Chapter 2

2.1 Review of literature

(C.ROLLE) stated that at many occasions, continuation of mass and molar analysis requires each components behavior. So relationship between mass and volume should be known. In case of incompressible component density may vary depending upon the temperature and pressure. In ideal gas model, pressure of each component varies with its mole fraction. Thermal conductivity can be varied with direction when a material crosses boundary of system, Transfer of mass takes place. Flow of fluid through a pipe, tube is an example of mass transfer.

(DIGVENDRA SINGH) studied that in fluids transfer of energy takes place through heat exchangers. Heat exchangers are condensers, evaporators, boilers, refrigerators etc. Naming of heat exchangers are given according to their use. Efficiency of heat exchangers depends on heat transfer coefficient. Area and pressure drop plays important role in heat exchangers. Least possible area and pressure drop is main characteristic of a good design.

(DIXIT, 2016) studied in aeronautical, chemical, civil, electrical engineering, important role is played by heat transfer. In every area of energy production and conversion, concept of heat transfer is used. If size of any heat transfer equipment is less, then we will maximize the heat transfer rates. In electronic industry, due to small size, heat dissipation from limited area has become critical. It is also used in designing of equipment. This thermal barrier in aerodynamics play important role.

(F.MOUKALLED) studied that CPD (Computational Fluid Dynamics) plays important role in aeronautic and aerospace industry. In electronic industry, optimization of energy can be achieved by CFD. CFD is used in HVAC (heating, ventilating and air conditioning) in building industry.

It is an engineering tool. Partial differential equation of conservation laws can be transformed into discrete algebraic equation. For each element, to compute value of dependent variable, algebraic equation of system is solved. It is easy to implement.

(GUNTER) states rheological constitutive equation relates extra stress tensor to flow field and used in liquid after deformation. Newtonian fluid and momentum equation are similar for material law. Velocity component remains non zero for parallel flow. In convective terms momentum vanishes.

(MARCELE F. PASSOS) states healthier life can be provided by biomaterial and bi fabrication techniques. But to avoid it Chemical Vapor deposition (CVD) is used. In one
single processing step adherent film over substrate can be deposited. Concept of heat and mass transfer is used in CVD reactors. Velocity and temperature is simulated in CFD study. These simulations provide a better opportunity to study heat and temperature variation over operation range.

(S.JEEVARAJ) studied that to increase surface area flat base and an array of fins are used. They are used to fulfill the purpose of heat dissipation. Greater barrier is solid air at interface. To overcome this barrier heat sinks are used. Device operating temperature is lowered by heat sinks. Entropy minimization approach is used in heat for optimization in heat sinks.

(B.SUNDEN) studied using pressure difference and mean velocity, we can determine the experimental friction factor. Pressure loss increases due to high resistance ribs and increasing shear forces. Larger enhancement in comparison to B shows that the ribs are installed with higher number per unit length. But it is not an indication of increasing performance of DPHE. It can be improved by using efficiency index. Increase in pressure drop increases the thermal enhancement.

(H.THOMAS) stated when mixture is capable of taking energy from environment, above a fixed temperature is known as auto ignition temperature. When a liquid is boiled above its atmospheric pressure, then boiling liquid expanding vapor explosion occurs and vessel ruptures. The temperature at which a liquid gives enough vapor for the formation of ignitable mixture with air.

(LI-ZHI-ZHANG) stated that the basic unit of heat and mass exchanger are ducts. To make contact between gases, solid and liquid, interface provided by ducts. To reuse waste heat and in order to increase energy utilization efficiency, New equipment’s are invented such as total heat exchanger and desiccant wheels etc. To get rid of air and water pollution air and water and water cleaners are used. The basic mechanics involves behind it is transfer of heat and mass.

(VICTOR I. TEREKHOV) stated that we can bifurcate the conversion in two categories. When fluid flows by fan, pump or external wind etc. over a solid surface is known as forced convection. When fluid flows by buoyancy forces which are produced by difference in density and change in temperature is known as natural and free convection.

(ETHIRAJAN RATHAKRISHAN) stated that many engineering application requires study of flow with velocity at which gases becomes compressible. Isentropic, Fanon, Rayleigh type flows are practically impossible. No lift is generated when a smooth symmetric body moves with zero angle of attack. Sharp trailing edge of an aerofoil produces vortex flow.
(E. L. CUSSLER) stated as laminar flow and turbulent flow have different dispersion coefficient. Dispersion coefficient is a fraction of fluids velocity for turbulent flow. For laminar flow dispersion coefficient and diffusion coefficients are equal. Coupling between the fluctuations of velocity and concentration is the main cause of dispersion. Wind gust in velocity fluctuation L abrupt change in order in concentration takes place in smoke plume.

(PRADIPTA KUMAR) stated that in imaging techniques, He-Ne laser is used as a coherent source. To record optical images of convective field, monochrome CCD camera is used. Mode switching arises due to vortex shedding and symmetrical perturbation at the Strophe frequency. From cylinder to ambient fluid, heat transfer with in online oscillation is greater due to steady plume transformation regular periodicity in intensity variation is also observed in Imaging.

(ZAHID HI KHOKHAR) stated pipeline is also a way of ejecting the heat flux. CFD provides information about flow. It is further used in equipment design and selection. In many industrial applications, computer simulation of turbulent flow is used. In design investigations, it is widely used. Asimple device for two fluid streams is a pipe tee.

(DE-YI-SHANG) For varying thermo physical properties of polyatomic gas, temperature parameter method adopted. In dimension less velocity component method, the governing equation are converted into dimension less ordinary equation. Accurate heat transfer coefficient can be determined using curve-fit formula.

(J.M.P.Q DELGADO) stated that decay in building takes place through migration of moisture by capillary action. Height and moisture depends upon the quantity of water in contact with building. This mechanism is very complex in nature. Diffusion and convection plays important role during vapor phase in liquid phase control of transfer of moisture is under effect of capillary action gravity and pressure gradient. Main mechanism of moisture fixation are hygroscopic, condensation and capillary

**Aerodynamic efficiency**;- 

Ratio of lift to drag, L/D is a good tool for the measurement of aerodynamic efficiency. Net value of drag cannot be calculated simply by the sum of drag on each component. If we try to measure separately the drag on each component then it gives an extra term named interference drag as in case of wing and body.

Total drag coefficient is calculated by the equation given below as

$$ C_D = c_d + \frac{c_l}{\pi e AR} $$

Where $C_D$ denotes the total drag coefficient of the finite wing.
\( C_D = \text{profile drag coefficient due to the separation of flow} \)

\( e = 1 \) for elliptical lift distribution

For the whole airplane configuration the above equation can be written as

\[ C_D = C_{D,e} + \frac{e^2}{\pi e AR} \]

Here \( C_D \) = total drag coefficient for the airplane

\( C_{D,e} \) = parasite drag coefficient.

As angle of attack changes due to the flow separation over the body surface and hence \( C_{D,e} \) also changes. As we know that the lift coefficient \( C_L \) is a function of angle of attack so we can consider the parasite drag coefficient as a function of \( C_L \).

So

\[ C_{D,e} = C_{D,0} + r C_L^2 \]

where \( r \) denotes the empirically determined constant.

In case of zero lift \( C_L = 0 \), \( C_{D,e} = C_{D,0} \)

Oswald efficiency factor for straight wing aircraft is given by

\[ e = 1.78(1-0.045A^{0.68})-0.64 \]

Value of \( e \) is based on actual airplanes data.

Hence total drag coefficient for the whole airplane configuration is the cornerstone for airplane design and performance prediction.

For a given Reynolds number and Mach number lift to drag ratio depends on the two factors:-

- shape of the body
- angle of attack

**W.M. Kays** In both the transfer conductance of heat and mass, three quantities involved namely conductance \((h \text{ and } g)\) and driving force \(B\) which are fluid mechanic and thermodynamic properties respectively.

In the process of diffusion there may be more than two components which depends on the molecular weight. Hence in the turbulent flow we can take ease of Fick's law.

Concentration and potential gradient affects the phenomenon of mass diffusion. Value of velocity components changes drastically within the boundary layer. Approximation in the boundary layer are made when it is negligible small in comparison to other flow parameters.

Boundary layer thickness can be estimated from the tube flow.

Flow moves inside the pipe in the same way as moves through an object. In the beginning it grows and suddenly stopped as soon as attains the radius of tube. Formation of momentum boundary layer takes place due to tangential flow of velocity at which value of velocity
change from zero to free-stream velocity. As the fluid flows over the surface, then its thermal, chemical, kinetic and potential energy also changes.

It results the change in temperature of fluid and object surface. Heat is transferred during this flow movement. Also due to concentration gradients, mass is transferred. Normal stress is created due to thermodynamic pressure. Hence the work is done and consequently value of enthalpy changes which we make use in the boundary layer approximation. Before writing the N-S equation firstly we represent the 3D flow of control volume. To find out the solution in thermal boundary layer, integral method is used. Approximate solutions are equally applicable as for momentum boundary layer. John D. Anderson

In the analysis of fluid flow, prescription of co-ordinate system is necessary. Coordinate system is chosen according to the purpose of application. To solve the fluid flow equation, different type of coordinate system are used like Cartesian, rectangular, spherical polar coordinate system all the three axis x, y and z are mutually perpendicular and represented by the unit vectors x, y and z respectively. But in many problems these coordinate system is no longer sufficient and non-orthogonal technique is adopted which requires the change while solving the problem. This type of technique are used in grid generation.

In Cartesian coordinate system to locate the position we have to fix the origin as a reference point and all measurement are done with respect to origin and r represents the position vector of a point. To localize the point in cylindrical coordinate system three coordinates r, θ and z are required.

Cartesian coordinates can be transformed into cylindrical coordinate through transformation relations. Spherical coordinate are represented by r, θ and ϕ. Those quantities which have magnitude only are known as scalar quantities. For e.g. temperature, pressure etc. and the quantities which have both magnitude as well as direction are known as vector quantities. For e.g. velocity gradient at a point gives the maximum change in magnitude and direction per unit length. Curves of constant pressure are known as isoclines. A line sketched in space at which is tangent is known as gradient line. Both gradient and isocline are perpendicular to each other in space. Rotational and irrotational field can be predicted by knowing the curl of field.

**FLUID MODEL:**

While solving the problem of fluid flow, we cannot assume it as a solid rather it is considered as a squeeze substance. So we have to model the flow so that we can apply the fundamental laws like conservation of mass, energy and second law of newton on it. We apply these laws
to obtain the mathematical equations and different techniques are taken into account to obtain
the solution of these equations. Thus obtained solution helps in the visualization of flow
physics. Three approaches are adopted in aerodynamics to obtain the suitable model for flow.
In finite control volume approach, a finite region of flow is imagined with control volume in
which streamlines.
Control Volume under investigation is represented by $V$.

Represent the flow of field

**Two cases possible:-**
1. Fluid moves in fixed volume
2. Both control volume and fluid moves

Instead of analyzing the whole flow field only a small region of flow is considered and
fundamental principles are applied on it.

**Steps in infinitesimal fluid element approach:-**
1. Assume the streamlines in general flow.
2. Imagine very small fluid element.
3. Denote the volume of element by $dv$.
4. Label it as a continuous medium
5. Velocity of fluid element is $v$.

In spite of applying the fundamental law, we can apply them over an infinitesimal fluid
element.

**Molecular approach:-**
1. Motion of fluid is due to movement of atoms, ions and molecules.
2. Flow is visualized on a microscopic level
3. Fundamental law are applied on the constituents of fluid.
4. It is somehow related to kinetic theory.

**FLOW FIELD:-**
1. Can be represented by scalar/vector fields.
2. Applicable for aerodynamic flow also
3. Analyzed in 3D space.
4. Aerodynamic parameters are written as a function of space and time coordinates.
5. Flow variables are invariant in unsteady flow.
6. Variables of flow do not change in steady flow.

**Continuity Equations:-**
When we examine the approaches of finite element volume and molecular approach and impose the conditions then continuity equations shows that

- Net mass flow is conserved
- Total energy is conserved
- Mass and energy can be transferred from one form to another form.
- Assumes the flow inviscid and unsteady.

**Force exerted by the fluid;**-

- Those forces which acts along the distance are known as body force.
- Type of force which acts on the control surface are known as surface forces.

**Strength of Vortices:**

It is a vector quantity which is double of angular velocity and depends on the velocity derivatives.

**Flow Field;**-

- Curl of non-zero velocity indicates the flow has finite value of angular velocity and flow is referred as irrotational flow
- Zero curl of velocity shows that fluid particles possess only translation motion
- Two quantities, vortices and curl could be equal if the field is purely velocity field.
- Various curved trajectory is followed by the particle of fluid in rotational flow.
- Study of irrational flow is easy and large number of aero dynamical problems are based on it. But simultaneously this flow also exit due to thin boundary layer.
- Dilation of fluid element is represented by the diagonal element of matrix.
- Strain and rotation represented by the elements which are non-diagonal.

**Difference between stream function and velocity potential;**-

- Velocity potential stream function
- Valid for irrotational flow valid for both type of flow
- 3D flow is considered applicable for 2D FLOW
- easy to solve meets some difficulty
- Velocity potential are gradient line can be sometime referred as termed as streamline  
  equipotential line.

In contradiction of it both satisfies the Laplace's equation.
Important features of aerodynamic flow can be represented in terms of mathematical equations. We write the equation for our problem and then its solution is obtained for the flow variable in term of independent variables. Then we make use of this solution for the flow over body.

**Boundary Conditions:-**

(1) Infinite boundary conditions:-

- Applicable at very large distance from the body
- Equally applicable for all directions
- Attains freestream conditions
- Represented by using the subscript on that parameter

(2) Wall boundary conditions:-

- Surface should be solid
- On the surface velocity should be zero for viscous flow penetration is not allowed.
- Wall tangency condition should be fulfilled
- Velocity should be finite for inviscid flow.

For an incompressible flow, velocity and pressure can be calculated through Laplace's and Bernoulli's equation.

<table>
<thead>
<tr>
<th>Source flow</th>
<th>Sink flow</th>
</tr>
</thead>
<tbody>
<tr>
<td>Streamlines tends to away</td>
<td>streamline meet</td>
</tr>
<tr>
<td>from origin</td>
<td>at origin</td>
</tr>
<tr>
<td>Relation between distance</td>
<td>velocity is inversely</td>
</tr>
<tr>
<td>and velocity is not inverse</td>
<td>proportional to the distance</td>
</tr>
<tr>
<td>Lines are not radial</td>
<td>Curved line obtained</td>
</tr>
<tr>
<td>Except at origin, divergence</td>
<td>$\nabla \cdot \mathbf{V} = 0$</td>
</tr>
</tbody>
</table>

For transport phenomena, Bryon Bird et.al 2002

As the fluid passes over the object three quantities are transferred namely

- Mass (due to concentration gradients)
- Momentum
- Energy
- Angular momentum
When we study the transport phenomena then both microscopic and macroscopic levels are taken into account. On the macroscopic level, balances are used which examine the quantities entering and leaving the flow field. It is understood that these are conserved quantities but their values change across boundary layer. To evaluate this change we examine the flow at microscopic level. Equation of change dictate the heat parameters and quantities involved in it and can obtain the various profile at different conditions.

According to Newton's law of viscosity, shearing force in a fluid is directly proportional to the negative of velocity gradients. On the basis of it, fluid can be categorized as:

- Newtonian fluids
- Non-Newtonian fluids

Principle of corresponding state becomes a powerful tool when experimental data is inadequate. This approach helps to correlate the data to state of equation. As we increase the temperature, gas viscosity rises.

**Assumptions of molecular theory of gases:-**

- Molecules of gas are perfectly rigid, non-attracting and spherical in shape.
- Number of particles are small
- Equilibrium state is considered

Results obtained from this theory gives best results for monoatomic and polyatomic gases. It is due to the reason that center of mass coordinate plays important role rather than other coordinates.

In the laminar flow, momentum balance can be used for single component of velocity.

**On the basis of experimental data obtained three regions of flow can be defined as:**

- Less ripples in laminar flow \( \text{Re} < 25 \)
- More ripples in laminar flow \( 25 < \text{Re} < 1450 \)
- Turbulent \( \text{Re} > 1450 \)

These results are valid for mediocre length of plate but on the plate surface ripples exists at all Reynolds number and vary with it. This dimensionless number disturbs the flow uniformity.

The quantity which is not conserved across the boundary layer is mechanical energy. This energy is converted into heat energy which in turn produces the large velocity gradients and viscosity gradients. Equation of charge can be written from the equation of continuity also. In case of very slow flow rate, quadratic term of velocity can be ignored.
In this situation, Stokes equation is known as creeping equation. Equation of motion transformed into Euler equation when a blind eye is given to viscous forces. To get the similarly of a given problem, geometric and dynamic similarity are necessary. In other words we can say the dimensionless numbers, ratio of dimensions and boundary conditions imposed on system should be same. Scale factor can be obtained through characteristic value of parameters.

Von Karman vortex formation begins when fluid flows over a cylindrical surface flow separates from the body at separation point and flow becomes unsteady.

Sometime in fluid flow problems, velocity depends on more than one coordinate then partial differential equation are written and then transformed into ordinary differential equations because their solution is easy to obtain.

Three main methods used are:-

- Combination of variables
- Separation of variables
- Sinusoidal response

While solving the problem, we encounters the ratio of which is known as dimensionless error function. It is denoted by erfcn. Complementory error function is denoted by erfcn.

Profile of velocity distribution can be obtained from vortices equation.

Jets and wake can be considered as the free turbulence. Fluctuations in the velocity becomes larger near the wall surface. Isotropic turbulence is found at the center of duct. At the center equal amplitude is retained by the fluctuations. Turbulence viscosity is strongly affected by the turbulence position and intensity. Wall turbulence meets at the interface of fluid and solid.

In the turbulent flow, mixing length of Prandtl gives the appropriate expression for momentum transfer.

**Characteristics of Newtonian field:-**

- Development of secondary flow
- Flow moves on the cylindrical surface
- Used in reducing the drag
- Friction can be minimized upto 45%
- Enhances the rate of flow of water
- State of caloric equation
Elastic nature of fluid can be visualized through small amplitude oscillatory motion. Heat is always transferred from hot body to cold body. In the isotropic medium, conduction of heat is given by Fourier's law.

When we solve the partial differential equations then we found some constants of integrations. These constant are evaluated by imposing the boundary conditions.

**Boundary conditions include:-**

- On the surface, distribution of temp.
- Rate of change of heat per unit time over the surface
- Heat flux and temperature at the interface

Coefficient of heat transfer can be calculated using the Newton's law of cooling. Efficiency of cooling fins can be estimated through conduction of heat. The mechanism of heat transfer in cooling fins is transfer of heat between metal and fluids. Complexity of governing equations can be avoided using the dimensionless numbers. In the phenomenon of free convection, level of fluid increases due to decrease of density with temperature. In the transformation of equation of state buoyant force comes into picture and simple equations are used.

In the steady state problems we make assumptions for the simplification of problem.

- No change in physical properties
- Net flux is zero

**Importance of dimensionless numbers:-**

- Lower value of Reynolds number indicates that internal forces are small in comparison to viscous forces.
- Lower Brinkman number means heat is transferred through conduction

In this way can say that dimensional analysis makes the problem easier.

**Viscous fluid flow, Frank M.White**

Wakes are jet like structures and represent the defect in uniform flow field. They are regions of constant pressure and much stronger in comparison to jet.

Depend on amount defect, wake can be classified as

- Circular wake
- Plane wake

According to momentum theorem, drag force does not depends on the coordinate x.

**Hypersonic interaction parameter:-**

This parameter defines the strength of interaction over the surface of leading edge.
• If its value is smaller than one, it indicates the negligible interaction
• If value approaches to one, it shows the weak interactions
• If value is larger than one then it represents the strong interaction.

Equation of state do not remains same in turbulent flow rather get modified.

Level of fluctuations rises in case of turbulent flow and it is represented by placing the bar 
over the time averaged value. Semiempirical models are applicable to compressible turbulent 
flow. Models adopted provides the quantitative explanation but not in agreement in case of 
wall temperature. Hence velocity at wall surface can be calculated as:
  • Correlating the profile of velocity
  • Using the velocity data of Van Dries
  • Considering the viscosity of eddies

Experimental uncertainty exists in the data obtained in laminar and turbulent flow. Transition 
strongly depends on two factors for given Reynolds number:
Type of disturbances
Mach number

**In the boundary layer pressure gradient can be evaluated using**

Integral approach of Karman
Transformations of compressibility
Integral approach for an inner variable
Approach of finite difference

**Instability criterion of Rayleigh:**

An inviscid flow can be assumed unstable if its circulations decrease as it moves away from 
the surface.

• Unstable stateis obtained when outer cylinder comes to rest while inner surface rotates.

<table>
<thead>
<tr>
<th>Rest</th>
<th>Motion</th>
<th>Flow</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outer cylinder</td>
<td>Inner cylinder</td>
<td>Unstable</td>
</tr>
<tr>
<td>Inner cylinder</td>
<td>Outer cylinder</td>
<td>Stable</td>
</tr>
</tbody>
</table>

If the direction rotation of inner and outer cylinder are opposite then it yields unstable flow

Gradient of convective acceleration gives linear and non-linear solution depends on the zero 
and non-zero value.

Depends on the geometry following type of flow obtained:
Steady flow unsteady flow
Wall duct flow
Driven flow                               flow of
Poiseuille or pressure                moving boundaries

Besides these flow, suction and injection duct flow, Ekman flow and stagnation flow also obtained.

According to Couette, When flow passes over an cylinder then viscosity can be calculated through Mach number and geometry.

At the microscopic level, parameters of heat are defined at a point. These parameters are sometimes represented by the lines and isocline are obtained when along the orbit value of scalar remains constant.

As we know that fluid flow depends on space time coordinates and accordingly described as steady and unsteady flow. But in many practical problems for e.g. in aerodynamics we need to convert the steady state parameters into unsteady state. It is possible if we make use of transformation equation which relates the coordinates of one reference frame to another and vice versa. To understand it let us suppose that in a stationary frame of reference an airplane flies with constant velocity. We observe that flow is not steady at a given point. Now we examine the situation in a moving frame of reference then steady flow is observed. Here we can relate the coordinate of unsteady and steady flow. This approach is widely adopted in the wind tunnels. Data obtained for a static model can be equally applied for the moving model. Many times only little discrepancy exists in the results.

Transfer of mass over a control volume can be calculated through a quantity mass efflux.

1. Negative value of mass efflux represent the mass is leaving the control volume
2. Total value of mass efflux represents the mass is entering the control volume.
3. Zero value represents the mass is neither entering no leaving.

**Pump:-** It is a device which supplies energy to fluid particles.

**Turbine:-** It is rotating device which is used to extract the energy from the surrounding fluid.

To evaluate the performance of turbine, moment of momentum approach is used.

According to Newton's viscosity statement:-

it is ability of fluid to resist the deformations produced by shear forces. In case of solid objects, modulus of elasticity opposes the deformations of fluid. Elastic and plastic behavior can be fully explained on the basis of shear strain. No-slip condition meets when fluid layer moves with some finite velocity. It shows that fluid is viscous. In case of inviscid flow, this type of no-slip condition can be ignored e.g. of viscous fluids are tar and molasses.
The phenomenon of viscosity of liquid can be defined on the basis of molecular motion in the same way as defined for the gases. As the molecules of gas moves in random direction and transfer energy. In the same way liquid molecules moves through boundary layer and during this transfer of mass and energy takes place. Viscosity is directly proportional to the square root of temperature.

**Units of parameters involved in viscosity**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Viscosity</td>
<td>Pascal-second</td>
</tr>
<tr>
<td>Temperature</td>
<td>kelvin</td>
</tr>
<tr>
<td>Boltzmann constant</td>
<td>ergs/k</td>
</tr>
<tr>
<td>Collision diameter</td>
<td>angstrom</td>
</tr>
</tbody>
</table>

Shear stress is responsible for the deformation and it has direction, magnitude and orientation w.r.t. plane, so referred as a tensor quantity using double subscript.

When the flow moves around a central point by making the circular pattern, termed as vortex flow e.g. whirlpool.

**Sinusoidal ducts:-**

It can be constructed using corrugated tins and plate. To obtain the flow parameter in these ducts, temperature across the wall is assumed uniform. These ducts are used to transfer the heat and mass. In the sinusoidal ducts, heat and mass transfer of one is accompanied by the another. Governing flow field equations can be simultaneously for these entities. Numerical approach is used to solve the respective equation for heat and mass. Total mass leaving and entering over the surface is not constant hence its solution also provides the inhomogeneous distribution of temperature and concentration.

In all type of ducts, wall is not a solid metallic surface whereas possibility of functional material is also there in place of metals which can involves the process like absorption, desorption etc. Rate of heat and mass transfer is influenced by the interfacial properties. Biot number depends on thickness of solid wall thermal conductivity and coefficients of heat and mass transfer. It can also be represented in terms of Nusselt and Sherwood numbers. In case of heat transfer it gives small value whereas large value is obtained for mass transfer. Resistance inside the wall can be neglected in case of very small Biot number. Honeycomb structures are made of porous fiber glass and efficient to absorb the large amount of heat transferred.

**Comparison of honeycomb and packed beds:-**
1. Fluids does not move in the walls of absorbent while packed pallets are less dense
2. Honeycomb beds are very sensitive to external conditions.
3. Exchange of heat and mass transfer takes places with walls.

**Honeycomb:-**

1. Used in the dehumidification of air
2. In the recovery of enthalpy
3. Examples of honeycomb are desiccant and energy wheels

**Louis Cattafesta**

In fluid mechanics, testing of component is done using the wind tunnels. It also opens a new door for the development of research for future space-vehicles. As a matter of fact that, when CFD and theoretical techniques does not survive then we have only one option none other than wind tunnel-Both full and semi model scale can be employed in the testing process. It saves time, as well as cost. In order to pursue the parameter matching of a flight generally full scale testing is adopted.

Now a days wind tunnels are widely used in many areas of research. We can classify the wind tunnels in a number of ways. In the fluid physics, we solve the flow pattern around an object. Then we model the particular problem and write the governing equations in terms of dimensionless numbers.

For e.g. while writing the governing equation for energy, Eckert number appears in it. Boundary conditions are responsible for the appearance of these parameters. Classification of wind tunnel can be done on the basis of flow inside the nozzle and relative value of dimensionless quantities. Compressible and incompressible flow can be defined by Reynolds and Mach numbers. Froude number is used to define the flow of liquid. In the wind tunnel testing, critical value of all these parameters must be matched. Moreover size can be taken as a tool in the bifurcation of wind tunnel. It the dimensions of test section is less than unity and Mach number is less than 0.3 then it is described as small scale wind tunnel.

In case of large wind tunnels, dimensions of test section also vary and used for the model testing of spacecraft components. As the power consumption is very high in case of supersonic /transonic hence small wind tunnels are frequently used. Based on the application purpose different type of wind tunnels are used for e.g. ground planes, icing, climate smoke propulsion spin stability etc. In the testing of models, dynamic stability achieved using the scaled moment and force data.

Reynolds number is used as a key factor in the low speed wind tunnels. Attaining the similarity of Reynolds number in wind tunnel is very problematic mainly when the flow is
incompressible. In order to simulate the Reynolds number one have to comprise. On maximum speed of tunnel, a limit is placed, which decides the validity of experiment performed.

In the test section of wind tunnel fluid must flow smoothly. For this a system with tunnel drive is used which depends on the type of fluid. To supply the air in the air wind tunnel compressor and axial fans are used in the test section. In the drive system if we use compressor then it gives high performance and yields little cost. This type of systems cannot work continuously. As an alternate, in the drive system fan is used. At low speed it gives good results. If the working fluid is not giving satisfactory results then closed loop wind tunnel should be used. By doing so leakage through it can be avoided. Secondly a gas at high pressure can also be used. It changes the density and it is possible to achieve the similarity conditions. If we cool down the air, then properties of flow also changes.

At this stage, cryogenic engine can be used. In open circuit wind tunnel any amount of air can be used which enters at one end of test section. But in closed loop wind tunnel air passes through different component but amount remains fixed. These is so many advantages of open circuit because in every cycle fresh air is used but requires high energy in operating. There should be no obstacles on the inlet and outlet section of wind tunnel.

Closed tunnel highly depends on the conditions of weather. Although its cost of construction is high yet cost of operation is low. In both type of wind tunnel the major difference lies in flow regime. Supersonic and hypersonic facilities are provided by a nozzle which should be converging and diverging. It is designed in such a way that generation of shock wave can be reduced. To achieve Reynolds similarity in transonic facility scale of testing model is increased. Similarity of Mach number is not necessary in case of incompressible flow. While designing the wind tunnel, three points should be taken into consideration as:

- Model under testing
- Goal of research
- Cost

A wind tunnel under investigation must provide the appropriate data. In the test section, rate of turbulence generation should be low. Uniformity of flow should also be maintained in the test section. If separation of flow takes place in the test section then noise level and losses can also increase. Free stream condition in the test section should also avoided.

In the Designing procedure of low speed wind tunnel, firstly a target is inserted in the test section with suitable Reynolds number of flow.
Size of inlet duct is determined after deciding the size of inlet. Uniformity of flow is influenced by the honeycomb. Recovery of pressure is increased upto a great extent using the large diffuser. Flow can forced to move along the axis of tunnel by making use of honeycomb.

Turbulences present in the flow can be removed using these honeycomb structures. In the cells of honeycomb stalling can be avoided by making yaw angle less than ten degree. There are different shapes of honeycomb structure but the most preferred shape is hexagonal because it provides very low value of coefficient of pressure. Performance of honeycomb cell can be improved using the appropriate length to diameter ratio. It should be rigid enough so that no deformation takes place during operation.

Turbulence of test section can be minimized by placing the screens of increasing porosity in the settling chamber.

Quality of flow can be analyzed using the inlet contraction. Intensity of turbulence is largely depends on the inlet contraction. To reduce the thickness of boundary layer, its length must be small. In this way cost can also decrease. Separation of flow is caused by the large gradients of pressure.

Test section of wind tunnel permits to enter the model. In closed test section performance is best suited with full scale. Blockage ratio chord are the main factors which decides the type of test section. From the test section, flow can be decelerated with the help of diffuser. Flow inside the diffuser is also affected by the test section flow. To ensure that the flow is not separating, diffuser area must be varied with axis. In order to get optimization a special attention is paid on the angle. Pressure loss compensation is obtained at the expense of rate of volume flow.

Load curves of fan gives the performance of fan. These curves are drawn from the rotational speed of fan. Operating point of wind tunnel is the intersection point of pressure loss and performance of tunnel. Operating points of wind tunnel lies closer to the fan maximum efficiency for the optimal performance. From the drive system of fan, noise is created. Jet is captured by the collector in an open jet test section .Configuration of test influence the flow pattern. Selection of collector is made through computational fluid dynamics .Adverse effects are avoided by inserting the turning vanes. Cooling can be provided by the turning vanes.

In the test section velocity can be measured using the pitot tube. In the test section turbulences are generated and then they interact with the boundary layer and produces the additional noise. Hot wire is used to visualize the turbulence of test section. To detect, this hot wire moves in the test section and the bulgy signal of noise are detected at some points.
Component of velocity can be different in lateral and axial directions. In this case hot wire probe is initially placed along the test section axis and then rotated to make perpendicular the axis. If the fluctuations in both directions are measured simultaneously then dual wire is used. To measure the components along all the three directions probe of triple wire is used. Whole space and time data can be collected using the series of hot wire probes. High pass filter is used to measure the intensity of turbulence. Microphone is used to detect the noise level in test section. Accelerometer is used to measure the vibrations.

To access the performance of full span scale model, a high speed tunnel test was conducted for a cruise with $M=0.85$. In this test different parameters were quantified. Many challenges are posed in the operation and design of aircraft. It occurs because optimum value for the flight regime cannot be attained easily due to the increase in Mach number. In transonic regime, formation of shock wave becomes very strong and affects the aerodynamic efficiency. In this test multiple configuration of a cruise are also assessed. 33m long aircraft is tested for a height 13 km and range from eleven thousand to thirteen thousand kilometer. Slender shaped aircraft is affected by aeroelastic effects. Winglets are neglected in the datasheet and separation of shock wave is avoided by adopting the pylons. In this model many features like pressure, moment, altitude etc. are measured simultaneously.

Six span wise stations are taken in the evaluation of distribution of load at wing. Construction is affected by these taps so stress is laid to minimize the pressure. While fixing the central balance house and wings tube and cable routing is analyzed critically. Alloy of aluminum is used to construct the parts of model. In the analysis of finite elements CFD is used. In HST pressurized air is used as a fluid in the test section. In test section one variable is kept fixed at a time and another is varied for e.g. if Reynolds number is constant then Mach number will be variable. Range of Mach number lies between 0.2 and 1.3. Axial compressor is used to supply the air. Dimensions of solid side walls, rectangular shaped test section are $2m \times 2m$. Testing model in the test section is supported by the vertical strut and ventilation is provided by the plenum chamber, Initially z- sting is used because of its flexible nature.

In the second step, internal balance is used. Set up made in the test section affects the aerodynamic so two measurements are made to detect it. Model is sealed downstream to avoid the internal flow. External effect on the model is evaluated using with and without the dummy support. To avoid the interaction of support system dorsal sting is used. Q – flex inclinometers and gauge balance is used to evaluate the orientation of pitch, roll and force, moment respectively. Inclinometer output is compensated by the vibrations. When the wings
of aircraft are deformed then SPR technique is employed. It sketches the images. Wings are coated to insulate thermally and three lines are drawn on the upper surface.

Data is collected at different angle of attack by maintaining the constant Reynolds number. Self-adhesive dots are marked on chord. Wing, upper and lower surfaces. In the downstream, pylons and nacelle are also marked. Some corrections like blockage buoyancy and z support are made in the database. In contrary to it, up wash correction is not made. Distribution of static pressure provides the favorable flow conditions. Change in coefficients is calculated by making use of with and without support.

In 3D space, pixels are fixed by the stereo images with wind on and wind off situation. In this way we can predict that up to which extent model is deformed. A pair of markers is used to locate the deformed regions and specially highlighted in the reconstruction of model.

Dots on the effective region are marked by the marker and to detect its influence IRT is employed. Temperature steps are clearly visible which indicates the change in coefficient of heat transfer. In the creation of temperature steps shock waves are behind it.

Dots represents the areas where the rate of heat transfer is very high. It is the reason that in case of wedge shapes they are invisible. Repetition of data point are calculated using the linear interpolation. With the increase in angle of attack, level of interferences also increases.

At the tip of aircraft deformation rate is very high and greatly depends on the angle of attack. Vibrations produced in the test section causes loss in lift coefficient. Losses deformation and differences are analyzed through computational fluid dynamics.

M.Di. Clemente This paper presents a detailed description of shock waves in the wind tunnel. How shock waves interacts with boundary layer is explained in a lucid way. On the control surface of aircraft flow detached and results loss in efficiency of aircraft. In experiment different flow conditions are provided and the obtained results are implanted in the construction and design of spacecraft. In hypersonic flow gas is no longer perfect with increase in temperature.

Gas molecules gain energy from temperature and starts vibrating. Ultimately dissociation of molecules takes place. As a result condition of thermochemical non-equilibrium is set up. At this stage value of temperature and pressure changes in a drastic way and extra terms are added in the Navier- Stokes equations. These equations can be solved using the finite volume / element technique. Central difference technique is adopted to consider the viscous effects.

In plasma wind tunnel, we can test the hypersonic jet and simulate the flight during re-entry. Performance of aircraft is calculated in terms of total pressure and enthalpy. In the test section of wind tunnel air is used and to expand the compressed air electric power of 70 mega
watt is supplied. Conical shaped convergent- divergent nozzle is used to expand the air and establish the desired flow conditions. Value of Mach number can vary up to eleven. In the beginning, pressure expands at stagnation point and upto flat plate quasi zone is obtained. Value of pressure change quickly at the locations where the flow separates and then attains a flat value. Intensity of heat decreases in the recirculation region.

Temperature of wall cannot be measured in non-equilibrium conditions. Test conditions obtained from the plasma wind tunnel can be duplicated for the flap. Size of nose can be decreased by employing the advanced cooling techniques. Which in turn causes the decrease in separation zone.

In the under investigation configuration, 0.1 meter nose radius is best suited. If we take the lower value of nose radius then it makes difficult to find the exact value and large values are difficult to handle in the test section. Localized heating can be minimized using the round edges. Model under investigation is made of steel and coated by thermally insulating material. Hence interaction of model with the fluid can be avoided. No peak heating occurs due to cylindrical shaped covering. At stagnation point result obtained from the computation and referred estimate are not same. A little difference is found in the values.

Peak value of different parameters can be evaluated through curvefit. Along the trajectory two points p and q are located which indicates the maximum value of heat flux at stagnation point and point of highest height respectively. Contours on the flap are predicted in equilibrium condition. Components of model can be easily replaced. Thermal stress can be reduced by proper selection of flat plate- and integrity of panel can remain intact. Copper is used in the inner part of leading edge. A cooling jacket is inserted between inner and outer part of leading edge.

Commercial software package ICEMCFD is used to generate grids. Simulations in two dimension does not affect the results obtained from model.

Topology:- Model- O-grid
Flap- quarter o-grid

Grids are generated only for a half model to pressure the symmetry
Level – Finest grid
Approach used – block structure
Blocks -62
Cells -1.5

Calculated results are verified using the medium grid level. Catalytic effect can be neglected on the flap surface.
Boundary layer is a region of variable velocity and its formation takes place when the fluid passes over the surface of object. Thickness of boundary layer is very small in comparison to the body thickness. Concept of boundary layer was given by the Prandtl and his scholars. Thickness of boundary layer directly depends on the boundary layer. Thicker boundary layer is obtained in case of low velocity.

Factors affecting the Reynolds number:
- Amount of turbulence in fluid flow
- Leading edge shape
- Wall roughness
- Vibrations

Transition in boundary layer takes place at high Reynolds number. Laminar flow is resulted by the flat plate surface in the flow field. In place of flat plate, if curved surface is taken then turbulence starts earlier. Heat transfer rate depends on the thin thermal sublayer. It forms on the wall surface quickly. Both velocity and temperature profile can be measured at inlet. Shape of inlet can alter the entry length.

Fluid flow depends on the pipe roughness. It is sufficient to destroy the thin thermal sublayer. In the heat transfer rate friction factor plays an important role. Tubing and periodic ribs are used by the manufacturers deliberately in order to reduce the surface area. Impurities present in the liquid tends to decrease the Nusselt number.

Terms and conditions of flow for circular ducts cannot be applied on rectangular ducts or irregular cross-sections and hence value of Nusselt number also changes accordingly. Flow of heat transfer in the enclosure serves the purpose of energy conservation and can be used in large number of applications.

To calculate the skin temperature of rocket, an exact solution of temperature distribution is obtained through complex calculation. Surface area of rocket and heat capacity determines the skin temperature we can neglect the lateral component of temperature due to its small contribution.

The term stagnation defines the isentropically rest position and accordingly we can define stagnation temperature, pressure etc. This value is obtained at the cost of skin friction and always greater than that of boundary layer. To obtain the full scale temperature profile, firstly skin temperature is evaluated for each station and then combined using the approach of conservation.

As the value of different parameters changes with altitude so it makes extrapolation difficult. Inside the skin temperature can be assumed uniform due to instant heat conduction. For a
perfect black body, value of emissivity is one. Exact value of skin for real rocket is unknown. Solid materials conduct more in comparison to gases. We can assume the value of shortening factor for the materials like aluminum. But in case of composite materials its value comes out less than one.

Reason behind is ablation of material. In this way validity is also affected. Basic method cannot be applied for all values. Extreme lower and higher value at station creates critical situation. Actual flow situation helps to calculi K value. Recovery factor must be zero in case of zero cone angle. At the highest point of nose value of recovery factor is one.

Temperature and pressure gradients forms due to boundary layer. K depends on Reynolds, Mach and Prandtl number. Decrease in recovery factor shows the decrease in dimensionless numbers. Laminar and turbulent flow is also classified on the basis of Reynolds number.

**Heat transfer coefficient depends on**

- Temperature of boundary layer
- Emissivity

Stagnation temperature is attained by the surface of rocket if there is no loss due to radiation. Hence upper boundary is decided by the stagnation temperature. Energy is transferred between the flow and surface of rocket so to attain the steady state is quite difficult. In the model we assumes that boundary layer is not affected by during ablation and it occurs at known values. In the present study skin temperature is less than the temperature of ablation otherwise the whole material could be evaporated.

**DiranBasmadjian** Increase in fluid velocity allows the more turbulence around the boundary layer. Boundary layer get thicken due to penetration of eddies and rate of mass transfer increases suddenly. According to film theory, in the turbulent zone, fluctuations can be minimized through profile of linear concentration. In the field of flow it is difficult to localize the upper boundary.

Coefficient of mass transfer for a plane surface varies inversely to Schmidt number. In case of gases value of this number is equal to one. Mass balances are used according to application purpose for example it can either be used for finite space or element of incremental space. Mass balance used for finite size are termed as macroscopic balance. In this we make use of algebraic and ordinary differential equations.

Impurities of gas can be removed by adding solvent and the device is known as gas scrubbers. After adding the solvent, impurities get settle down and pure gas at the top of column. Liquid mixtures can be separated and purified using Batch distillation. During the process of diffusion, concentration varies with space and time. If the number of variables are
more than one then one should employ the method of partial derivative. Solution obtained from Laplace’s equation are converted into rate equations in order to make use in practical problems.

**Keith J. Moss** Fan and pump can be used to obtain the forced convection in finned tube. When sun radiation passes through atmosphere, obstructed by the particles suspended in the path and ozone layer. During this obstruction intensity is reduced and radiation get filtered. These obstruction affects the receiving surface and decreases the rate of heat transfer. Heat can be transferred only to the visible surfaces.

If the surface is covered from the dust particles or by some other means then heat flux will be very small.

Heating devices can provide the heat radiations from shorter to longer wavelengths which lies in between UV to IR region of spectrum for example electric quartz heater etc. Pattern followed by the heat radiations resembles with the light waves. Heat transfer depends on both receiving and emitter surface area.

Surface get heated only when it absorb the radiation. For e.g. if a surface reflects ninety five percent of heat received then it will not heated up too much. Maximum heat can be transferred if we place the object normal to the surface which emits radiation. Lambert’s law is applicable for surface element because it emits unequal heat flux in space. Heat radiations emitted by the line source are confined to only one plane.

**Sangho Choi** In this paper, it has been shown that how heat and mass transfers on the cryogenic surface. Due to the temperature difference between wall and cryogenic surface, frosting phenomenon takes place. At very low temperature water converts into ice and a layer grows on the cryogenic surface. During this process transfer of heat–mass takes place between wall and cryogenic surface. This concept is used in propulsion system of rocket. In cryogenic systems liquid oxygen is used and this oxygen get evaporated and stratified due to penetration of heat flux. Hence reliable results are obtained through rockets (oxygen loading). Experimental results obtained from uninsulated cryogenic surfaces are different from that of theoretical results.

Sixty minutes are taken during initial wall cooling. As the time passes, frost starts to collect on the cryogenic surface and plays a role of insulator. At this stage exact value of heat transfer can be calculated.

**Purpose of experiment:**

- Account the effect of initial wall cooling
• Validate the result under different conditions

In the experimental chamber ambient conditions for air, temperature and humidity is provided. Condition of natural convection is maintained through air velocity of 0.1 meter per second in the test section. Constant flow rate is established through a constant mass flow rate. Connections between tank and support are made of carbon so that no transfer of heat takes place. Liquid nitrogen is levelled through level gauge.

**Frost measurement:**

• Assumption of axisymmetric mass distribution is made

• Test section frost is measured through balance method.

Frost forms on the tank after filling the liquid nitrogen. Initially no frost is on cryogenic surface. Interval of frost mass and wall temperature measurement is 30 minute and 60 minute respectively.

Formation of frost depends on the temperature of wall and freezing point. In the normal direction of cryogenic surface, layer get thicken. A quasi – steady and one-dimensional transfer is obtained in the natural convective condition. There is no variations in the density and conductivity of frost which grows over the cryogenic surface. In the frost layer heat is not transferred through radiation and saturation of water vapor is obtained.

**Condition of heat transfer:**

Wall temperature > Dew point

**Condition of simultaneous heat transfer:**

Dew point > Wall temperature > Freezing point

Analysis of this process is quite complex because the phenomenon takes place inside and outside of tank. Energy is transferred during the phase change and conduction. As we assumes the uniform density distribution of frost so at all points of layer amount of frozen vapor will be same. Boundary conditions help us to determine the wall heat flux and distribution of temperature.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Test number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial wall cooling</td>
<td>1 and 2</td>
</tr>
<tr>
<td>Air temperature</td>
<td>2, 3 and 4</td>
</tr>
<tr>
<td>Relative humidity</td>
<td>2, 5 and 6</td>
</tr>
</tbody>
</table>

**Measurements are done:**

• When gas is filled in the tank
Insulator is removed
Frost starts to grow
Steady state is attained

Frost thickness on the cryogenic surface in natural convective condition is 3mm. Resistance of frost layer increases with increase in growth of frost. Cryogenic surface temperature is strongly affected by the thermal resistance and lowering of wall temperature. Maximum heat is transferred when temperature of surface is minimum. It obeys the fundamental principle of heat transfer. Frost density increases with increase in water vapor. Consequently thickness decreases with increase in temperature. Mass transfer rate directly depends on the driving potential. Numerical results are in good agreement with experimental data.

Y.D.Tu, R.Z.Wang

Desiccant coated water-sorbing heat exchanger are used in the dehumidification of air. Desiccant absorbs the heat produced during cooling and drying process.

**Key factors of WSHE:**

- Operating temperature is very low
- Increases the capacity of uptake water
- Air can be condensed
- Requirement of air get fulfilled
- Efficient in dehumidification

In the continuous operation of heat and mass transfer two water sorbing heat exchanger are required. In the operation mode switch deep analysis of dynamic behavior is needed. In WSHE, we imagine the uniform and constant temperature distribution. It is one dimensional system and a model is accepted to represent the transport process.

In order to get the reliability, experiment is performed many times and a steady state is achieved in every test. Both cooling and heating modes are achieved during the performance. We use water as a transferring medium. In the expression of temperature higher terms can be neglected due to their small contribution.

**Remarks:**

- Different conditions gives the same result
- A joint process of cooling and heating is obeyed
- During the performance, temperature falls and rises
- Fluctuations vary with time
- Difference in concentration gradients on the surface of fin promotes the rate of mass transfer
M.J. GibbonsHeat transfer plays an important role in electronic devices. Due to advancement in technology, size of electronic components are decreasing day by day and to manage the thermal dissipation has become a challenging task. Electrospray cooling is taken in to account for the validation and reliability of a device.

**Single cone jet electrosprays:**

- Energy is supplied through Coulomb forces
- Electric field created between target and nozzle
- Shape of liquid cone is affected by the large electric field on the tip surface
- Droplets are produced by the jet
- In the electrospray cooling, voltage depends on many parameters like properties of fluid etc.
- Value of applied voltage is not fix rather can vary from one situation to another
- Transfer of heat and droplet size depends on the conditions imposed on EC
- A series of multiple source is needed to increase the rate of heat transfer
- As the temperature of test section increases, rate of evaporation increases
- To calculate the heat transfer rate, Nusselt number is used and results obtained from experiments shows harmonics with ten percent deviation.
- Stainless steel is used in the manufacturing of thermal exchange section.
- Temperature of cavity is measured using the thermocouple
- To increase the emissive rate, inner layer is coated with black paint
- Roughness of steel and thickness of painted layer is decided by the profile data
- Electrospray cooling is performed at room temperature and atmospheric pressure. In the course of time it attains a steady state
- To detect and resolve the images of IR camera, computers re used
- Bi number is very low
- Jet formation takes place on the tip of cone which in turn depends on the fluid parameters
- All the three phases of matter intersect at triple line and two type of cooling evaporative and electrospray is obtained
- When impinging droplets do not supply the sufficient energy, then pool cooling takes place
- Spreading and thickness directly depends on the amount of mass flux
- Evaporative and electrospray cooling depends on the fluid, applying conditions, triple line and independent of the size of nozzle.
In this paper, four different models are discussed to evaluate the chemical kinetic rates over the hypersonic vehicle. When the spacecraft moves in atmosphere at very high Mach number, then flow around it affects its performance. So, to access the performance at this stage, a model is proposed which also takes into account the chemical effects. Wave pattern around the vehicle is quite complex and huge amount of heat is transferred during the formation of these patterns. As the vehicle is moving so field get separated out.

Following phenomenon happens in air:

- Atmospheric air (gas) becomes chemically reactive due to ions because molecule dissociates.
- Gas molecules start vibrating.
- Atoms / molecules of gas goes in to higher states by absorbing this tremendous heat energy.
- Dissociation causes formation of ions.
- Efficiency of hypersonic vehicle get reduced.

Thermochemical non-equilibrium set up in the flow field around the vehicle due to same time scale of both field parameters and chemical reactions. At Mach 13, when ELECTRE flies then severe heating is produced and vehicle has to face large drag induced during flight. Both of these factors alters the aerodynamic efficiency. Drag force and heating can be minimized by adopting the techniques like aero disk, counter flow of jet etc.

ATD and aerodynamic results can be evaluated through Computational Fluid Dynamics. Navier- Stokes equations are taken in to consideration for writing the governing equations corresponding to each type of energy of flow field.

**Complete description of flow field is given by:**

- Using the three momentum equations
- Using three energy equations

Seven species and seven type of chemical reactions are included in the Blottner’s model. This model gives the appropriate rates, which are in close agreement with the data obtained through experiments.

<table>
<thead>
<tr>
<th>Model</th>
<th>Species</th>
<th>Chemical reactions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blottner</td>
<td>7</td>
<td>7</td>
</tr>
<tr>
<td>Dunn and kang</td>
<td>11</td>
<td>26</td>
</tr>
<tr>
<td>Gupta</td>
<td>11</td>
<td>20</td>
</tr>
<tr>
<td>Park</td>
<td>11</td>
<td>17</td>
</tr>
</tbody>
</table>
In Gupta model, forward and backward reaction rate coefficients are calculated using the formula given by Arrhenius. The above mentioned models are accepted by computational fluid dynamics software for e.g LAURA, FUN3D etc. and provides the information regarding ATD in case of hypersonic flow like on ELECTRE vehicle etc.

A little uncertainty also exists because the insight mechanism is not completely known. Governing equations of mass, momentum and energy is solved using finite volume approach. In this approach we take only the selected region of flow field and obtain the solution on structured meshes. A step by step process is followed in this approach. Flow velocity at the solid wall is zero and flow is considered as a continuum. It results non-slip condition and concentration gradients become zero in this situation also surface behaves as an isothermal surface.

Super catalytic B.C allows the maximum transfer of heat and it remains conserved in the boundary layer. It contributes in the prediction of surface chemistry. In this all the constituents of fluid are taken simultaneously.

**Assumptions of symmetric boundary conditions:**

- In symmetry plane, zero value is obtained for normal component of velocity.
- Scalar quantities involved in the boundary condition is assumed to zero.
- If the plane and normal gradients are in same direction, then it should also be equal to zero.

Multi block structured meshes are employed in the discretization process. To account the viscous effect, second order central difference scheme is used. In the marching process, LUSGS approach is used.

All the four models provides the same rate coefficients. To get the reliable heating rates, grids are aligned such that value of Reynolds number becomes 10. Shock waves are detected using the 0.34 million cells. Some error occurs in the measurement of heat flux by thermocouples. In the free stream assessment, twenty percent error results in flight parameters.

Pressure gradients and Mach contours are characteristics of symmetry plane. When ELECTRE vehicle flies at hypersonic speed then compressible effects dominate and shock wave creates large pressure around the body especially on the surface of nose. As soon as the flow expands, pressure decreases. It happens due to increasing Mach number at this stage. Creation of recompression takes place due to interaction of expansion waves at shoulder and separated flow. In the design of vehicle, radius of nose plays an important role. In case of small nose radius, rate of heat accumulation becomes very high. Dissociation and
recombination of nitrogen and oxygen influence the rate of chemical reactions and heat flux over the body surface.

Evgeny A. Chinnov Heat and mass transfer can be increased using the thin liquid films. In these films, movement of flow creates natural waves which leads to enhancement in heat and mass transfer. To investigate this enhanced rate of transfer, an experiment conducted in which Reynolds number varies from one to forty five. To detect the coefficient of heat transfer disturbances are created on the falling water film. In this process, Reynolds number is varied up to 250. In this phenomenon two separate rivulets are formed on the upper surface of heater and separated by some finite wavelength. Boundary conditions are assumed constant over the surface and deformation in the layer thickness takes place. Thickness of the film increases due to stress produced by the thermo capillary action.

Rivulets forms on the surface at critical number. On the “B” part rivulet forms due to unequal temperature of surface. Unstable hydrodynamic waves are formed far away from the nozzle. Two dimensional synchronous waves distributes the water in a particular direction. Quasi regular metastable structure formation takes place on the water surface and inside it temperature is measured using infrared camera. Confocal method is used to estimate the film thickness. Experimental results shows that thickness of local film decreases with increase in temperature. As the heat increases over the surface, temperature becomes non uniform. In this way large pulsation is induced which in turn reaches on the lower part.

Rivulet crest are independent of amplitude of wave and temperature. Film get ruptured when we impose the condition of external forces. It is worthy to mention that these phenomenon are necessary in the assessment of transfer coefficients. Shape of film is changed when disturbances are created by a cylinder. In the “A” region, most dangerous wavelength is obtained for a Reynolds number less than forty and temperature of this region increases sharply. Dimensionless parameters are introduced to evaluate the liquid surface parameters. Under the effect of external forces, rivulet distance alters until Reynolds number approaches to 500. Fluorescent method can also be employed in the analysis of isothermal flow over the surface.

<table>
<thead>
<tr>
<th>INSTRUMENT/DEVICES</th>
<th>PARAMETER CALCULATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Particle image velocimetry</td>
<td>Flow of liquid films</td>
</tr>
<tr>
<td>IR detector</td>
<td>Temperature and velocity</td>
</tr>
<tr>
<td>Fluorescent method</td>
<td>Isothermal flow</td>
</tr>
<tr>
<td>Capacitive /Confocal method</td>
<td>Thickness of film</td>
</tr>
</tbody>
</table>
Laser 7 To excite the phosphor
Rhoda mine 6G Fluorescent dye
Thermo gram Average phase velocity

Structure, velocity, thickness and energy is evaluated in this experiment with Mach number up to 500.

**Components of experiment:**

1. Heater inside the test section
2. Plate and filter
3. Electric pump and nozzle
4. Closed circulation

In the experimental set up, the ambient temperature is achieved through temperature stabilizer. A 263mm distance is fixed from nozzle to heater. On the surface of heater, boundary conditions remains constant and Reynolds number varies from 300 to 500. To generate the most dangerous wavelength, cylinders are dipped into liquid to create the disturbances. A wide range of concentrated solutions are used which affects the working solution and brightness of liquid. Water film appears black in the range from 3.7 to 4.8 micrometer. Distance between the rivulet decreases the increase in external effects and consequently rivulet get displaced. Thickness of film changes in upper and lower region. At the lower surface of heater, rate of deformation becomes very high. When deformations becomes independent of heat flux then breakdown through film resistance can not be avoided.

As it is clear that transverse deformations like shear stress and rivulets are directly proportional to the heat flux. At lower value of heat flux no appreciable change occurs but threshold value is attained at higher value. Inertial forces becomes effective at high Reynolds number. Variations in the film thickness can be avoided by making the movement along the heater. In this process, thermo capillary forces are diminished a smooth surface is obtained but the wave amplitude increases.

Periodically moving heat source alters the position of fluid particles and they comes in to motion. It can be used in various applications of daily life. In this paper focus is on the flow mechanism during passage of moving heat source and minimization of undesired results. For an example, in the welding process, two materials get mixed and frozen.
This leads to phase change and surrounding area is also affected. In the varicose vein, only selected area is effectively influenced by EVLA. A fiber consisting of core, cladding and jacket is entered to vein through a catheter and blood is heated through laser by accompanying the various mode of heat transfer. Some critical conditions are necessary for it. Bubbles appear on the tip surface. Light of very high intensity is emitted and a carbon layer surrounds the tip. Thickness of layer changes with moving source. Generally water is taken in the fluid so that occurrence of bubbles can be visualized.

A large composition of blood is comprised by water so the results obtained are resembles with blood. However in spite of having maximum similarity, the major drawback encounters in absorption coefficient of water and blood. In the experimental set up a moving source of 25W is required and convection is caused by difference in temperature. This temperature difference reaches to 20 degree Celsius at the surface of tip. Different parameters like velocity, temperature etc. are visualized through Gr number. Temperature falls down in the y direction. Convection effects can be ignored at Gr = 10. If the absorption rate of light is very high then velocity factor can be neglected because at this stage temperature gradients no longer depends on velocity.

**Procedure:**

1. Quartz reservoir is used.
2. Lauda thermal bath is used to maintain the desired temperature.
3. Length of test section = 84mm
4. Inner radius = 2m
5. External radius = 3mm
6. Quartz holder is joined to test section.
7. Fiber without cladding is preferred.
8. In the controlled volume, 6 thermocouples are used.
9. CMOS sensors are used to record data.
10. Constant speed of fiber is maintained through linear electric motor.
11. All components of experiment are triggered through a triggering system.
12. Cartesian coordinate system approach is employed in the profile.

**John Shrimpton**

Atomizer is used to inject the charge in a controlled way. Current and flow rates are produced using a chemically reactive tip with radius of one micrometer. In the atomizer, negative potential is given to central electrode. High electric field of etched tip injects the charge but
this process is not so simple. A series of injection point can be induced using eutectic material. Brass is used to construct the conducting part of atomizer. High voltage electrode and electrically insulated liquids are enclosed in the body. It is done so to prevent the corona discharge due to high potential on electrodes. To establish the flow field a reservoir with high pressure is used. The liquid of reservoir is filtered and monitored in order to get ambient condition of flow. A digital multi meter is used to measure the spray current and conduction of electric charge in the upstream can be ignored. All the spray current is not measured which in turn causes error.

It has been experimentally proven that when a liquid is recycled again and again then its molecules dissociates and charge through them can be easily injected. Hence in the experiment one liquid is used only once. No repetition is allowed. Which liquid will be suitable? It is decided on the basis of ion nobilities. Critical atomizer potential is generally greater than 6 kilo volt. Resistivity of diesel oil decreases due to traces of detergent. In an atomizer, spray and leakage current increases with increase in voltage and total current. Total current is assumed to be equal to their addition.

**Relationship between voltage and current can be predicted through:**

1. Sub critical breakdown
2. Supercritical breakdown

While observing the breakdown, we find that current depends only on the applied voltage and no role is played by the bulk velocity in the total current. V-I characteristics strongly depends on the liquid property whereas leakage current and tip radius are main parameters in atomizer. Lower viscosity of liquid is avoided because their ions are strongly deflected by electric field. Maximum component of electrical velocity can be predicted using the field strength of $10^8$ volt per meter.

**Factors affecting the spray current:**

- Applied voltage
- Rate of flow
- Viscosity of liquid
- Drag force
- Movement of ions
- Radius of etched tip
As we know that large number charge accumulates on the tip so its interaction with the boundary layer can affect the injection process. Thin boundary layer induces more charge and increases the total current.

**Electric field depends on:**
- Electrode used.
- Geometry
- Distribution of space charge

When flow of liquid is reduced at a given voltage then it can sustain the condition of electrical breakdown. In nozzle, breakdown takes place when field strength becomes greater than the breakdown strength. Hence this situation causes cessation and leakage current. In version 1, initially spray and leakage current increases at a constant rate and then gains an optimum condition.

However after gaining the steady state, spray current further reduces and current start to flow through nozzle and body. 3 types of liquid provides best results over this regime. The major difference between the sub-critical and supercritical breakdown is that no hindrance is created during charge injection in super critical breakdown. Field of flow and electricity are coupled in an atomizer. Characteristics of catastrophic breakdown and critical breakdown are not resembles at all so charge injection mechanism is also different.

During partial breakdown, corona discharge mechanism is established in terms of radio frequency. In surface corona discharge, spray current reduces because a part of it goes to the earth. Hence we can conclude that following happens:

<table>
<thead>
<tr>
<th>Regime</th>
<th>Phenomenon</th>
</tr>
</thead>
<tbody>
<tr>
<td>Supercritical</td>
<td>Corona discharge</td>
</tr>
<tr>
<td>Subcritical</td>
<td>Electrical phenomenon</td>
</tr>
</tbody>
</table>

As flow rate increases, critical voltage also increases. At the tip of needle, surface of electrode melts at needle tip radius of 5 micrometer.

Sharp decrease and truncated curves are obtained in nozzle in case of super and subcritical breakdown respectively. In the channel, radial component of electric field and space charge can be related through Poisson equation. To solve the Poisson equation it is imagined that radial profile of charge is uniform throughout and potential is zero at boundary conditions.

**K.S.Kulkarni** Transfer of heat and mass from the solid body to the fluid is accompanied by the diffusion process. To find out the flow properties, firstly a model is assumed and corresponding equations are written and solved using various techniques. All the assumed
constant properties resembles in case of unity Lewis number. To visualize he mass transfer, a solid naphthalene is taken in which sublimation and convection both supports the mass transfer. Sublimation rate of naphthalene can be estimated through coefficient of mass transfer. Hence in this way a problem of heat transfer can be transformed in to problem of mass transfer

Why we observe naphthalene sublimation?

- If we take the surface which is depleted by naphthalene, then we can assume the surface as an isothermal surface.
- In the isothermal surfaces, radiation effect does not takes place.
- Conduction through wall is not possible.
- If conduction through wall is not there then heat transfer coefficient can be easily calculated.
- In the obtained solution of governing equations, singularity point can be easily attained.
- For practical model application, we can give the desired shape to solid naphthalene.
- Measurement of heat transfer coefficients is easy because it does not requires the intrusive technique rather surface scanning devices are employed.
- Better results in resolution and accuracy.

Heat and mass transfer rate in air and naphthalene are not same so to equate them a analogy named as heat and mass transfer analogy is taken in to consideration. It may be some constant. In some cases, it depends on the dimensionless numbers. For boundary layer flow, its quantitative analysis is necessary.

It is shown experimentally that turbulent diffusivities are equal for simple flows. Transfer coefficients are related by analogy factor by taking air for heat transfer and sublimation of naphthalene for mass transfer. Approximations are made because temperature of air and solid surfaces are not same. In this situation, condition of isothermal wall is imposed. In the present case, transformation equations are related through a constant.

**These two processes can be differentiated in case of impermeable wall as:**

1. In case of heat transfer, value obtained for normal component of velocity is zero.
2. In case of sublimation, nonzero value is obtained for normal velocity component.

The latter can be neglected in the transformation process. Embedded thermocouples measure the temperature of surface. NIT (Numerical Integration Techniques) are used to find out the similarity solution. In the experimental setup, to extract the properties of flow field and heat / mass transfer, facility of open circuit wind tunnel is taken in to consideration.

**Dimensions of wind tunnel and heat transfer plate:**
And
Length of heat transfer plate = 228.6mm
Breadth = 152mm
Thickness = 12.7mm

In the test section, air is accelerated to get the free stream turbulence flow. Sharp edges of Plexiglas are used as leading edge and uniform laminar flow is maintained over it. A finite thickness of boundary layer forms on the plate. In the laminar and turbulent boundary fluctuations vary with time. Velocity profile can be evaluated through hot wire probe. True velocity of wall increases due to convection of heat. Top of heat transfer plate is made of aluminum and fixed the base at insulator. Within the plates, ten heaters are inserted. To attain the controlled temperature near the leading edges, four narrow strip heaters are used. Upstream conduction can be minimized through wood plate at leading edge.

In this way, boundary layer can be heated sharply. Errors occurs in the measurement can be minimized by doing all the measurements along the line. Desired boundary conditions can be obtained by a computer controlled power supply and steady state can be attained within half an hour.

**Steps of thermal boundary layer calculation:**

- Find out the profile of thermal boundary layer
- Apply Fourier’s law within the layer
- Extrapolate the profile of thermal boundary layer
- Set the profile properly
- Fit the wall finding circuit
- Connect the thermocouple and plate

We can calculate the heat flux of boundary layer by making use of temperature profile. Some uncertainties exit in the velocity profile of turbulent layer. It is just because of human error. So the profile is analyzed through a technique. For a finite radius of thermocouple bead, uncertainties predominant, so we can take conservative section away from this region. In the boundary layer, profile of temperature can be determined through cubic polynomial fit. Thermal gradients are the slope of these polynomials. Wires used in thermocouple influence the heat flux and temperature of laminar and turbulent boundary layer.
In the mass transfer analysis, a plate of 2.54mm × 3mm dimension is used with polished surfaces. Smoothness of solid plate can be maintained by mirror polishing. Surface profile of mass transfer can be calculated from the polished rim. This rim can be taken as the edges of reference plane. E type thermocouples are placed in the solid naphthalene which controls the temperature. Wood section is attached to the plate which matches the heat and mass transfer conditions. Scanning system used to detect the surface of naphthalene can move in the x and z directions. This movement is controlled through computer. A multi meter is used to detect the output of LVDT. Uncertainties arises in the measurement of mass transfer, which can be avoided by careful mounting of plates.

Process of sublimation consists of following steps:

- A layer of naphthalene is placed on the under investigated surface area.
- Surface is exposed to flow
- Thickness of sublimation can be evaluated by again measuring the profile of surface
- Analysis is made to achieve the coefficient of mass transfer
- A dimensionless number, Sherwood number is used to transform the mass transfer coefficient which requires a reasonable length scale.

Experiment shows that the origin of boundary layer and leading edge are not same rather an origin is imagined for our calculation purpose.

**Final steps:**

- Modelling of flow is done by making use of commercial software, FLUENT.
- It is impossible to model the every component of wind tunnel.
- A boundary layer mesh with cell dimensions of 5 micrometer is used to get the accurate solution.
- Steady state solver is employed in the boundary layer to find out the solution for energy and momentum equations
- Upwind discretization of first and second order is taken in to consideration for generalized equations
- Convergence limit is attained in case of constant parameter