mcs}{\lambda}$ is 0.16 for the 95 Km case and 0.69 for 85 Km one.

### 7.2.2. Slip Flow Boundary Conditions Validation

In order to validate these formulas, the 85 km altitude as test conditions of ORION re-entry trajectory has been chosen as reference case to compare the results obtained by means of the *continuum* approach with slip flow correction and the ones obtained by means of the DSMC code, since in these conditions both methodologies can be applied.

Figure 7-4 shows the good agreement between the slip velocity distribution at wall predicted by CFD with the boundary conditions (1) and DSMC result for a fully catalytic wall.
Figure 7-4  Slip velocity distribution; altitudes 85 km; fully catalytic wall; T\(_w\)=1184 K.

Also the predicted slip temperature (Figure 7-5) obtained by means of the CFD code is similar to the DSMC one. In particular, the slip temperature calculated using the CFD code is slightly higher with respect to the one computed by the particle code. This discrepancy is caused by the different modelling of temperature at wall. In fact, in the CFD code, the translational and rotational temperatures are assumed in equilibrium, being only the vibrational temperatures considered in non-equilibrium, while a three-temperature gas model is implemented in DSMC code (rotational temperature T\(_r\), translational temperature T\(_t\), vibrational temperature T\(_v\)).

Figure 7-5  Slip temperature distribution; altitudes 85 km; fully catalytic wall; T\(_w\)=1184 K.
The implemented slip conditions are also able to predict the variation with fixed wall temperature, in fact, as shown in the next figures, for a $T_w$ equal to 300 K instead of 1184 K, the considerations done for the nominal case remains.

![Figure 7-6](image1.png)  
**Figure 7-6** Slip velocity distribution; altitudes 85 km; fully catalytic wall; $T_w=300$ K.

![Figure 7-7](image2.png)  
**Figure 7-7** Slip temperature distribution; altitudes 85 km; fully catalytic wall; $T_w=300$ K.

### 7.2.3 Results

The focus of the present investigation is mainly on the 85 km and 95 km cases. For these two trajectory points, both CFD and DSMC computations have been performed, including the sensitivity analysis with respect to some boundary conditions.
The next Figure 7-8 shows the pressure iso-lines for the altitude of 85 km and fully catalytic wall condition. The flow field calculated by DSMC tool is compared with the one calculated by CFD code (no significant difference is visible in the flow fields computed with and without slip flow boundary conditions), while Figure 7-9 exhibits the pressure profiles along the stagnation line for all the analyzed methodologies. It can be clearly seen that the shock wave thickness computed by the particle code is larger with respect to the Navier-Stokes codes; this value, generally expressed as a multiple of the upstream mean free path (reference 10) is well predicted by DSMC code for all the analyzed cases.

**Figure 7-8** Pressure iso-lines; altitudes 85 km; fully catalytic wall; $T_w=1184$ K.
At 85 km of altitude, fully catalytic wall and $T_w=1184$ K, the wall pressure computed by all methodologies (DSMC, CFD and CFD with slip conditions) is very close each other (see Figure 7-10).

The global Drag coefficient is very similar between all the performed calculations being the value of pressure very close to zero where it varies with the used methodology. It has been verified that the catalysis and the temperature at wall does not influence the pressure profile at wall.
Also at 95 km, in the forebody region and after the shock wave, all the used methodologies show a similar behaviour in prediction of wall pressure (see Figure 7-11), instead (Figure 7-12), some slight differences can be seen in the expansion zone (after the shoulder), where the flow is more rarefied (the local Knudsen number based on the boundary layer thickness is about $10^{-1}$); in this zone the slip flow correction brings the CFD result to be closer to the DSMC one. Therefore, the drag predicted by CFD and DSMC simulations is very similar, greater differences are expected at higher altitudes. As expected, the wall catalysis does not influence the wall pressure profiles.

![Figure 7-11](image1.png)

**Figure 7-11** Pressure profile; altitude=95 km; fully catalytic wall; $T_w=951$ K.

![Figure 7-12](image2.png)

**Figure 7-12** Pressure profile; altitude=95 km; fully catalytic wall; $T_w=951$ K- zoom.

The Figure 7-13 and Figure 7-14 show, for the two trajectory points, the surface heat flux calculated by DSMC tool and CFD ones (with and without slip flow boundary...
conditions). For both the altitudes, DSMC result is about 10% higher with respect to continuum results.

As expected, the heating reaches the maximum value in the stagnation region, in the shoulder region where the strong expansion occurs a second peak is predicted by both CFD calculations but not by DSMC. It must be noted that the effect of the slip conditions in the CFD computations is very small; moreover, if one computes the heat flux by means of classical Fourier formula, a decrease of heat flux is predicted with respect to the "slip" case. Definitely, the slip flow correction gives good results in prediction of wall parameters such as slip velocity (see Figure 7-5), temperature and pressure, but not in surface heat flux calculation.

**Figure 7-13** Heat flux distribution; altitudes 85 km; fully catalytic wall; $T_w=1184$ K.

**Figure 7-14** Heat flux distribution; altitudes 95 km; fully catalytic wall; $T_w=951$ K.
An evaluation of the single contributions to the global heat flux is shown in the Figure 7-15, for 85 km of altitude and fully catalytic conditions. In particular, it can be seen that the chemical contribution to the heat flux calculated by CFD is very close to DSMC one, while the conductive one is lower. This difference on global heat flux is confirmed from non-catalytic wall case showed in the next the Figure 7-16. It is interesting to note that in the non catalytic case the percentage difference between CFD and DSMC results increases from about 10% to 60%, but the absolute value of this difference is the same as for fully catalytic wall; this confirms that the discrepancy is not due by the chemical part of the heat flux, but from the conductive one. The same considerations apply to the altitude 95 km case.

![Figure 7-15](image)

**Figure 7-15** Heat flux distribution; altitude=85 km; fully catalytic wall.
Figure 7-16 Heat flux distribution; altitude=85 km; non-catalytic wall.

For the higher altitude points (i.e., 105 km, 115 km and 125 km) of the reference of the Table 7-1 continuum modelling is no longer applicable; therefore they have been computed only by means of DSMC code. The 75 km case instead has been computed only by means of CFD code, because DSMC tool should need too much RAM Memory. The next figure summarizes the heat flux of stagnation point for all the considered cases computed by the correct methodology for each altitude. As a reference, the Fay-Riddel formula, typically used for preliminary design of the Thermal Protection System (TPS), has been plotted. In the zone where both modelling could be applied (i.e., 85 km and 95 km), DSMC values should be used, since CFD and the approximate formula underestimate the heat flux and, therefore, a correct prediction of rarefaction effects is needed.
7.3. **Concluding Remarks**

In the framework of RTO-RTG043 working group the axis-symmetric aerothermodynamic of ORION Crew Exploration Vehicle (CEV) in the transitional part of the re-entry trajectory has been presented.

To this aim, the analysis has been mainly conducted on the altitudes 85 km and 95 km, corresponding to a value of free stream Knudsen number 0.0019 and 0.01 respectively. Two theoretical approaches have been used: particle approach (DSMC) and a continuum one (CFD with and without slip flow boundary conditions).

Shock wave thickness calculated by DSMC code is larger with respect to CFD and in agreement with theoretical considerations for all the analyzed cases. All methodologies show a good match in prediction of the wall pressure in the forebody region, while some discrepancies appear in the expansion and base regions and increase with rarefaction of the gas (altitude). The global aerodynamic forces are not affected by the used methodology, being the differences located in a region where the pressure is much lower with respect the forebody region.
The surface heat flux calculated by DSMC code is about 10% higher than CFD one for the fully catalytic wall assumption, while this difference raises to about 60% in the case of non catalytic wall, since the discrepancy is mainly due to the conductive heat flux. Slip flow boundary conditions (equations 1) implemented to take into account the effects of rarefaction of gas in the CFD code, show good results in prediction of slip velocity, slip temperature and pressure, but not in the evaluation of the heat flux, as the higher value of the wall temperature causes an underestimation. Therefore, in general, DSMC should be preferred with respect to CFD in the prediction of heat fluxes.
Chapter 8 Conclusions

In the frame of the Italian National Space Program (PRORA), the Italian Aerospace Research Centre (CIRA) is carrying on the design of an experimental Unmanned Space Vehicle (USV) to be used as a flight test bed (FTB) for the re-entry technologies.

The present Ph. D. thesis relies on technological program CLAE being developed by CIRA to study the main problems occurring during FTB-X re-entry, since one of the these investigations relies on the analysis of rarefaction effects occurring around at high altitude in hypersonic flow conditions.

In particular the purpose of the present research activity is the study of the capabilities in prediction of aero-thermal loads acting on a space re-entry vehicle at high altitude and improves understanding and modelling of relevant physics in rarefied flows.

The focus of this investigation is to provide information on the most challenging problems to be solved in design of a space re-entry vehicle:

1. Aerodynamic efficiency,
2. Blunt body heating
3. Efficiency of the control surfaces.

These problems will be analyzed in low density conditions. It is worth to underline that such kind of studies are still under investigation by the hypersonic community, since the old space re-entry vehicle pass through the transitional zone of the re-entry path rapidly, performing a ballistic re-entry (with an high angle of attack) differently from FTB-X.

So, the improvement of the present research activity is the understanding the study of the thematic 1, 2 and 3 in conditions (i.e. low density) not yet in depth analyzed.

The attention, in all the conducted analyses, has been focused by comparing continuum approach (CFD with and without slip flow boundary conditions) and particle one (DSMC), where both modelling could be used.

For what concern the first problem, the aerodynamic behaviour of FTB-X (1.1.2) has been evaluated in the altitude interval 90-110 km, where the vehicle is in transitional regime, since one of the goals of FTB-X is the high efficiency at high
altitude to perform a long re-entry to land in different earth sites. Computations have been performed both in symmetric and side-slip flights by a 3-D DSMC code. The angle of maximum aerodynamic efficiency, which is the re-entry angle of attack, has been verified to be, at each altitude, 30 deg.

Engineering methods, allowing the computation of the FTB-X aerodynamic parameters in a faster way, have been also tested. The panel methods verified to be lacking at high altitudes, in particular for an altitude higher than 60 km for the FTB-X re-entry trajectory. On the opposite, the bridging formula by Potter verified to be suitable compared with DSMC data.

The computations of the heat flux at the nose stagnation point have been carried out by a 2-D DSMC code and CIRA CFD code H3NS, in axial-symmetric flow, in the altitude interval 60-110 km and considering fully catalytic and non-catalytic wall.

When the most conservative hypothesis of catalytic wall is considered, the maximum thermal load on the nose is about 1400 kW/m² and the altitude where it is met is about 80 km from both DSMC and CFD codes. The Scott formula underestimates the heat flux; this is a critical point that must be taken into account in the design of the thermal protection system of FTB-X. In non catalytic wall conditions, CFD code greatly underestimates the heat flux with respect DSMC.

The analysis of the flow fields computed by DSMC and CFD codes for both the wall assumptions (i.e. fully catalytic and non-catalytic wall), are fair different and, in particular, the shock wave thickness, generally expressed as a multiple of the upstream mean free path is well predicted by DSMC code.

Local effects of rarefaction on Shock-Wave-Boundary-Layer-Interaction for the evaluation of efficiency of the control surfaces have been studied by using both the continuum approach with the slip flow boundary conditions and the kinetic one by means of a DSMC code.

The hollow cylinder flare test case for R5Ch wind tunnel conditions was numerically rebuilt in order to validate the methodologies. The free stream Knudsen number for the selected test case implies that much of the flow is in continuum conditions, even though local effects of rarefaction have been checked. In particular, the comparison with experimental data has shown that rarefactions effects are not negligible in prediction of the separation length. The CFD code with slip flow boundary conditions has shown good predicting capabilities of the size of the recirculation bubble and the analysis of the density profiles inside boundary layer has shown a good agreement.
between DSMC slip velocity and CFD with slip conditions in different sections along the body. Definitively, the present wind tunnel test case, simulated with the three different methodologies (classic CFD, CFD with slip flow boundary conditions and DSMC), has shown that local rarefaction effects are significant for the prediction of important aspects of shock wave boundary layer interaction as the sizing of recirculation bubble and it has been also shown that CFD with slip flow boundary conditions is, in this case, a good compromise between computational cost and accuracy.

The same considerations apply to a CIRA Plasma Wind Tunnel test case planned within 2008, where significant rarefactions effects were found on the SWBLI phenomenon; therefore they must be taken into account when designing plasma wind tunnel tests at limit conditions of the facility envelope, in particular for very low pressures and high enthalpies as in the present case.

The aerothermodynamic in rarefied conditions has also been analyzed for a re-entry capsule, then, in the framework of RTO-RTG043 working group, the axisymmetric aerothermodynamic of ORION Crew Exploration Vehicle (CEV) in the transitional part of the re-entry trajectory has been studied.

The analysis has been mainly conducted on the altitudes 85 km and 95 km, corresponding to a value of free stream Knudsen number 0.0019 and 0.01 respectively. Two theoretical approaches have been used: particle approach (DSMC) and a continuum one (CFD with and without slip flow boundary conditions).

Shock wave thickness calculated by DSMC code is larger with respect to CFD and in agreement with theoretical considerations for all the analyzed cases. All methodologies show a good match in prediction of the wall pressure in the forebody region, while some discrepancies appear in the expansion and base regions and increase with rarefaction of the gas (altitude). The global aerodynamic forces are not affected by the used methodology, being the differences located in a region where the pressure is much lower with respect the forebody region.

The surface heat flux calculated by DSMC code is about 10% higher than CFD one for the fully catalytic wall assumption, while this difference raises to about 60% in the case of non catalytic wall, since the discrepancy is mainly due to the conductive heat flux. Slip flow boundary conditions implemented to take into account the effects of rarefaction of gas in the CFD code, show good results in prediction of slip velocity, slip temperature and pressure, but not in the evaluation of the heat flux, as the higher value
of the wall temperature causes an underestimation. Therefore, in general, DSMC should be preferred with respect to CFD in the prediction of heat fluxes.

Definitively, the experience done with the present research activity had allowed to develop the Know-how needed to cover an entire re-entry trajectory, from the higher altitudes (fee molecular flow regime), through the transitional zone, till the *continuum flow*; this skill will be used in the next phases of FTB-X design.
References


Ringrazio il Prof. Aldo Bonfiglioli, il Prof. Vinicio Magi e l'Università degli Studi della Basilicata.