Chapter 4

4.1 Summary:

Incompressible flow with zero Mach number is of no physical significance. It is a mathematical definition only and represents stagnation state. So in engineering applications we assume the flow incompressible if its density change is less than 5% of the free stream density. Based on the flow Mach number, compressible flow can be divided into.

1. Subsonic (M = 0.3 to 1)
2. Supersonic (M = 1 to 5)
3. Hypersonic (M = 5 to 40)

While solving the 3-D motion problem including shocks, friction and heat transfer, we assume the simple modes of flow, which themselves demand the analytical treatment but simultaneously provides the valuable information regarding the real and difficult flow patterns. Following points should remember:

- For the region of potential flow, we use Prandtl’s boundary layer concept.
- For compressible flow the basic potential equation used.
- Disturbances and Mach lines can be produced only by boundaries and can travel only in the downstream direction.
- In compressible flow, the general equation can be solved for flow past slender or planer bodies. Planer bodies have one dimension smaller than others like wing etc.
- In small perturbation theory, it is assumed that there is no significant contribution of velocities components.

\[ u \ll V_\infty, \]
\[ v \ll V_\infty \]
\[ w \ll V_\infty \]

Therefore \( V_x \ll V_\infty \)
\[ V_x \ll V_\infty \]
\[ V_y \ll V_\infty \]
\[ V_z \ll V_\infty \]

It is valid for all type of flow for small perturbations.

After solving the equations, the linearized potential flow equation is obtained.

Potential flow equation for transonic flow (\( M_\infty \approx 1 \)) is
\[ -(\frac{r+1}{V_\infty})\phi_x \phi_{xx} + \phi_{yy} + \phi_{zz} = 0 \]
Due to non-linear nature of this equation it is difficult to obtain solution for transonic flow than subsonic or supersonic flow problems. Commonly used bodies of revolution are fuselage of airplane, rocket shells, missile bodies and circular body etc. In cylindrical coordinates, equation for subsonic and supersonic flow it.

$$(1 - M_{\infty}^2) \phi_{xx} + \phi_{rr} + \frac{1}{r} \phi_r + \frac{1}{r^2} \phi_{\theta\theta} = 0$$

Equation for transonic flow is-

$$-\left(\frac{\gamma + 1}{V_\infty}\right) \phi_x \phi_{xx} + \phi_{rr} + \frac{1}{r} \phi_r + \frac{1}{r^2} \phi_{\theta\theta} = 0$$

In case of axially symmetric, subsonic and supersonic flow, becomes zero, then equation reduced to

$$(1 - M_{\infty}^2) \phi_{xx} + \phi_{rr} + \frac{1}{r} \phi_r = 0$$

In the same way, transonic equation become,

$$-\left(\frac{\gamma + 1}{V_\infty}\right) \phi_x \phi_{xx} + \phi_{rr} + \frac{1}{r} \phi_r = 0$$

Equations obtained for subsonic and supersonic flow are linear under small perturbation but the same is non-linear for transonic flow. Subsonic and supersonic flow equations do not involves the term $r$ but transonic flow for equation contains $r$. Under small perturbation effect. The result we get, can be applied to any gas but not applicable to transonic flows. For slender bodies, these equations are valid. For e.g. Missiles, rockets etc.

These equations are also applicable to aerofoils but not for bluff shape bodies. Flow for non-slimber bodies can be calculated by original nonlinear equation. The non-dimensional difference between local and free stream pressure is known as pressure coefficient.

Calculating the velocity and pressure distribution lift, moment and pressure drag can be find out easily.

Pressure coefficient for 3-D flow is given as:

$$C_p = \frac{2}{\gamma M_{\infty}^2} \left[ \frac{\gamma - 1}{2} \left( \frac{V_\infty + u}{V_\infty} \right)^2 + \frac{v^2 + u^2}{V_\infty^2} \right] + 1$$

Neglecting higher order terms after expanding binomially we have.

$$C_p = -2 \frac{u}{V_\infty} + (1-M_{\infty}^2) \frac{u^2}{V_\infty^2} + \frac{v^2 + u^2}{V_\infty^2}$$

In case of planer bodies, we get.

$$C_p = -2 \frac{u}{V_\infty}$$

This fundamental equation is equally applicable to compressible and low speed two dimensional flows.
Streamlines for compressible flow is \(\frac{1}{\sqrt{1-M_\infty^2}}\) times than in incompressible flow. With increase in thickness ratio profile, critical Mach number decreases. Value of lift coefficient are same in compressible and incompressible flow. Occurrence of shock and boundary layer separation are characteristics of transonic flow. For small perturbation, shock should be sufficiently weak. Small disturbance can be felt, when disturbance at a line and which causes came generation are known as lines of Mach.

It can be structured as:
\[
\sin u = \frac{a_t}{v_t} = \frac{a}{v}
\]

Shock can be defined as the regions where parameter values very small. Shock thickness is nearly equal to the mean free path of gas molecules in flow field. Relation between characteristics Mach number and actual Mach number is given by.
\[
M^* = \frac{(\gamma+1)M^2}{(\gamma-1)M^2+2}
\]

Across the shock, change in flow properties is confined to very small distance, which is of the order of 10cm. This is the reasons why velocity and temperature gradients in the shock are very large. These gradients are responsible for increase in entropy, heat conduction and viscous dissipation. For a moving shock, neither enthalpy nor total temperature remains constant. Normal shock waves are special case of oblique shock waves. Pressure, density and temperature increases and Mach number decreases across the shock wave. On the basis of dimensionless parameters shock can be termed as weak, and strong.

* If downstream Mach number > 1, then attached wave is weak oblique
* If Mach number < 1, strong oblique meets

Strong oblique shocks are generated in the engine intakes of supersonic flight vehicle which controls the backpressure.

Mach angle – Mach number relation is,
\[
u = \sin^{-1} \left( \frac{1}{M} \right)
\]

Supersonic flows situations causes linear Mach wave and Mach lines. But in non uniform supersonic flow it is variable. Mach lines are divergent in expansions process. Creation of shocked expansion wave formation at the body surface flow is responsible for the aerodynamic force.

**Shock thin aerofoil theory:**
According to shock expansion theory, shock is a non-isentropic wave which increases entropy and across the total pressure decreases. In thin aerofoil theory shock is considered as isentropic compression wave across which pressure is less is negligibly small. $C_p$ varies with the local flow direction as:

$$C_p = \frac{2\theta}{\sqrt{M^2 - 1}}$$

Drag splits into three components:

- Drag induced at the cost lift
- Produced by chamber
- Thickness

Convective coefficient ($h$) and temperature gradients are related as:

$$h = -\frac{K}{T_{jet} - T_{wall}}$$

Aerodynamic heating produced when an object passes through fluid with very high speed and increases with speed. Design of vehicle plays a vital role in aerodynamic heating. Information regarding the heat and mass transfer can be extracted from impinging jet.

In the characterization of heat and mass transfer Nusselt number, Schmidt number and Reynolds number are used. The massive accumulation of heat and mass transfer can be minimize using the heat sinks, ablation and radiative cooling in TPS (Thermal Protection Systems). Heat sinks absorb the heat and hence lower the peak temperature. They are extra materials which increase the volume of material. In TPS carbon and ceramics can be used because of very high latent heat of fusion. Coating of these materials can protect the vehicle.

Melting process of material is termed as ablation. Radiative cooling is also employed for the same purpose. Special techniques are used in the design of spacecraft like stand-off bow shock blunt shape dissipates mostly heat to the surrounding air. As temperature increases up to 1100°C, coated layer evaporates and carry the heat with it. However spacecraft craft get heated but not harmfully. This is the reason why insulating tiles are used on the lower surface of space shuttle.

At hypersonic speed, aerodynamic heating becomes severe so nose and leading edges should be blunt otherwise aerodynamic heating can destroy the vehicle. The unfortunate example of this type is space shuttle Columbia destructed on Feb. 1, 2003 during liftoff. In many industrial applications, information regarding these transfer phenomenon is extracted from impinging jets. Massive transfer can be transferred effectively when flow is released against a
surface. Jet impingement devices provides utility of it. Various regions are crossed by jet. It behaves like a free submerged jet when it passes far from impingement surface. When flow propagates to wall in stagnation region, axial velocity is lost.

To evaluate heat transfer coefficient, Nusselt number is used.

\[ \text{Nu} = \frac{N_w}{h D_i / K_t} \]

Sherwood number is used to predict the rate of mass transfer as

\[ Sh = K_i D_i / D, \]

\[ \frac{k_i}{D_i} = D_i \left[ \left[ \frac{\partial C}{\partial n} \right] / \left[ C_{obj} - C_{wall} \right] \right] \]

where \( \frac{\partial C}{\partial n} \) provides the concentration change with normal. Spatial distribution of concentration forms similar pattern as the temperature with sufficiently low mass concentration. Heat and mass transfer can be related via equation given below

\[ \frac{N_w}{Sh} = \left[ \frac{P}{S_e} \right]^{0.4} \]

In the evaluation of fluid properties, entering point of flow in nozzle flow is assumed as the referred site. Temperature of fluid, mean velocity, viscous nature and scaling for length can be determined using position characteristics. In the impinging jet for the geometry and flow condition, it is necessary to know the nature of target and field source.

As height of nozzle decreases, Nusselt number increases so small tolerable height of nozzle is preferred by the designer. Mass, energy and momentum conservation equations are used to define the turbulent flow at low value of Mach number as:

\[ \rho \frac{\partial \bar{u}}{\partial t} + \rho \bar{u} \frac{\partial \bar{u}}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_j} + \mu \left( \frac{\partial \bar{u}}{\partial x_j} + \frac{\partial \bar{u}}{\partial x_i} \right) + \frac{\partial}{\partial x_i} \left( -\rho \bar{u}_i \bar{u}_j \right) \]

\[ \text{(Alternate form)} \]

\[ \rho C_p \frac{\partial \bar{T}}{\partial t} + \rho C_p \bar{u}_j \frac{\partial \bar{T}}{\partial x_j} = \sigma_g \frac{\partial \bar{p}}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \bar{u}}{\partial x_j} \frac{\partial \bar{u}}{\partial x_i} \right) + \frac{\partial}{\partial x_i} \left( -\rho C_p \bar{u}_i \bar{u}_j \right) + \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \frac{\partial \bar{u}_i}{\partial x_j} \]

\[ \sigma_g = \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \]

\[ \tau_g = -\rho \bar{u}_i \bar{u}_j \]

Complex interacting shock pattern and recirculation bubble are formed by the jet if flow at the nozzle is under expanded supersonic jet, in lieu of it transportation rate rises.
It is important to note that, less amount of heat is produced by the laminar flow in comparison to turbulent flow so attention is paid to maintain the boundary layer on vehicle surface. At the beginning of reentry vehicle large amount of potential energy is possessed by the vehicle due to high altitude and kinetic energy due to high speed. But as soon as vehicle lift off to earth its potential and kinetic energy becomes zero. It indicates that huge amount of energy is consumed by the body and the surrounding air. Now the question is this how large amount of energy can be transferred to the surrounding air than vehicle’s surface? It is possible with blunt nose body because it generates the stronger shock wave at the nose. Total aerodynamic heating comprises of convective and radiative heating.

Result obtained for total heat shows that:

\[ Q_{total} = \frac{1}{2} C_f \left( \frac{1}{2} mV_e^2 \right) \]

(i) Total input heat \( Q_{total} \) is directly proportional to \( \frac{1}{2} mV_e^2 \) (K.E. of vehicle)

(ii) It is also directly proportional to \( \frac{C_f}{C_D} \) where \( C_f \) and \( C_D \) represents skin friction drag and total drag respectively.

Total aerodynamic drag on a vehicle is, \( C_D + C_f \)

To minimize total aerodynamic heating in equation (17), it is necessary to minimize, \( \frac{C_f}{C_D + C_f} \)

Now consider the aerodynamic configuration as shown in fig given below,

\[ \frac{C_f}{C_D} \approx 1 \text{ for slender body} \]
\[ \frac{C_f}{C_D} << 1 \text{ for blunt body} \]

Due to inertial impaction, body force actions like gravity, electrical, magnetic forces, lift forces, particle inertia interaction ad fluid turbulence, convective mass flux comes in to picture.

Sherwood number dictates the transfer of mass for smooth surfaces with \( R_s = \frac{U_{ref} x}{\nu} \).

In aerodynamics, interaction of fluid flow with high speed object can be predicted in terms of heat and mass transfer. For TPS purpose heat sinks, ablation and radiative cooling are employed and transfer of heat and mass to the vehicle’s surface can be minimized giving the
blunt shape. Important parameters in impinging jet are nozzle, nature of target and field source. Rate of heat and mass transfer can be evaluated using non dimensional numbers.

In hypersonic and supersonic ATD, Euler and NavierStoke’s modelling has been used. CFD (Computational Fluid Dynamics) is used for the T-116, T-117, T-128 tests and Mach range from 0.8 to 25 is covered. For the better understanding of flow physics phenomena and controlled simulations, WTT is employed. Nevertheless the fundamental information is provided by the ground based facilities only single facility can not furnish all the information. A single facility can provide a piece of information and ground based facilities cannot be duplicated.

The simulations which are not possible through ground based facilities can be possible in pre-x the inflight experiment with some constraints as:

- During mission Mach range from 25 to 5 is covered.
- No oxidation is allowed
- Vehicle recovery and measures are compulsory.
- TPS expertise is necessary.
- “Vega’ and “Dnepr” are used as launch vehicle.
- Reliability of mission should be 0.95 after separating from the launcher.
- Criteria of safety should be less than $10^{-7}$.

**Moment of the aerofoil can be bifurcated in to three components as:**

1. Pitching moment
2. Rolling moment
3. Yawing moment

**Governing parameters are:**

1. Aero foil geometry
2. Angle made during attack
3. Density of flow
4. Reynolds number (viscous effects)
5. Mach number (compressibility effects)

Span wisecombination is employed in the designing of wings, tails and control surfaces. A body with large radius of curvature can minimize the gradual turning of flow. Object surface should be smooth otherwise it can perturb the boundary layer. The vehicle is designed to have good flying qualities. Thickness distribution affects the pressure distribution and boundary layer surrounding the aerofoil. Turbulent boundary layer produces more skin friction drag in
comparison to laminar boundary layer. Trailing edge also affects the location of aerodynamic center and it should be at quarter chord point in a subsonic flow.

Local regions of high heat transfer are created. It influences the flow around the vehicle and creates the problem for the aerospace vehicle designers.

**Similarity parameters for the ATD test campaign are:**

1. Dissociation parameter $\rho L$ for real gas effects
2. At stagnation point $q$ for thermal flux.
3. Mach number
4. Reynolds number for viscous effects on elevons and nose.

Above mentioned similarities are not possible to meet all requirements then it leads to simulations with CFD. Dissociation of oxygen and nitrogen molecules comes in to picture at $M > 10$. This phenomenon affects the shock wave and aerodynamics to a great extent resulting the position of pressure on the surface of body.

As height increases, velocity of chemical reactions also increases and becomes prominent at high static pressure. Non equilibrium conditions meets at high altitudes for dissociated molecules. Reduction in elevon control efficiency and increase in skin friction arises due to viscous effects at high altitudes. In the characterization of performance Mach range from 10 to 25 is preferred due to lack of data from wind tunnel.

**Hypothesis for boundary layer are:**

Turbulent boundary layer from the nose and downstream is $4 \leq M \leq 10$

1. Sudden transition at elevons hinge line or laminar regime at $M=17.75$
2. For all computations, laminar boundary layer at $M=25$.

In the supersonic region, T-116 is performed at Mach 2 and 4. In this region Reynolds number increases and to estimate this effect CFD is employed. Flow remains attached to the flap area at Mach 2 and effect of Reynolds number can be ignored.

In hypersonic flight, to verify the CFD validations and lateral behavior of vehicle T-117 is performed and 1/13.5 scale is used in this model.

Angle of attack range = $30^\circ$ to $60^\circ$

Sideslip $-10^\circ \div 10^\circ$ with flap deflection $-10^\circ \div 15^\circ$

VKI tests have been performed to identify the hypersonic aerodynamics in longitudinal, lateral directions and flap/aileron efficiency. As a perfect gas nitro gas is used and nozzle of contour is selected at $M=14$.

Factors which are greatly affected by air dissociations are:
Bow shock
Elevon efficiency
Order of pressure position

Effect of chemistry modelling on pressure distribution is shown in fig. given below:
At elevon deflection 20°, this translates into a change on pitchingmoment. Negative value of lateral control departure parameter ensures lateral/ directional stability.

Qualities:
Main objective of the flying qualities consists in:-

1. Assurance of longitudinal and lateral stability and control ability.
2. Measure the maximum sideslip and elevon deflection needed for longitudinal and lateral trim.

Flight qualities are computed with a given uncertainty-

• With worst case off nominal static margin greater than 4.5%, longitudinal dynamic short period mode is always statically stable.
• Within the required margin, lateral dutch roll dynamic oscillation remains stable.

Lateral and longitudinal trim:-

1. In the range of about ±7°, maximum longitudinal deflection needs remain.
2. Worst off nominal asymmetric deflections are ailerons efficiency < 17° for Mach = 2 and it is less than 10° for Mach > 8.

For longitudinal trim at Mach > 10 almost all available elevon deflection is used. Due to thermal constraint this value becomes greater than maximum allowed 8°. At the same time, lateral trim must be achieved. In flight dynamic analysis, focus is made on model configuration obtained from WT data.

Longitudinal trim deflection can be reduced by moving the Centre of mass forward which is also favorable for lateral directional flight qualities. The vertical position must be kept. The deflection envelope is decreased in this case. The maximum deflection constraint of 8° is not fulfilled in this case. By using a movable mass along the y axis together with sideslip and Centre of mass at $X_{CM} = 58\%$, it is possible to reach lateral trim satisfying the maximum allowable elevon deflection. Three control means elevons, RCS and movable mass are used.

To study aerofoil aerodynamic analysis, a wind tunnel is designed with four corners two diffusers and one square test chamber. To balance the pressure loss and mass flow rate in whole circuit an axial fan is used. Settling chamber and mesh screen is used to eliminate the transverse flow and turbulence respectively. In the interaction of object and surrounding fluid, crucial information is furnished by the wind tunnels. Furthermore with the advent of
wind tunnels now it became easy to measure the properties of object. While designing the wind tunnel components, special attention is paid in the chamber that it must be uniform in space.

Global dimensions of wind tunnel are mainly rely on the type of testing. After defining the chamber dimensions, other components are designed accordingly.

Wind tunnels can be classified as:

- **(OCWT)**
- **(CCWT)**

Also wind tunnel test section can be divided into

- Open WTTS
- Closed WTTS

In closed circuit wind tunnel straight path is followed by the air and no leakage takes place. It has low cost and purging of tunnel is not required in it. Open circuit wind tunnel causes environmental problems but closed loop wind tunnel is environment friendly. Open test section provides good results with CCWT.

CCWT design:

- Define shape of chamber
- Dimensions of test chamber
- Cross section of test chamber
- Fan power according to the dimensions of test chamber

The main objective of wind tunnel is to produce uniform flow within the test chamber. Stress is laid to obtain the largest possible Reynolds number which in turn depends on the type of test performed and may or may not harm the validity of result.

Main components of closed loop wind tunnel is shown in fig. given below:

instance, pressurized air, or water and properly sealed to avoid leakage.

To detach the boundary layer, a long test chamber is used and to avoid air velocity reduction, sharp edges of test chamber must be rounded. Flanges and windows are used to introduce the measuring tools and sample observations.

Design of nozzle is very difficult because flow velocity and uniformity directly depends on nozzle. Area ratio of 7 is used in the present study. Fifth order Bell-Mehta is used to define the nozzle silhouette.
Mathematically it can be represented in terms of higher order polynomial in which \(x = \frac{x}{L}\) and L is the total axial nozzle length and \(y = h\), where \(h=\)half of the cross section side length (value of \(x\) lies, \(0 \leq x \leq L\)).

Boundary conditions are used to determine the polynomial coefficients as:

\[
\begin{align*}
X = 0 & \rightarrow \quad Y = y_0 \\
X = 1 & \rightarrow \quad Y = y_1 \\
X = 1 & \rightarrow \quad \frac{dy}{dx} = 0 \\
X = 0 & \rightarrow \quad \frac{d^2y}{dx^2} = 0 \\
X = 1 & \rightarrow \quad \frac{d^2y}{dx^2} = 0
\end{align*}
\]

If the ratio is less than 0.667 then it can detach the air flow at the nozzle exit.

In the present case ratio of \(\frac{L}{2y_0}\) is set to 0.92 to obtain 1.3 length of nozzle. Sharp edges of nozzle’s outlet should be rounded off with 45° chamfers, in order to connect the testing section. Inlet cross section of second diffuser is decided by the dimensions of fan. Irregular flow is produced at the fan entrance if we choose the ratio of 3. Ratio of 2 is best suited in order to fulfill the dimensions and cost. Flanges are used to connect the second diffuser with other wind tunnel parts. Shape adapter is inserted between the fan and small corner. Outlet cross section side is determined using the equation as:

\[l_{out} = \frac{\theta r}{2} \sqrt{\pi}\]

Value obtained for \(l_{out} = 0.710\) m.

Flow is deflected four times at four corners with minimum turbulence in closed loop wind tunnel. It is the reason that corners are in pairs. Corner section width and choice of corner division helps in calculation of chord. Appropriate results can be obtained with vane number of 25.

Minimum radius of bent flat plate can be determined using equation:

\[r_c = \frac{C_{r_{1,2min}}}{2} \cdot \frac{1}{\sin^2 \frac{\theta}{2}}\]  \(9\)

where \(C_{r_{min}}\) represents chord.

\(r_c = \) radius of bent flat plate curvature.

\(\theta = \) Central angle subtended by the chord.

For cleaning and maintenance purpose a set of 24 blades are required.
In the degradation of turbulence flow before entering the nozzle, settling chamber with constant area of cross section is used. Settling chamber with length 2.070 in the present case can comprises three screens and one honeycomb.

### 4.2 Recommendations

1. In the design of an aircraft these data proves a key tool.
2. It can be used in the design and selection of diffusers.
3. Nozzle can be constructed for particular application purpose.
4. Various losses values obtained for example energy loss, pressure loss are very helpful in the analysis WT.
5. Rate of heat and mass transferred can be evaluated accurately which can further used in the flow analysis of flow field over the aircraft.
6. In the selection of shape of vehicles (whether it will be blunt or slender), it is very useful.
7. It can be very helpful in the minimization of aerodynamic heating.
8. Drag can be reduced effectively.
9. Aspect ratio and full efficiency can meet the requirements using these results.
10. Interaction of fluid flow over the body surface can be predicted.
4.3 **Future scope:**

To get rid of polluted water and air, air and water cleaners are used. Basic mechanics involves behind it is heat and mass transfer.

It also encounters in imaging techniques.

It is used in building services design because difference between emitting and receiving surface depends on the temperature of surface.

Rate of heat transfer directly depends on the fluid motion so it helps to calculate the thermal conductivity which is a scalar quantity and depends only on temperature.

Heat and mass transfer plays important role in the decay of buildings which takes place due to migration of moisture by capillary action.

Performance of newly invented devices like total heat exchanger and desiccant wheels etc. can be improved.

It helps in selection like rate of conduction is very high in crystalline form than amorphous form.
4.4 LIMITATIONS

High cost of access to space.

Dependence on geometry.

i. Potential equation is solved only in terms of cylindrical polar coordinate.

ii. Pressure coefficient is determined only for compressible flow in 3D.

iii. Study is limited up to perturbation theory to find out relation between compressible and incompressible flow.

iv. Study does not cover transformation relation for hypersonic flow.

v. Study is limited to moving disturbances only.

vi. Interactions are not analyzed in the bulk of fluid.

vii. In case of IR thermographic representation, no details are provided for unsteady state of flow. It makes temperature prediction difficult.

viii. To analyze the data of CFD and experimental investigation precisely in case of flying object, a complex pattern is obtained. Furthermore description of thermal loads makes it bothersome.

ix. During processing, signal fluctuates and can affect the reliability of result.
x. Results obtained from CFD and WTT induce a little discrepancies of aero heating predictions due to complex physical phenomena and mechanism among models.

xi. Lacking of enough flight data makes the prediction somehow difficult.

xii. A little uncertainties exit in results obtained from wind tunnel.

xiii. Local regions of high heat transfer are created which affects the quality of flow entering the vehicle and pose challenging problems for the designers of future aerospace vehicles.