"Introduction to Aerodynamics".

- In Section 1 of this course you will cover these topics:
- Fluid Properties
- Fundamentals Of Fluid Mechanics
- Dynamics Of An Incompressible, Inviscid Flow Field

Topic Objective:

At the end of this topic student will be able to understand:

- Concept of a Fluid
- Fluid as a Continuum
- Fluid Properties
- The Standard Atmosphere

Definition/Overview:

The properties outlines below are general properties of fluids which are of interest in engineering. The symbol usually used to represent the property is specified together with some typical values in SI units for common fluids. Values under specific conditions (temperature, pressure etc.) can be readily found in many reference books. The dimensions of each unit are also giving in the MLT system (see later in the section on dimensional analysis for more details about dimensions.)

Key Points:

1. Concept of a Fluid

A fluid is defined as a substance that continually deforms (flows) under an applied shear stress. All liquids and all gases are fluids. Fluids are a subset of the phases of matter and include liquids, gases, plasmas and, to some extent, plastic solids.

In common usage, "fluid" is often used as a synonym for "liquid", with no implication that gas could also be present. For example, "brake fluid" is hydraulic oil and will not perform its required function if there is gas in it. This colloquial usage of the term is also common in medicine ("take plenty of fluids"), and in nutrition.

Liquids form a free surface (that is, a surface not created by the container) while gases do not. The distinction between solids and fluid is not entirely obvious. The distinction is made by evaluating the viscosity of the substance. Silly Putty can be considered to behave like a solid or a fluid, depending on the time period over which it is observed. It is best described as a viscoelastic fluid.

Depending on the relationship between shear stress, and the rate of strain and its derivatives, fluids can be characterized as:

- Newtonian fluids : where stress is directly proportional to rate of strain, and
- Non-Newtonian fluids : where stress is proportional to rate of strain, its higher powers and derivatives.

The behavior of fluids can be described by the Navier-Stokes equationsa set of partial differential equations which are based on:

- continuity (conservation of mass),
- conservation of linear momentum
- conservation of angular momentum
- conservation of energy.

The study of fluids is fluid mechanics, which is subdivided into fluid dynamics and fluid statics depending on whether the fluid is in motion.

2. Fluid as a Continuum

Fluid as a continuum is a material continuum that responds to stress by irrecoverable flow. Continuum mechanics is a branch of mechanics that deals with the analysis of the kinematics and mechanical behavior of materials modeled as a continuum, e.g., solids and fluids (i.e., liquids and gases). A continuum concept assumes that the substance of the body is distributed throughout and completely fills the space it occupies. The continuum concept ignores the fact that matter is made of atoms, is not continuous, and that it commonly has some sort of heterogeneous microstructure, allowing the approximation of physical quantities, such as energy and momentum, at the infinitesimal limit. Differential equations can thus be employed in solving problems in continuum mechanics. Some of these differential equations are specific to the materials being investigated and are called constitutive equations, while others capture fundamental physical laws, such as conservation of mass (continuity equation), the conservation of momentum (equations of motion and equilibrium), and energy (first law of thermodynamics).

Continuum mechanics deals with physical quantities of solids and fluids which are independent of any particular coordinate system in which they are observed. These physical quantities are then represented by tensors, which are mathematical objects that are independent of coordinate system. These tensors can be expressed in coordinate systems for computational convenience. In fluids, the Knudsen number is used to assess to what extent the approximation of continuity can be made.

Materials, such as solids, liquids and gases, are composed of molecules separated by empty space. In a macroscopic scale, materials have cracks and discontinuities. However, certain physical phenomena can be modeled assuming the materials exist as a continuum, meaning the matter in the body is continuously distributed and fills the entire region of space it occupies. A continuum is a body that can be continually sub-divided into infinitesimal elements with properties being those of the bulk material. The concept of continuum is a macroscopic physical model, and its validity depends on the type of problem and the scale of the physical phenomena under consideration. A material may be assumed to be a continuum when the distance between the physical particles is very small compared to the dimension of the problem. For example, such is the case when analyzing the deformation behavior of soil deposits in soil mechanics. A given volume of soil is composed of discrete solid particles (grains) of minerals that are packed in a certain manner with voids between them. In this sense, soils evade the definition of a continuum. To simplify the deformation analysis of the soil, the volume of soil can be assumed to be a continuum because the grain particles are very small compared to the scale of the problem. The validity of the continuum assumption needs to be verified with experimental testing and measurements on the real material under consideration and under similar loading conditions.

2.1. Relationship to continuum mechanics

Fluid mechanics is a subdiscipline of continuum mechanics, as illustrated in the following table.

<table>
<tr><td rowspan="4">Continuum mechanics the study of the physics of continuous materials</td><td rowspan="2">Solid mechanics: the study of the physics of continuous materials with a defined rest shape.</td><td colspan="2">Elasticity: This describes materials that return to their rest shape after an applied stress.</td></tr>
<tr><td>Plasticity: This describes materials that permanently deform after a large enough applied stress.</td><td rowspan="2">Rheology: the study of materials with both solid and fluid characteristics</td></tr>
<tr><td rowspan="2">Fluid mechanics: the study of the physics of continuous materials which take the shape of their container.</td><td>Non-Newtonian fluids</td></tr>
<tr><td colspan="2">Newtonian fluids</td></tr>
</table>

[Table 1: Fluid mechanics is a subdiscipline]

In a mechanical view, a fluid is a substance that does not support tangential stress; that is why a fluid at rest has the shape of its containing vessel. A fluid at rest has no shear stress.

3. Fluid Properties

Fluids display such properties as:

- Not resisting deformation, or resisting it only lightly (viscosity), and
- The ability to flow (also described as the ability to take on the shape of the container).

These properties are typically a function of their inability to support a shear stress in static equilibrium.

Solids can be subjected to shear stresses, and to normal stresses - both compressive and tensile. In contrast, ideal fluids can only be subjected to normal, compressive stress which is called pressure. Real fluids display viscosity and so are capable of being subjected to low levels of shear stress.

In a solid, shear stress is a funvion of strain, but in a fluid, shear stress is a function of rate of strain. A consequence of this behavior is Pascal's law which describes the role of pressure in characterizing a fluid's state.

1.1. Here are some important properties of fluids

1.1.1. Density or Mass Density of fluid

Mass density or simply density of the fluid is defined as the ratio of the mass of fluid to its volume. Density of the fluid can also be defined as the mass per unit volume of the fluid. Density of the fluid is denoted by the symbol ρ (rho).

In SI system the unit of measurement of the mass of the fluid is kg and that of the volume of fluid is cubic meter m^3. Thus the unit of measurement of density is kg/ cubic meter (kg/m^3). The density of the liquids is considered to be constant with the temperature, but of the gases it changes with the temperature.

The density of water is considered to be 1000kg/m3 or 1gm/cm3 and it is standard for the measurement of the density of the other fluids.

1.1.2 .Specific Weight or Weight Density of the fluid

The specific weight or the weight density of the fluid is defined as the ratio of the weight of the fluid to the volume of the fluid. The weight density of the fluid is also defined as the weight of the fluid per unit volume of the fluid. Weight density of the fluid is denoted by the symbol w.

In SI system the unit of measurement of the weight of the fluid is N (Newton) and that of volume is cubic meter m3, hence the unit of measurement of specific weight is N/m3 (Newton per meter cube).

The value of the specific density of water in SI unit is 9.81 x 1000 Newton/m3.

4. The Standard Atmosphere

- A standard reference value for air pressure:
    1. Atmosphere (unit), an approximation of the value at sea level
    2. Atmospheric pressure, other reference values
- A model of how atmospheric pressure varies with altitude:
    1. The U.S. Standard Atmosphere, a series of models that give values for pressure, density, and temperature over a range of altitudes
    2. The International Standard Atmosphere (ISA), an international standard model, defining typical atmospheric properties with altitude, at mid-latitude.

Topic Objective:

At the end of this topic student will be able to understand:

- Fluid Dynamics

- Conservation of Mass
- Conservation of Linear Momentum
- Reynolds Number and Mach Number as Similarity Parameters
- Concept of the Boundary Layer
- Conservation of Energy
- First Law of Thermodynamics

Definition/Overview:

Fluid mechanics is the study of how fluids move and the forces on them. (Fluids include liquids and gases.) Fluid mechanics can be divided into fluid statics, the study of fluids at rest, and fluid dynamics, the study of fluids in motion. It is a branch of continuum mechanics, a subject which models matter without using the information that it is made out of atoms. Fluid mechanics, especially fluid dynamics, is an active field of research with many unsolved or partly solved problems. Fluid mechanics can be mathematically complex. Sometimes it can best be solved by numerical methods, typically using computers. A modern discipline, called Computational Fluid Dynamics (CFD), is devoted to this approach to solving fluid mechanics problems. Also taking advantage of the highly visual nature of fluid flow is Particle Image Velocimetry, an experimental method for visualizing and analyzing fluid flow. Fluid mechanics is the branch of physics which deals with the properties of fluid, namely liquid and gases, and their interaction with forces.

The study of fluid mechanics goes back at least to the days of ancient Greece, when Archimedes investigated fluid statics and buoyancy. Medieval Persian natural philosophers, including Abū Rayhān al-Bīrūnī and Al-Khazini, combined that earlier work with dynamics to presage the later development of fluid dynamics. Rapid advancement in fluid mechanics began with Leonardo da Vinci (observation and experiment), Evangelista Torricelli (barometer), Isaac Newton (viscosity) and Blaise Pascal (hydrostatics), and was continued by Daniel Bernoulli with the introduction of mathematical fluid dynamics in Hydrodynamica (1738). Inviscid flow was further analyzed by various mathematicians (Leonhard Euler, d'Alembert, Lagrange, Laplace, and Poisson) and viscous flow was explored by a multitude of engineers including Poiseuille and Gotthilf Heinrich Ludwig Hagen. Further mathematical justification was provided by Claude-Louis Navier and George Gabriel Stokes in the Navier-Stokes Equations, and boundary layers were investigated (Ludwig Prandtl), while various scientists (Osborne Reynolds, Andrey Kolmogorov, and Geoffrey Ingram Taylor) advanced the understanding of fluid viscosity and turbulence.

Key Points:

1. Introduction to Fluid Dynamics

Fluid dynamics is the sub-discipline of fluid mechanics dealing with fluid flow: fluids (liquids and gases) in motion. It has several sub-disciplines itself, including aerodynamics (the study of gases in motion) and hydrodynamics (the study of liquids in motion). Fluid dynamics has a wide range of applications, including calculating forces and moments on aircraft, determining the mass flow rate of petroleum through pipelines, predicting weather patterns, understanding nebulae in interstellar space and reportedly modeling fission weapon detonation. Some of its principles are even used in traffic engineering, where traffic is treated as a continuous fluid. Fluid dynamics offers a systematic structure that underlies these practical disciplines and that embraces empirical and semi-empirical laws, derived from flow measurement, used to solve practical problems. The solution of a fluid dynamics problem typically involves calculation of various properties of the fluid, such as velocity, pressure, density, and temperature, as functions of space and time.

2. Conservation of Mass

The law of conservation of mass/matter, also known as law of mass/matter conservation says that the mass of a closed system will remain constant, regardless of the processes acting inside the system. An equivalent statement is that matter cannot be created/destroyed, although it may be rearranged. This implies that for any chemical process in a closed system, the mass of the reactants must equal the mass of the products. This is also the central idea behind the first law of Thermodynamics. The law of "matter" conservation (in the sense of conservation of particles) may be considered as an approximate physical law that holds only in the classical sense before the advent of special relativity and quantum mechanics. Mass is also not generally conserved in open systems, when various forms of energy are allowed into, or out of, the system. However, the law of mass conservation for closed systems, as viewed over time from any single inertial frame, continues to hold in modern physics. This historical concept is widely used in many fields such as chemistry, mechanics, and fluid dynamics.

3. Conservation of Linear Momentum

The law of conservation of linear momentum is a fundamental law of nature, and it states that the total momentum of a closed system of objects (which has no interactions with external agents) is constant. One of the consequences of this is that the center of mass of any system of objects will always continue with the same velocity unless acted on by a force from outside the system. Conservation of momentum is a mathematical consequence of the homogeneity (shift symmetry) of space (position in space is the canonical conjugate quantity to momentum). So, momentum conservation can be philosophically stated as "nothing depends on location per se". In analytical mechanics the conservation of momentum is a consequence of translational invariance of Lagrangian in the absence of external forces. It can be proven that the total momentum is a constant of motion by making an infinitesmial translation of Lagrangian and then equating it with non translated Lagrangian. This is a special case of Noether's theorem. In an isolated system (one where external forces are absent) the total momentum will be constant: this is implied by Newton's first law of motion. Newton's third law of motion, the law of reciprocal actions, which dictates that the forces acting between systems are equal in magnitude, but opposite in sign, is due to the conservation of momentum. Since position in space is a vector quantity, momentum (being the canonical conjugate of position) is a vector quantity as well - it has direction. Thus, when a gun is fired, the final total momentum of the system (the gun and the bullet) is the vector sum of the momenta of these two objects. Assuming that the gun and bullet were at rest prior to firing (meaning the initial momentum of the system was zero), the final total momentum must also equal 0. In an isolated system with only two objects, the change in momentum of one object must be equal and opposite to the change in momentum of the other object. Mathematically,

Momentum has the special property that, in a closed system, it is always conserved, even in collisions and separations caused by explosive forces. Kinetic energy, on the other hand, is not conserved in collisions if they are inelastic. Since momentum is conserved it can be used to calculate an unknown velocity following a collision or a separation if all the other masses and velocities are known. A common problem in physics that requires the use of this fact is the collision of two particles. Since momentum is always conserved, the sum of the momenta before the collision must equal the sum of the momenta after the collision:

Where:

- u signifies vector velocity before the collision
- v signifies vector velocity after the collision.

Usually, we either only know the velocities before or after a collision or would like to also find out the opposite. Correctly solving this problem means you have to know what kind of collision took place. There are two basic kinds of collisions, both of which conserve momentum:

- Elastic collisions conserve kinetic energy as well as total momentum before and after collision.
- Inelastic collisions don't conserve kinetic energy, but total momentum before and after collision is conserved.

3.1. Elastic collisions

A collision between two pool balls is a good example of an almost totally elastic collision; a totally elastic collision exists only in theory. In addition to momentum being conserved when the two balls collide, the sum of kinetic energy before a collision must equal the sum of kinetic energy after:

Since the 1/2 factor is common to all the terms, it can be taken out right away.

3.2. Special case: m1>>m2

Now consider the case when the mass of one body, say m1, is far greater than that of the other, m2 (m1>>m2). In that case, m1+m2 and m1-m2 are approximately equal to m1.

Its physical interpretation is that in the case of a collision between two bodies, one of which is much more massive than the other, the lighter body ends up moving in the opposite direction with twice the original speed of the more massive body.

3.4. Special case: m1=m2

Another special case is when the collision is between two bodies of equal mass. Say body m1 moving at velocity v1 strikes body m2. Putting this case in the equation derived above we will see that after the collision, the body that was moving (m1) will start moving with velocity v2 and the mass m2 will start moving with velocity v1. So there will be an exchange of velocities.

Now suppose one of the masses, say m2, was at rest. In that case after the collision the moving body, m1, will come to rest and the body that was at rest, m2, will start moving with the velocity that m1 had before the collision.

Note that all of these observations are for an elastic collision.

This phenomenon is demonstrated by Newton's cradle, one of the best known examples of conservation of momentum, a real life example of this special case.

3.5. Multi-dimensional collisions

In the case of objects colliding in more than one dimension, as in oblique collisions, the velocity is resolved into orthogonal components with one component perpendicular to the plane of collision and the other component or components in the plane of collision. The velocity components in the plane of collision remain unchanged, while the velocity perpendicular to the plane of collision is calculated in the same way as the one-dimensional case.

For example, in a two-dimensional collision, the momenta can be resolved into x and y components. We can then calculate each component separately, and combine them to produce a vector result. The magnitude of this vector is the final momentum of the isolated system.

3.6. Inelastic collisions

A common example of a perfectly inelastic collision is when two snowballs collide and then stick together afterwards. This equation describes the conservation of momentum:

It can be shown that a perfectly inelastic collision is one in which the maximum amount of kinetic energy is converted into other forms. For instance, if both objects stick together after the collision and move with a final common velocity, one can always find a reference frame in which the objects are brought to rest by the collision and 100% of the kinetic energy is converted. This is true even in the relativistic case and utilized in particle accelerators to efficiently convert kinetic energy into new forms of mass-energy (i.e. to create massive particles).

In case of Inelastic collision, there is a parameter attached called coefficient of restitution (denoted by small 'e' or 'c' in many text books). It is defined as the ratio of relative velocity of separation to relative velocity of approach. It is a ratio hence it is a dimensionless quantity.

When we have an elastic collision the value of e (= coefficient of restitution) is 1, i.e. the relative velocity of approach is same as the relative velocity of separation of the colliding bodies. In an elastic collision the Kinetic energy of the system is conserved.

When a collision is not elastic (e<1) it is an inelastic collision. In case of a perfectly inelastic collision the relative velocity of separation of the centre of masses of the colliding bodies is 0. Hence after collision the bodies stick together after collision. In case of an inelastic collision the loss of Kinetic energy is maximum as stated above.

In all types of collision if no external force is acting on the system of colliding bodies, the momentum will always be preserved.

3.7. Explosions

An explosion occurs when an object is divided into two or more fragments due to a release of energy. Note that kinetic energy in a system of explosion is not conserved because it involves energy transformation (i.e. kinetic energy changes into heat and sound energy).

4. Reynolds Number and Mach number as Similarity Parameters

The Reynolds number is the most important dimensionless number in fluid dynamics providing a criterion for dynamic similarity. It is named after Osbourne Reynolds (1842-1912). Typically it is given as follows:

- $Re = \rho * v * L / \eta$ or
- $Re = v * L / \nu$

Or With:

- vs - mean fluid velocity,
- L - characteristic length (equal to diameter 2r if a cross-section is circular),
- η - (absolute) dynamic fluid viscosity,
- ν - kinematic fluid viscosity: $\nu = \eta / \rho$,
- ρ - Fluid density.

The Reynolds number is used for determing whether a flow is laminar or turbulent. Laminar flow within e.g. pipes will occur when the Reynolds number is below the critical Reynolds number of Recrit, pipe = 2300 (or practically Re > 3000) and turbulent flow when it is above 2300 where the Reynolds number is based on the pipe diameter and the mean velocity vs. within the pipe. The value of 2300 has been determined experimentally and a certain range around this value is considered the transition region between laminar and turbulent flow. Please note that the critical Reynolds number Recrit depends on the flow type and the definition of the Reynolds number.

Mach number is so frequently used to reference speed because it is what we in the aerodynamics world call a similarity parameter. What that term implies is that any vehicle traveling at a given Mach number will experience similar aerodynamic properties, regardless of its altitude or dimensional airspeed. In addition, similarity parameters make it possible to measure the aerodynamic properties of a subscale model in a wind tunnel rather than having to test a full scale vehicle in free flight.

To further investigate the concept, let's take a moment to think about where these similarity parameters come from. In a number of previous questions, we have talked about the lift equation. In those discussions, we have pointed out that lift is dependent upon a number of variables. These same factors also dictate five other aerodynamic forces or moments that act on a vehicle in flight, those being the drag, side-force, pitching moment, yawing moment, and rolling moment. In this example, let us simplify the discussion by focusing on an airfoil cross-section of an airplane's wing. In that case, the aerodynamic quantities of lift, drag, pitching moment generated by the airfoil are dictated by the following variables.

- The velocity, or airspeed, at which the plane flies, usually denoted as V (pronounced "V infinity").
- The altitude of the aircraft, which defines the density, denoted by the Greek symbol r (pronouned "rho").
- The size of the lifting surface, or the reference area. For an airplane, we use the wing area denoted by the variable S. Since our airfoil does not have an area, only length, we may substitute a reference length l.
- The angle of attack at which the vehicle meets the oncoming airflow, denoted by the Greek symbol a (pronounced "alpha").
- The overall shape of the airfoil.
- The viscosity of the fluid through which the vehicle flies, which is also a function of the altitude. Viscosity generates friction as the particles of air rush past the airfoil, and this friction induces aerodynamic forces. We represent viscosity by the Greek symbol m (pronounced "mu").
- The compressibility of the airflow. The faster air travels, the more it becomes compressed, or squeezed. It it is compressed enough, shock waves will form causing sudden changes in quantities like density, velocity, and viscosity. Since compressibility is governed by Mach number, we can represent these effects by the variable M. But since the Mach number is defined as the velocity divided by the speed of sound and we have already mentioned the importance of velocity, we will simplify to the speed of sound, denoted by the variable a.

In other words, for a given airfoil at a given angle of attack, we can say that the lift (L) it generates is a function of velocity, density, reference area or length, viscosity, and speed of sound.

We can write a similar relationship for both the drag (D) and pitching moment (M). If we wanted to use a wind tunnel to measure the variation in L, D, and M for a new airfoil, we could conduct a massive series of tests in which all five of those variables would be changed individually. However, such a series of experiments would take an inordinate amount of time. What if we could combine those variables in such a way that we would only have to vary a couple of parameters in order to fully investigates our airfoil?
Some of these quantities we have seen before. In particular, the parameter a/V is the inverse of the Mach number M. The third quantity is also a well known value in the world of aerodynamics called the Reynolds number.

We can therefore simplify our equation to the form

Let us now define a variable called the lift coefficient, which is of the form

When we plug that definition back into the previous equation, we obtain the following equation for lift, one we have seen many times before.

This equation can be simplified even further when we recognize the quantity dynamic pressure:

Which leaves us with this deceptively simple equation for lift?

So what has happened here? All of the complex aerodynamics that depend on quantities like viscosity and compressibility effects have been hidden away in the lift coefficient. When we use a wind tunnel to collect aerodynamic data, it is this coefficient we want to measure; because it contains all the dependencies we are concerned about. That's why, in a wind tunnel, we measure the actual lift force L and convert it into a non-dimensional coefficient by rearranging the above equation.

Instead of having to vary the five parameters discussed earlier, we now only need to vary two--Mach number and Reynolds number--and the effects of all other atmospheric properties are automatically included as well. Recall also that the above derivation assumed a constant angle of attack a. We typically must vary this parameter also such that the lift coefficient is a function of three variables.

The true significance of this methodology becomes apparent when we realize that the Mach and Reynolds numbers are both non-dimensional numbers. In other words, they are essentially independent of the size of the vehicle. This property allows us to test a small scale version of a real aircraft in a wind tunnel at the same Mach and Reynolds numbers as it would experience in actual flight, and the force and moment coefficients that we measure on this model will be identical to those the real aircraft would experience. When the airflows passing over the real aircraft and a scale model are at the same Mach and Reynolds numbers, we call them dynamically similar flows, which is where the term similarity parameter comes from.
In summation, the reason we so often quote aircraft performance in Mach numbers rather than velocities is because an aircraft traveling at the same Mach number will experience similar aerodynamic behaviors even if all other operating conditions are different. For example, an aircraft flying 685 mph (1,100 km/h) at sea level would be flying subsonically at about Mach

0.9. The same aircraft flying 685 mph at 50,000 ft (15,255 m) would be flying supersonically at about Mach 1.05. A difference of 0.15 Mach may not sound like much, but the behavior of the air flowing around the aircraft changes dramatically as it passes through the speed of sound.

5. Concept of the Boundary Layer

In physics and fluid mechanics, a boundary layer is that layer of fluid in the immediate vicinity of a bounding surface. In the Earth's atmosphere, the planetary boundary layer is the air layer near the ground affected by diurnal heat, moisture or momentum transfer to or from the surface. On an aircraft wing the boundary layer is the part of the flow close to the wing. The boundary layer effect occurs at the field region in which all changes occur in the flow pattern. The boundary layer distorts surrounding non-viscous flow. It is a phenomenon of viscous forces. This effect is related to the Reynolds number.

Laminar boundary layers come in various forms and can be loosely classified according to their structure and the circumstances under which they are created. The thin shear layer which develops on an oscillating body is an example of a Stokes boundary layer, whilst the Blasius boundary layer refers to the well-known similarity solution for the steady boundary layer attached to a flat plate held in an oncoming unidirectional flow. When a fluid rotates, viscous forces may be balanced by the Coriolis Effect, rather than convective inertia, leading to the formation of an Ekman layer. Thermal boundary layers also exist in heat transfer. Multiple types of boundary layers can coexist near a surface simultaneously.

6. Conservation of Energy

In physics, the law of conservation of energy states that the total amount of energy in an isolated system remains constant. A consequence of this law is that energy cannot be created or destroyed. The only thing that can happen with energy in an isolated system is that it can change form, that is to say for instance kinetic energy can become thermal energy. Another consequence of this law is that perpetual motion machines can only work if they deliver no energy to their surroundings, and that devices that produce more energy than is put into them are impossible.

7. First Law of Thermodynamics

In thermodynamics, the first law of thermodynamics is an expression of the more universal physical law of the conservation of energy. Succinctly, the first law of thermodynamics states:

The increase in the internal energy of a system is equal to the amount of energy added by heating the system, minus the amount lost as a result of the work done by the system on its surroundings.

7.1. Description

The first law of thermodynamics basically states that a thermodynamic system can store or hold energy and that this internal energy is conserved. Heat is a process by which energy is added to a system from a high-temperature source, or lost to a low-temperature sink. In addition, energy may be lost by the system when it does mechanical work on its surroundings, or conversely, it may gain energy as a result of work done on it by its surroundings. The first law states that this energy is conserved: The change in the internal energy is equal to the amount added by heating minus the amount lost by doing work on the environment. The first law can be stated mathematically as:

Where dU is a small increase in the internal energy of the system, δQ is a small amount of heat added to the system, and δW is a small amount of work done by the system.

Notice that a lot of textbooks (e.g., Greiner Neise Stocker) formulate the first law as

$$dU = \delta Q + \delta W:$$

The only difference here is that δW is the work done on the system. So, when the system (eg. gas) expands the work done on the system is − PdV whereas in the previous formulation of the first law, the work done by the gas while expanding is PdV. In any case, both give the same result when written explicitly as:

The δ's before the heat and work terms are used to indicate that they describe an increment of energy which is to be interpreted somewhat differently than the dU increment of internal energy. Work and heat are processes which add or subtract energy, while the internal energy U is a particular form of energy associated with the system. Thus the term "heat energy" for δQ means "that amount of energy added as the result of heating" rather than referring to a particular form of energy. Likewise, the term "work energy" for δw means "that amount of energy lost as the result of work". Internal energy is the property of the system whereas work done or heat supplied is not. The most significant result of this distinction is the fact that one can clearly state the amount of internal energy possessed by a thermodynamic system, but one cannot tell how much energy has flowed into or out of the system as a result of its being heated or cooled, nor as the result of work being performed on or by the system. The first explicit statement of the first law of thermodynamics was given by Rudolf Clausius in 1850: "There is a state function E, called energy, whose differential equals the work exchanged with the surroundings during an adiabatic process."

Topic Objective:

At the end of this topic student will be able to understand:

- Inviscid Flows

- Bernoulli's Equation
- Use of Bernoulli's Equation to Determine Airspeed
- The Pressure Coefficient
- Circulation
- Irrotational Flow
- Kelvin's Theorem
- Incompressible, Irrotational Flow
- Superposition of Flows & Elementary Flows
- Adding Elementary Flows to Describe Flow around a Cylinder
- Flow around a Cylinder with Circulation

Definition/Overview:

In fluid mechanics or more generally continuum mechanics, an incompressible flow is solid or fluid flow in which the divergence of velocity is zero. This is more precisely termed isochoric flow. It is an idealization used to simplify analysis. In reality, all materials are compressible to some extent. Note that isochoric refers to flow, not the material property. This means that under certain circumstances, a compressible material can undergo (nearly) incompressible flow. However, by making the 'incompressible' assumption, the governing equations of material flow can be simplified significantly.

Key Points:

1. Inviscid Flows

In fluid dynamics there are problems that are easily solved by using the simplifying assumption of an ideal fluid that has no viscosity. The flow of a fluid that is assumed to have no viscosity is called inviscid flow.

The flow of fluids with low values of viscosity agrees closely with inviscid flow everywhere except close to the fluid boundary where the boundary layer plays a significant role. This is generally true where viscous (friction) forces are small in comparison to inertial forces. The assumption that viscous forces are negligible can be used to simplify the Navier-Stokes solution to the Euler equations.

In the case of incompressible flow, the Euler equations governing inviscid flow are:

Which, in the steady-state case, can be solved using potential flow theory? More generally, Bernoulli's principle can be used to analyse certain time-dependent compressible and incompressible flows.

2. Bernoulli's Equation

Bernoulli's equation is one of the most important/useful equations in fluid mechanics. It may be written,

We see that from applying equal pressure or zero velocities we get the two equations from the section above. They are both just special cases of Bernoulli's equation.

Bernoulli's equation has some restrictions in its applicability, they are:

- Flow is steady;
- Density is constant (which also means the fluid is incompressible);
- Friction losses are negligible.

- The equation relates the states at two points along a single streamline, (not conditions on two different streamlines).

  All these conditions are impossible to satisfy at any instant in time! Fortunately for many real situations where the conditions are *approximately* satisfied, the equation gives very good results.

An element of fluid, as that in the figure above, has potential energy due to its height z above a datum and kinetic energy due to its velocity *u*. If the element has weight mg then

Potential energy =

Kinetic energy =

Kinetic energy per unit weight =

At any cross-section the pressure generates a force; the fluid will flow, moving the cross-section, so work will be done. If the pressure at cross section AB is *p* and the area of the cross-section is *a* then

Force on AB =

When the mass *mg* of fluid has passed AB, cross-section AB will have moved to A'B'

Volume passing AB =

therefore

Distance AA' =

Work done = force distance AA'

=

Work done per unit weight =

This term is know as the pressure energy of the flowing stream.

Summing all of these energy terms gives

Or

As all of these elements of the equation have units of length, they are often referred to as the following:

Pressure head =

Velocity head =

Potential head =

Total head =

By the principle of conservation of energy the total *energy* in the system does not change, thus the total *head* does not change. So the Bernoulli equation can be written

As stated above, the Bernoulli equation applies to conditions along a streamline. We can apply it between two points, 1 and 2, on the streamline in the figure below

[Fig 3: Two points joined by a streamline total energy per unit weight at 1 = total energy per unit weight at 2 or total head at 1 = total head at 2 or]

This equation assumes no energy losses (e.g. from friction) or energy gains (e.g. from a pump) along the streamline. It can be expanded to include these simply, by adding the appropriate energy terms:

3. Use of Bernoulli's Equation to Determine Airspeed

When the Bernoulli equation is combined with the continuity equation the two can be used to find velocities and pressures at points in the flow connected by a streamline.
Here is an example of using the Bernoulli equation to determine pressure and velocity at within a contracting and expanding pipe.

A fluid of constant density = 960 is flowing steadily through the above tube. The diameters at the sections are . The gauge pressure at 1 is

and the velocity here is . We want to know the gauge pressure at section 2.
We shall of course use the Bernoulli equation to do this and we apply it along a streamline joining section 1 with section 2.
The tube is horizontal, with $z_1 = z_2$ so Bernoulli gives us the following equation for pressure at section 2:

But we do not know the value of . We can calculate this from the continuity equation:
Discharge into the tube is equal to the discharge out i.e.

So we can now calculate the pressure at section 2

Notice how the velocity has increased while the pressure has decreased. The phenomenon - that pressure decreases as velocity increases - sometimes comes in very useful in engineering. (It is on this principle that carburettor in many car engines work - pressure reduces in a contraction allowing a small amount of fuel to enter).

Here we have used both the Bernoulli equation and the Continuity principle together to solve the problem. Use of this combination is very common. We will be seeing this again frequently throughout the rest of the course.

4. The Pressure Coefficient

The pressure coefficient is a dimensionless number less than one which describes the relative pressures throughout a flow field in fluid dynamics. The pressure coefficient is used in aerodynamics and hydrodynamics. Every point in a fluid flow field has its own unique pressure coefficient, Cp.

In many situations in aerodynamics and hydrodynamics, the pressure coefficient at a point near a body is independent of body size. Consequently an engineering model can be tested in a wind tunnel or water tunnel, pressure coefficients can be determined at critical locations around the model, and these pressure coefficients can be used with confidence to predict the fluid pressure at those critical locations around a full-size aircraft or boat.

5. Circulation

In fluid dynamics, **circulation** is the line integral around a closed curve of the fluid velocity. Circulation is normally denoted Γ. If is the fluid velocity and is a unit vector along the closed curve *C*:

The dimensions of circulation are length squared over time.
Circulation was first used independently by Frederick Lanchester, Wilhelm Kutta, and Nikolai Zhukovsky.

6. Irrotational Flow

This lecture is devoted to the study of irrotational plane flows of an inviscid fluid. The ultimate goal is to determine the force exerted by a uniform flow on a Joukowski wing (airfoil). It should be noted that, as in a wind tunnel, the force would be the same if the wing moves with the same velocity in a fluid at rest.

In the absence of shear stresses, forces exerted on a body - the wing in this case - by an inviscid fluid are always normal to the surface and due to the pressure. The force on the body is obtained through the determination of the flow and pressure fields on the surface of the object.

It will be shown that this force is always directed at a right angle with respect to flow (lift), that it is proportional to the fluid velocity far from the body and to the circulation around the profile. The component of the force in the direction of the flow (drag) is always zero. This apparent paradox can only be solved by dropping the hypothesis of irrotational (and inviscid) flow in a region close to the body (boundary layer) where viscosity plays a fundamental role and the flow is rotational and viscous.
Nevertheless, boundary layer theory, not to be considered herein, shows that pressure forces are transported throughout the boundary layer without being altered, if separation is absent. Thus, the information obtained on lift by means of the irrotational theory can be regarded as a good approximation of the actual lift of the wing.

7. Kelvin's Theorem

The circulation along any closed contour *C* inside the fluid is defined as

Stokes's theorem:

where *S* is any surface that has the contour *C* as its edge. (Of course, it is also necessary that the velocity field is defined everywhere on *S*.)

Kelvin's theorem: if

- the closed contour *C* is a material contour (always made up of the same fluid particles);
- the flow is inviscid;
- the flow is barotropic, where the density depends at most on the pressure (not on both pressure and temperature, say);

Then is constant:

This includes incompressible inviscid flows and isentropic inviscid compressible flows.

Proof of the theorem:

8. Incompressible, Irrotational Flow

The basic equations are

(5.1)

and

(5.2)

Equation (5.2) is automatically satisfied by setting and then (5.1) becomes

(5.3)

and the solution to Laplace's equation is required.

9. Superposition of Flows & Elementary Flows

GREEK "Phi" WARNING

Greek Lower Case Letter Phi can be written two ways or (each Internet Browser does it differently). Also, sometimes Upper Case Phi, , and Upper Case Psi, , is used for Velocity Potential and Stream Function, respectively.

It was shown previously that two-dimensional incompressible, inviscid, and irrotational flow can be described by the velocity potential, φ, or stream function, ψ, using the Laplace's equation:

In the following sections, some simple plane potential flows (e.g., uniform flow, source and sink, vortex and doublet) will be introduced. Also, since Laplace's equation is linear, various solutions can be combined to form other solutions. Therefore, some of the real flow problems (e.g., half-body) are obtained by combining these simple plane potential flows using the method of superposition.

Uniform Flow

Uniform flow is the simplest form of potential flow. For flow in a specific direction, the velocity potential is

$$\varphi = U\,(x\cos\alpha + y\sin\alpha)$$

while the stream function is

$$\psi = U\,(y\cos\alpha - x\sin\alpha)$$

[Fig 6: Uniform Flow]

Where α represents the angle between the flow direction and the x-axis (as shown in the figure). Recall that the velocity potential and stream function are related to the component velocity in the 2 dimensional flow field as follows:

Cartesian coordinates:

and

Cylindrical coordinates:

and

Note that the lines of the constant velocity potential (equipotential lines) are orthogonal to the lines of the stream function (streamlines).

Source and Sink

[Fig 7: Source]

When a fluid flows radially outward from a point source, the velocities are

$v_r = m/2\pi r$ and $v_\theta = 0$

Where m is the volume flow rate from the line source per unit length. The velocity potential and stream function can then be represented as

$\varphi = (m/2\pi) \ln(r)$ and $\pi = (m/2\pi)\, \theta$

[Fig 8: Sink]

respectively. When m is negative, the flow is inward, and it represents a sink. The volume flow rate, m, indicates the strength of the source or sink. Note that as r approaches zero, the radial velocity goes to infinity. Hence, the origin represents a singularity. As shown in the figure, the equipotential lines are given by the concentric circles while the streamlines are the radial lines.

Vortex

A vortex can be obtained by reversing the velocity potential and stream functions for a point source such that

$\varphi = K\theta$ and $\psi = -K \ln(r)$

Where K is a constant indicating the strength of the vortex. Now, the equipotential lines are radial lines while the streamlines are given by the concentric circles. The velocities of a vortex are,

[Fig 9: Vortex]

$v_r = 0$ and $v_\theta = K/r$

The strength of a vortex can be described using the concept of circulation (Γ), which is defined as,

Where the integral is taken around a closed arbitrary curve C. For irrotational flows ( ), the circulation becomes

[Fig 10: Vortex Example: Tornado]

However, when the closed curve consists of a singularity point (such as the case of a vortex), the circulation is non-zero,

Doublet

By combining a source and a sink of equal strength using the method of superposition, the stream function is given by

$\psi = \psi_{source} + \psi_{sink} = -(m/2\pi)\ (\theta_1 - \theta_2)$

Through considerable manipulation (i.e., geometric relationships and trigonometric identities), the above equation can be rewritten as

[Fig 11: Superposition of a Source and a Sink]

A doublet is obtained by letting the distance between the source and sink approach zero (i.e., distance "a" tends to zero). The stream function for a doublet becomes

$\psi = -K\sin\theta/r$

Where K (=ma/π) is the strength of the doublet. The streamlines of a doublet are shown in the figure. The corresponding velocity potential is

[Fig 12: Streamlines for a Doublet]

$\varphi = K\cos\theta/r$

For simplicity, the details of the derivation are not given here. Students are encouraged to go through the above derivation process themselves for practice.

Flow around a Half-Body

Flow around a half-body can be obtained by the superposition of a uniform flow with a source. The combined stream function is given by

$\psi = \psi_{\text{uniform flow}} + \psi_{\text{source}} = U\, r \sin\theta + (m/2\pi)\, \theta$

and the corresponding velocity potential is

$\varphi = \varphi_{\text{uniform flow}} + \varphi_{\text{source}} = U\, r \cos\theta + (m/2\pi) \ln(r)$

[Fig 13: Superposition of a Uniform Flow and a Source]

The velocity components are given by

It is interesting to note, the streamlines of this combined function can be used to represent a oval-like shape in a flow stream. The stagnation point of the flow can be used to define the half-bodyshape. The location of the stagnation point can be determined by setting v_r and v_θ equal to zero, yielding

$\theta = \pi$ and $r_{stagnation} = b = m/2\pi U$

The streamline that passes through the stagnation point is then obtained as

$\psi_{stagnation} = m/2 = \pi bU$

By replacing this streamline with a solid boundary, one can then clearly see that flow around a half-body can indeed be represented by the superposition of a uniform flow with a source. The magnitude of the resultant velocity, V, at any point of the flow field is given by

[Fig 14: Flow Past a Oval-type Body]

The shape of the top or bottom half-body is found by putting $\psi_{stagnation}$ value back into the stream function where $\theta = \pi$. This gives,

$\pi bU = Ur\sin\theta = bU\theta$

or

$r = b(\pi - \theta)/\sin\theta$

10. Adding Elementary Flows to Describe Flow around a Cylinder

If a uniform flow is superimposed on a doublet whose axis is parallel to the direction of the uniform flow and is so oriented that the direction of the efflux opposes the uniform flow:

Note that at every point where

Since the velocity is always tangent to a streamline, the fact that velocity component is perpendicular to a circle of r=R means that the circle may be considered as a streamline of the flow field and therefore to a body. The resultant two-dimensional, irrotational (inviscid), incompressible flow is that around a cylinder of radius R whose axis is perpendicular to the freestream direction.

11. Flow around a Cylinder with Circulation

The velocity at the surface of the cylinder is equal to

Thus, the resultant potential function also represents flow around a cylinder. For this flow, however, the streamline pattern away from the surface is not symmetric about the horizontal plane.

- In Section 2 of this course you will cover these topics:
- Viscous Boundary Layers
- Characteristic Parameters For Airfoil And Wing Aerodynamics

- Incompressible Flows Around Airfoils Of Infinite Span

Topic Objective:

At the end of this topic student will be able to understand:

- Boundary Conditions
- Incompressible, Laminar Boundary Layer
- Boundary layer equations
- Incompressible, Turbulent Boundary Layer
- Turbulent boundary layers
- Boundary layer turbine

Definition/Overview:

In physics and fluid mechanics, a boundary layer is that layer of fluid in the immediate vicinity of a bounding surface. In the Earth's atmosphere, the planetary boundary layer is the air layer near the ground affected by diurnal heat, moisture or momentum transfer to or from the surface. On an aircraft wing the boundary layer is the part of the flow close to the wing. The boundary layer effect occurs at the field region in which all changes occur in the flow pattern. The boundary layer distorts surrounding nonviscous flow. It is a phenomenon of viscous forces. This effect is related to the Reynolds number.

Key Points:

1. Boundary Conditions

There are three types of boundary conditions commonly encountered in the solution of partial differential equations:

- Dirichlet boundary conditions specify the value of the function on a surface .
- Neumann boundary conditions specify the normal derivative of the function on a surface,
- Robin boundary conditions. For an elliptic partial differential equation in a region , Robin boundary conditions specify the sum of and the normal derivative of at all points of the boundary of , with and being prescribed.

2. Incompressible, Laminar Boundary Layer

Laminar boundary layers come in various forms and can be loosely classified according to their structure and the circumstances under which they are created. The thin shear layer which develops on an oscillating body is an example of a Stokes boundary layer, whilst the Blasius boundary layer refers to the well-known similarity solution for the steady boundary layer attached to a flat plate held in an oncoming unidirectional flow. When a fluid rotates, viscous forces may be balanced by the Coriolis Effect, rather than convective inertia, leading to the formation of an Ekman layer. Thermal boundary layers also exist in heat transfer. Multiple types of boundary layers can coexist near a surface simultaneously.

3. Boundary layer equations

The deduction of the boundary layer equations was perhaps one of the most important advances in fluid dynamics. Using an order of magnitude analysis, the well-known governing NavierStokes equations of viscous fluid flow can be greatly simplified within the boundary layer.

Notably, the characteristic of the partial differential equations (PDE) becomes parabolic, rather than the elliptical form of the full NavierStokes equations. This greatly simplifies the solution of the equations. By making the boundary layer approximation, the flow is divided into an inviscid portion (which is easy to solve by a number of methods) and the boundary layer, which is governed by an easier to solve PDE. The NavierStokes equations for a two-dimensional steady incompressible flow in Cartesian coordinates are given by

where u and v are the velocity components, ρ is the density, p is the pressure, and ν is the kinematic viscosity of the fluid at a point.

The approximation states that, for a sufficiently high Reynolds number the flow over a surface can be divided into an outer region of inviscid flow unaffected by viscosity (the majority of the flow), and a region close to the surface where viscosity is important (the boundary layer). Let u and v be streamwise and transverse (wall normal) velocities respectively inside the boundary layer. Using scale analysis, it can be shown that the above equations of motion reduce within the boundary layer to become

and if the fluid is incompressible (as liquids are under standard conditions):

The asymptotic analysis also shows that v, the wall normal velocity, is small compared with u the streamwise velocity, and that variations in properties in the streamwise direction are generally much lower than those in the wall normal direction.

Since the static pressure p is independent of y, then pressure at the edge of the boundary layer is the pressure throughout the boundary layer at a given streamwise position. The external pressure may be obtained through an application of Bernoulli's equation. Let u_0 be the fluid velocity outside the boundary layer, where u and u_0 are both parallel. This gives upon substituting for p the following result

With the boundary condition

For a flow in which the static pressure p also does not change in the direction of the flow then

So u_0 remains constant.
Therefore, the equation of motion simplifies to become

These approximations are used in a variety of practical flow problems of scientific and engineering interest. The above analysis is for any instantaneous laminar or turbulent boundary layer, but is used mainly in laminar flow studies since the mean flow is also the instantaneous flow because there are no velocity fluctuations present.

4. Incompressible, Turbulent Boundary Layer

The treatment of turbulent boundary layers is far more difficult due to the time-dependent variation of the flow properties. One of the most widely used techniques in which turbulent flows are tackled is to apply Reynolds decomposition. Here the instantaneous flow properties are decomposed into a mean and fluctuating component. Applying this technique to the boundary layer equations gives the full turbulent boundary layer equations not often given in literature:

Using the same order-of-magnitude analysis as for the instantaneous equations, these turbulent boundary layer equations generally reduce to become in their classical form:

The additional term in the turbulent boundary layer equations is known as the Reynolds shear stress and is unknown a priori. The solution of the turbulent boundary layer equations therefore necessitates the use of a turbulence model, which aims to express the Reynolds shear stress in terms of known flow variables or derivatives. The lack of accuracy and generality of such models is the single major obstacle which inhibits the successful prediction of turbulent flow properties in modern fluid dynamics.

5. Turbulent boundary layers

The treatment of turbulent boundary layers is far more difficult due to the time-dependent variation of the flow properties. One of the most widely used techniques in which turbulent flows are tackled is to apply Reynolds decomposition. Here the instantaneous flow properties are decomposed into a mean and fluctuating component. Applying this technique to the boundary layer equations gives the full turbulent boundary layer equations not often given in literature:

Using the same order-of-magnitude analysis as for the instantaneous equations, these turbulent boundary layer equations generally reduce to become in their classical form:

The additional term in the turbulent boundary layer equations is known as the Reynolds shear stress and is unknown a priori. The solution of the turbulent boundary layer equations therefore necessitates the use of a turbulence model, which aims to express the Reynolds shear stress in terms of known flow variables or derivatives. The lack of accuracy and generality of such models is the single major obstacle which inhibits the successful prediction of turbulent flow properties in modern fluid dynamics.

6. Boundary layer turbine

This effect was exploited in the Tesla turbine, patented by Nikola Tesla in 1913. It is referred to as a bladeless turbine because it uses the boundary layer effect and not a fluid impinging upon the blades as in a conventional turbine. Boundary layer turbines are also known as cohesion-type turbine, bladeless turbine, and Prandtl layer turbine (after Ludwig Prandtl).

Topic Objective:

At the end of this topic student will be able to understand:

- Airfoil Geometry Parameters
- Wing-Geometry Parameters
- Wings of Finite Span.

Definition/Overview:

An airfoil (in American English) or aerofoil (in British English) is the shape of a wing or blade (of a propeller, rotor or turbine) or sail as seen in cross-section. An airfoil-shaped body moved through a fluid produces a force perpendicular to the motion called lift. Subsonic flight airfoils have a characteristic shape with a rounded leading edge, followed by a sharp trailing edge, often with asymmetric camber. Airfoils designed with water as the working fluid are also called hydrofoils.

Key Points:

1. Airfoil Geometry Parameters

Airfoil geometry can be characterized by the coordinates of the upper and lower surface. It is often summarized by a few parameters such as: maximum thickness, maximum camber, position of max thickness, position of max camber, and nose radius. One can generate a reasonable airfoil section given these parameters. This was done by Eastman Jacobs in the early 1930's to create a family of airfoils known as the NACA Sections.

The NACA 4 digit and 5 digit airfoils were created by superimposing a simple meanline shape with a thickness distribution that was obtained by fitting a couple of popular airfoils of the time:

$$y = (t/0.2) * (.2969*x^{0.5} - .126*x - .3537*x^2 + .2843*x^3 - .1015*x^4)$$

The camberline of 4-digit sections was defined as a parabola from the leading edge to the position of maximum camber, then another parabola back to the trailing edge.

NACA 4-Digit Series:

4	4	12
max camber in % chord	position of max camber in 1/10 of c	max thickness in % of chord

After the 4-digit sections came the 5-digit sections such as the famous NACA 23012. These sections had the same thickness distribution, but used a camberline with more curvature near the nose. A cubic was faired into a straight line for the 5-digit sections.

NACA 5-Digit Series:

2 3 0 1 2

Approx max position max thickness

Camber of max camber in % of chord in % chord in 2/100 of c

The 6-series of NACA airfoils departed from this simply-defined family. These sections were generated from a more or less prescribed pressure distribution and were meant to achieve some laminar flow.

NACA 6-Digit Series:

6 3, 2 - 2 1 2

Six- location half width ideal Cl max thickness

Series of min Cp of low drag in tenths in % of chord

In 1/10 chord bucket in 1/10 of Cl

After the six-series sections, airfoil design became much more specialized for the particular application. Airfoils with good transonic performance, good maximum lift capability, very thick sections, and very low drag sections are now designed for each use. Often a wing design begins with the definition of several airfoil sections and then the entire geometry is modified based on its 3-dimensional characteristics.

2. Wing-Geometry Parameters

The wing geometry may be specified in several ways. This section defines a few commonly used terms and how to compute them.

2.1. Wing Areas

The definition of wing area is not obvious and different companies define the areas differently. Here, we always take the reference wing area to be that of the trapezoidal portion of the wing projected into the centerline. The leading and trailing edge chord extensions are not included in this definition and for some airplanes, such as Boeing's Blended Wing Body, the difference can be almost a factor of two between the "real" wing area and the "trap area". Some companies use reference wing areas that include portions of the chord extensions, and in some studies, even tail area is included as part of the reference area. For simplicity, we use the trapezoidal area in this text.

[Fig 1: Reference Wing Area] [Fig 2: Exposed Wing Area] [Fig 3: Area Affected by Flaps]

In addition to the reference area, we use the exposed planform area depicted above in the calculation of skin friction drag and the wetted area which is a bit more than twice the exposed planform area.

2.2. Wing Span and Aspect Ratio

Of all the parameters that might be defined without a footnote, span seems to be the most unambiguous; however, even this is not so clear. The small effect of wing bending on the geometric span can become very measurable when the wing includes winglets. We ignore the differences here, but suggest that a reference span should be measured on the ground with a prescribed fuel load since this is the only condition in which it may be conveniently verified.

Aspect ratio is often used in place of the dimensional span in many of the aerodynamic equations of interest. Aspect ratio, or AR, is roughly the ratio of span to average wing chord. It may be computed by: $AR = b^2 / S_{ref}$. It is important that the same definition of reference area be used in the definition of aspect ratio as is used in the definition of coefficients such as C_L and C_D.

2.3. Reference Lengths

Various wing reference lengths are used in aerodynamic computations. One of the most important of these is the mean aerodynamic chord, or M.A.C.. The M.A.C. is the chord-weighted average chord length of the wing, defined as:

For a linearly tapered (trapezoidal) wing, this integral is equal to:

$$M.A.C. = 2/3 (C_{root} + C_{tip} - C_{root} C_{tip} / (C_{root}+C_{tip}))$$

For wings with chord extensions, the MAC may be computed by evaluating the MAC of each linearly-tapered portion then taking an average, weighted by the area of each portion. In many cases, however, the MAC of the reference trapezoidal wing is used. The M.A.C. is often used in the nondimensionalization of pitching moments. The M.A.C. of just the exposed area is also used to compute the reference length for calculation of Reynolds number as part of the wing drag estimation. The M.A.C. is chosen instead of the simpler mean geometric chord for quantities whose values are weighted more strongly by local chord that is reflected by their contribution to the area.

3. Wings of Finite Span.

The airflow past a finite span wing generates downstream vortices at the wing trailing edge and extremities. For specific wing geometry, the lift and the induced drag depend on the span wise variation of the circulation, which is related to the bound and free vortices. The wings of minimum induced drag are those corresponding to an elliptical variation of the circulation dong the span. In the wing design, by altering the wing twist and plan form geometry, a desired span wise variation of the circulation can be achieved. Hence, the calculation of the span wise variation of the circulation is very important when wing aerodynamic characteristics have to be specified.

Topic Objective:

At the end of this topic student will be able to understand:

- Circulation and the Generation of Lift
- General Thin-Airfoil Theory
- Thin airfoil theory
- Derivation of thin airfoil theory
- Physical description of lift on an airfoil
- Lift in an established flow
- Stages of lift production

Definition/Overview:

In fluid mechanics or more generally continuum mechanics, an incompressible flow is solid or fluid flow in which the divergence of velocity is zero. This is more precisely termed isochoric flow. It is an idealization used to simplify analysis. In reality, all materials are compressible to some extent. Note that isochoric refers to flow, not the material property. This means that under certain circumstances, a compressible material can undergo (nearly) incompressible flow. However, by making the 'incompressible' assumption, the governing equations of material flow can be simplified significantly.

Key Points:

1. Circulation and the Generation of Lift

In the context of a fluid flow relative to a body, the lift force is the component of the aerodynamic force that is perpendicular to the flow direction. It contrasts with the drag force, which is the parallel component of the aerodynamic force. Lift is commonly associated with the wing of an aircraft, although lift is also generated by rotors on helicopters, sails and keels on sailboats, hydrofoils, wing on auto racing cars, and wind turbines. While common meanings of the word "lift" suggest that lift opposes gravity, aerodynamic lift can be in any direction. When an aircraft is in cruise for example, lift does oppose gravity. However, when the aircraft is climbing, descending, or banking in a turn, for example, the lift is tilted with respect to the vertical. Lift may also be entirely downwards in some aerobatic manoeuvres, or on the wing on a racing car. In this last case, the term downforce is often used. The mathematical equations describing the generation of lift forces have been well established since the Wright Brothers experimentally determined a reasonably precise value for Smeaton's Smeaton coefficient more than 100 years ago, but the practical explanation of what those equations mean is still controversial, with persistent misinformation and pervasive misunderstanding.

2. General Thin-Airfoil Theory

The analysis methods of low-speed aerodynamics start with the idea that the effect of a lifting body on a flowfield can be simulated by solving the Laplace equation for the right boundary conditions. Thus we will see that the effects of an airfoil section can be simulated if the airfoil is replaced by a "vortex sheet" combined with the freestream. We will adjust the strength of the vortex sheet as needed to ensure that the flow remains tangential along the contours of our desired airfoil (i.e., the solid surface is satisfactority simulated by specifying that the flow cannot cross the contour of the surface. In its simplest form, the airfoil can be represented by a vortex sheet along the line equidistant from the upper and lower surfaces. The unknown then will be the strength of the vortex sheet required at each point along this mean "camber" line (defined below). In other words,

3. Thin airfoil theory

Thin airfoil theory is a simple theory of airfoils that relates angle of attack to lift. It was devised by German mathematician Max Munk and further refined by British aerodynamicist Hermann Glauert and others in the 1920s. The theory idealizes the flow around an airfoil as two-dimensional flow around a thin airfoil. It can be imagined as addressing an airfoil of zero thickness and infinite wingspan.

Thin airfoil theory was particularly notable in its day because it provided a sound theoretical basis for the following important properties of airfoils in two-dimensional flow:

- on a symmetric airfoil, the center of pressure lies exactly one quarter of the chord behind the leading edge
- on a cambered airfoil, the aerodynamic center lies exactly one quarter of the chord behind the leading edge
- the slope of the *lift coefficient versus angle of attack* line is units per radian

As a consequence of (3), the section lift coefficient of a symmetric airfoil of infinite wingspan is:

where is the section lift coefficient, is the angle of attack in radians, measured relative to the chord line. (The above expression is also applicable to a cambered airfoil where is the angle of attack measured relative to the zero-lift line instead of the chord line.)

Also as a consequence of (3), the section lift coefficient of a cambered airfoil of infinite wingspan is:

Where is the section lift coefficient when the angle of attack is zero.

Thin airfoil theory does not account for the stall of the airfoil which usually occurs at an angle of attack between 10 and 15 for typical airfoils.

4. Derivation of thin airfoil theory

The airfoil is modeled as a thin lifting mean-line (camber line). The mean-line, y(x), is considered to produce a distribution of vorticity γ(*s*) along the line, s. By the Kutta condition, the vorticity is zero at the trailing edge. Since the airfoil is thin, x (chord position) can be used instead of s, and all angles can be approximated as small.

From the Biot-Savart law, this vorticity produces a flow field *w*(*s*) where

Where x is the location at which induced velocity is produced, x' is the location of the vortex element producing the velocity and c is the chord length of the airfoil.

Since there is no flow normal to the curved surface of the airfoil, w(x) balances that from the component of main flow V which is locally normal to the plate the main flow is locally inclined to the plate by an angle $\alpha - dy / dx$. That is

This integral equation can by solved for $\gamma(x)$, after replacing x by

,

as a Fourier series in $A_n sin(n\theta)$ with a modified lead term $A_0(1 + cos(\theta)) / sin(\theta)$

That is

(These terms are known as the Glauert integral).

The coefficients are given by

And

By the KuttaJoukowski theorem, the total lift force F is proportional to

and its moment M about the leading edge to

The calculated Lift coefficient depends only on the first two terms of the Fourier series, as

The moment M about the leading edge depends only on A_0,A_1 and A_2 , as

The moment about the 1/4 chord point will thus be,

.

From this it follows that the center of pressure is aft of the 'quarter-chord' point 0.25 c, by

The aerodynamic center, AC, is at the quarter-chord point. The AC is where the pitching moment M' does not *vary* with angle of attack, i.e.

5. Physical description of lift on an airfoil

Lift is generated in accordance with the fundamental principles of physics such as Newton's laws of motion, Bernoulli's principle, conservation of mass and the balance of momentum (where the last is the fluid dynamics version of Newton's second law). Each of these principles can be used to explain lift on an airfoil. As a result, there are numerous different explanations with different levels of rigor and complexity. For example, there is an explanation based on Newtons laws of motion; and an explanation based on Bernoullis principle. Neither of these explanations is incorrect, but each appeals to a different audience.

To attempt a physical explanation of lift as it applies to an airplane, consider the flow around a 2-D, symmetric airfoil at positive angle of attack in a uniform free stream. Instead of considering the case where an airfoil moves through a fluid as seen by a stationary observer, it is equivalent and simpler to consider the picture when the observer follows the airfoil and the fluid moves past it.

6. Lift in an established flow

If one assumes the shape of the flow around an airfoil (say, from experiment or observation), then the explanation of lift is rather simple and can be explained primarily in terms of pressures using Bernoulli's principle (which can be derived from Newton's second law) and conservation of mass, following the development by John D. Anderson in *Introduction to Flight*.

The image to the right shows the streamlines over a NACA 0012 airfoil computed using potential flow theory, a simplified model of the real flow. The flow approaching an airfoil can be divided into two *streamtubes*, which are defined based on the area between two streamlines. By definition, fluid never crosses a streamline; hence mass is conserved within each streamtube. One streamtube travels over the upper surface, while the other travels over the lower surface; dividing these two tubes is a dividing line (the stagnation streamlines) that intersects the airfoil on the lower surface, typically near to the leading edge. The stagnation streamline leaves the airfoil at the sharp trailing edge, a feature of the flow known as the Kutta condition.

The upper stream tube constricts as it flows up and around the airfoil, the so-called upwash. From the conservation of mass, the flow speed must increase as the stream tube area decreases. The area of the lower stream tube increases, causing the flow inside the tube to slow down. It is typically the case that the flow traveling over the upper surface will reach the trailing edge before that traveling over the bottom.
From Bernoulli's principle, the pressure on the upper surface where the flow is moving faster is lower than the pressure on the lower surface. The pressure difference thus creates a net aerodynamic force, pointing upward and downstream to the flow direction. The component of the force normal to the free stream is considered to be lift; the component parallel to the free stream is drag. In conjunction with this force by the air on the airfoil, by Newton's third law, the airfoil imparts an equal-and-opposite force on the surrounding air that creates the downwash. Measuring the momentum transferred to the downwash is another way to determine the amount of lift on the airfoil.

7. Stages of lift production

In attempting to explain why the flow follows the upper surface of the airfoil, the situation gets considerably more complex. To offer a more complete physical picture of lift, consider the case of an airfoil accelerating from rest in a viscous flow. Lift depends entirely on the nature of viscous flow past certain bodies: in inviscid flow (i.e. assuming that viscous forces are negligible in comparison to inertial forces), there is no lift without imposing a net circulation, the proper amount of which can be determined by applying the Kutta condition. In a viscous flow like in the physical world, however, the lift and other properties arise naturally as described here.

When there is no flow, there is no lift and the forces acting on the airfoil are zero. At the instant when the flow is turned on, the flow is undeflected downstream of the airfoil and there are two stagnation points on the airfoil (where the flow velocity is zero): one near the leading edge on the bottom surface, and another on the upper surface near the trailing edge. The dividing line between the upper and lower streamtubes mentioned above intersects the body at the stagnation points. Since the flow speed is zero at these points, by Bernoulli's principle the static pressure at these points is at a maximum. As long as the second stagnation point is at its initial location on the upper surface of the wing, the circulation around the airfoil is zero and, in accordance with the KuttaJoukowski theorem, there is no lift. The net pressure difference between the upper and lower surfaces is zero.

The effects of viscosity are contained within a thin layer of fluid called the boundary layer, close to the body. As flow over the airfoil commences, the flow along the lower surface turns at the sharp trailing edge and flows along the upper surface towards the upper stagnation point. The flow in the vicinity of the sharp trailing edge is very fast and the resulting viscous forces cause the boundary layer to accumulate into a vortex on the upper side of the airfoil between the trailing edge and the upper stagnation point. This is called the starting vortex. The starting vortex and the bound vortex around the surface of the wing are two halves of a closed loop. As the starting vortex increases in strength the bound vortex also strengthens, causing the flow over the upper surface of the airfoil to accelerate and drive the upper stagnation point towards the sharp trailing edge. As this happens, the starting vortex is shed into the wake, and is a necessary condition to produce lift on an airfoil. If the flow were stopped, there would be a corresponding "stopping vortex". Despite being an idealization of the real world, the vortex system set up around a wing is both real and observable; the trailing vortex sheet most noticeably rolls up into wing-tip vortices. The upper stagnation point continues moving downstream until it is coincident with the sharp trailing edge (as stated by the Kutta condition). The flow downstream of the airfoil is deflected downward from the free-stream direction and, from the reasoning above in the basic explanation, there is now a net pressure difference between the upper and lower surfaces and an aerodynamic force is generated.

- In Section 3 of this course you will cover these topics:
- Incompressible Flows About Wings Of Finite Span
- Dynamics Of A Compressible Flow Field
- Compressible, Subsonic Flows And Transonic Flows

Topic Objective:

At the end of this topic student will be able to understand:

- Vortex System
- Lifting-Line Theory for Unswept Wings
- Panel Methods
- Vortex Lattice Method
- Factors Affecting Drag Due-to-Lift at Subsonic Speeds.
- Delta Wings
- Leading-Edge Extensions

Definition/Overview:

One may apply the results of 3-D potential theory in several ways. We first consider the theory of finite wings. It might be right to start out by saying that each section of a finite wing behaves as described by our 2-D analysis. If this were true then we would still find that the lift curve slope was 2p per radian that the drag was 0, and the distribution of lift would vary as the distribution of chord. Unfortunately, things do not work this way. There are several reasons for this:

One explanation is that the high pressure on the lower surface of the wing and the low pressure on the upper surface causes the air to leak around the tips, causing a reduction in the pressure difference in the tip regions. In fact, the lift must go to zero at the tips because of this effect. We will next see how and why we must model the 3-D wing differently from 2-D.

f we were to take the naive view that the 2-D model would work in 3-D, we might have the picture shown on the right. If each section had the distribution of vorticity along its chord that it had in 2-D, the lift would be proportional to the chord, and would not drop off at the tips as we know it must.

This sort of model does not conform to our physical picture of what happens at the wing tips. And indeed, it does not satisfy the equations of 3-D fluid flow. The reason that this does not work is that in this case the streamlines are not confined to a plane. They move in 3-D and the flow pattern is quite different.

Key Points:

1. A vortex system

A vortex system is for separating particles of varied size and density from a liquid stream. A particle bearing liquid is introduced tangentially into one end of an elongated cylindrical chamber to form a rotating liquid vortex therein. An impeller at one end of the chamber reinforces the vortex flow. Either a series of tangentially oriented jets introducing a gas or a gas saturated liquid through the chamber wall or a rotating central chamber section further increases the vortex rotational flow. The vortex produces a heavy particle rich outer region, a gas and light particle rich axial region and a cleaned liquid annular region there between. The cleaned liquid flows out of the chamber through an outlet at the chamber end opposite end at which the particle bearing liquid is introduced. The heavy particles in the outer region and the gas and light particles in the axial region are each removed through an outlet. A variety of component orientations and auxiliary devices may be used to optimize system operation.

2. Lifting-Line Theory for Unswept Wings

Lifting-line theory or Lanchester-Prandtl wing theory was published by Ludwig Prandtl in 19181919 after working with Albert Betz and Max Munk on the problem of a useful mathematical tool for examining lift from "real world" wings.

In this model, the vortex strength reduces along the wingspan, and the loss in vortex strength is shed as a vortex-sheet from the trailing edge, rather than just at the wing-tips.

The theory states that:

where

- is the lift coefficient,
- is the 2D lift coefficient slope (see Thin Airfoil Theory),
- is the Aspect Ratio, and
- is the angle of attack in radians.

The theoretical maximum for is 2π. Note that this equation becomes the Thin Airfoil equation if *AR* goes to infinity.

Lifting-line theory also states an equation for induced drag:.

Where

- is the drag coefficient for induced drag,
- is the lift coefficient, and
- is the Aspect Ratio.

3. Panel Methods

The Vortex Panel Method Applet is a computational tool for students studying the aerodynamics of airfoil sections. The vortex-panel method is a method for computing ideal flows - flows in which the effects of compressibility and viscosity are negligible. Ideal flow is often the first type of fluid motion that student engineers and scientists study, because it is the simplest. Large parts of the flows past ships, submarines, cars and light aircraft are closely ideal.

Ideal fluid flows are solutions to Laplace's equation. This differential equation is linear, which means that adding together (superposing) any number of ideal flows produces a new ideal flow. One approach to finding the solution to complex flow problems - termed superposition - is therefore to begin with very simple ideal flows, that are easily understood and described, and then to add them together to produce the complex flow patterns desired. This process is modeled in its most simplest form in the Ideal Flow Machine applet in which elementary flows such as sources, vortices and uniform flows can be combined and the resulting flows visualized. The Vortex Panel Method applet goes one step further, and applies the principle of superposition to a practical application - the analysis of airfoil flows.

The vortex panel method (see Kuethe and Chow for example) models the flow past an airfoil as the summation of a uniform flow (same speed and direction everywhere) and a series of vortex 'panels' (or 'sheets') arranged to form a closed polygon with a shape that approximates, as nearly as possible, the actual curved shape of the airfoil, see figure 1. The term panel is rather misleading since it conjures up a mental image of a solid plate, through which the flow cannot pass. This is not the case as is illustrated in figures 2 and 3. Figure 2 shows the flow produced by adding a uniform flow to a panel of constant strength and figure 3 shows the flow produced by the panel alone. Note that both these figures were produced with the Ideal Flow Machine.

As can be seen in figure 3, each panel produces an elongated rotating flow - the panel is actually a concentrated sheet of vorticity. In general the strength of a panel can vary along its length in an arbitrary way. In this realization of the vortex panel method we assume that the strength of each panel varies linearly over the panel and the strength is continuous across the panel joints, except at the trailing edge (see figure 1). This means that, after drawing the N panels around the airfoil, we have N+1 strengths to specify (one at each panel joint and two at the trailing edge, the variations being then linear across each panel). The problem is to choose these strengths so that the flow past the panels is a realistic representation of the flow over the airfoil. To do this we invoke the so-called 'non-penetration' condition - the condition that the flow cannot pass through the surface of the airfoil or, equivalently, that the component of velocity perpendicular to the airfoil surface is zero. We apply this condition approximately by writing equations for the velocity generated by all the panels, plus the free stream, at the central point of each panel (called the control points, see figure 1). We then set the component of this velocity perpendicular to the panel equal to zero. Since we have N control points this gives us N equations for N+1 strengths. The final equation is obtained using the Kutta condition. The Kutta condition encapsulates the observation that the flow can't go around the trailing edge, but must leave the airfoil there. This is a consequence of viscous effects, which are otherwise is absent from this calculation. For the Kutta condition to be satisfied the strengths of the vortex panels must be equal and opposite where they meet at the trailing-edge joint. All that now remains is the solution of N+1 simultaneous equations for the $N+1$ unknown strengths (via a matrix inversion), and then the evaluation of the flow properties of interest. This is precisely what is done by the Vortex Panel Method Applet.

4. Vortex lattice method

The Vortex lattice method, (VLM), is a numerical, Computational fluid dynamics, method used mainly in the early stages of aircraft design and in aerodynamic education at university level. The VLM models the lifting surfaces, such as a wing, of an aircraft as a infinitely thin sheet of discrete vortices to compute lift and induced drag. The influence of the thickness, viscosity and other things, is neglected. VLMs can compute the flow around a wing with rudimentary geometrical definition. For a rectangular wing it is enough to know the span and chord. On the other side of the spectrum, they can describe the flow around a fairly complex aircraft geometry (with multiple lifting surfaces with taper, kinks, twist, camber, trailing edge control surfaces and many other geometric features). By simulating the flow field, one can extract the pressure distribution or as in the case of the VLM, the force distribution, around the simulated body. This knowledge is then used to compute the aerodynamic coefficients and their derivatives that are important for assessing the aircraft's handling qualities in the conceptual design phase. With an initial estimate of the pressure distribution on the wing, the structural designers can start designing the load bearing parts of the wings, fin and tailplane and other lifting surfaces. Additionally, while the VLM cannot compute the viscous drag, the induced drag stemming from the production of lift can be estimated. Hence as the drag must be balanced with the thrust in the cruise configuration, the propulsion group can also get important data from the VLM simulation.

5. Factors Affecting Drag Due-to-Lift at Subsonic Speeds.

Because of the highly swept and relatively sharp leading edges for wing designs of interest for supersonic aircraft such as the proposed supersonic transport, a knowledge of the various factors affecting the drag due to lift at subsonic speeds is becoming increasingly necessary. In view of this interest, an investigation was conducted to determine the effects of Reynolds number, lift coefficient, and wing leading-edge radius on the drag due to lift of a series of thin, highly swept, low-aspect-ratio wing-body configurations. The purpose of this discussion is to review some of the results of this investigation.

6. Delta Wings

The delta wing is a wing planform in the form of a triangle, named after the Greek uppercase delta which is a triangle (Δ). Its use in the so called "tailless delta", i.e. without the horizontal tailplane, was pioneered especially by Alexander Lippisch in Germany prior to WWII, although none of his designs saw widespread service. After the war he moved to the United States where he worked at Convair. After the war the tailless delta became the favoured design for high-speed use, and was used (almost to the exclusion of other planforms) by Convair and Dassault in France. A number of British designs also used the delta, including the Avro Vulcan bomber. This early use of tailless delta wing aircraft was augmented by the tailed delta configuration created in the TsAGI (Central Aero and Hydrodynamic Institute, Moscow), taking advantage of both high angle-of-attack (i.e., manoeuvre) capability and high speeds. It was used on the MiG-21 (Fishbed) and Sukhoi Su-9/Su-11/15 fighters, built in tens of thousands.

More recently, with the advent of aircraft with relaxed or no natural stability, and the therefore necessary computer controlled/assisted control systems (fly-by-wire, or FBW), the horizontal control surfaces are often moved forward to become a canard in front of the wing to control the aeroplane as the normal elevator does. This favorably modifies the airflow over the wing, most notably during lower altitude flight. In contrast to the classic tail-mounted elevators, the canards add to the total lift, enabling the execution of extreme maneuvers, improving low-speed handling, lowering the landing speed, or the marked reduction of drag. An example of a canard-equipped delta-winged aircraft is the Tu-144.

The primary advantage of the delta wing design is that the wing's leading edge remains behind the shock wave generated by the nose of the aircraft when flying at supersonic speeds, which is an improvement on traditional wing designs. While this is also true of highly swept wings, the delta's planform carries across the entire aircraft, allowing it to be built much more strongly than a swept wing, where the spar meets the fuselage far in front of the center of gravity. Generally a delta will be stronger than a similar swept wing, as well as having much more internal volume for fuel and other storage.

Another advantage is that as the angle of attack increases the leading edge of the wing generates a vortex which remains attached to the upper surface of the wing, giving the delta a very high stall angle. A normal wing built for high speed use is typically dangerous at low speeds, but in this regime the delta changes over to a mode of lift based on the vortex it generates. The disadvantages, especially marked in the older tailless delta designs, are a loss of total available lift caused by turning up the wing trailing edge or the control surfaces (as required to achieve a sufficient stability) and the high induced drag of this low-aspect ratio type of wing. This causes delta-winged aircraft to 'bleed off' energy very rapidly in turns, a disadvantage in aerial maneuver combat and dog-fighting.

Additional advantages of the delta wing are simplicity of manufacture, strength, and substantial interior volume for fuel or other equipment. Because the delta wing is simple, it can be made very robust (even if it is quite thin), and it is easy and relatively inexpensive to build - a substantial factor in the success of the MiG-21 and Mirage aircraft.

Alexander Lippisch, Frenchman Payen, and the DFS (German Institute of Flight) studied a number of ramjet powered (sometimes coal-fueled) delta-wing interceptor aircraft during the war, one progressing as far as a glider prototype.prototype test footage After the war, Lippisch was taken to the US, where he worked at Convair. The Convair engineers became very interested in his interceptor designs, and started work on a larger version known as the F-92. This project was eventually cancelled as impractical, but a prototype flying test-bed was almost complete by that point, and was later flown as the XF-92. The design generated intense interest around the world. Soon many aircraft designs, particularly interceptors, were designed around a delta wing.

Pure deltas fell out of favors somewhat due to their undesirable characteristics, notably flow separation at high angles of attack (swept wings have similar problems), and high drag at low altitudes. This limited them primarily to high-speed, high-altitude interceptor roles. Some modern aircraft, like the F-16, use a cropped delta along with horizontal tail surfaces. A modification, the compound delta such as seen on the Saab Draken fighter or the prototype F-16XL "Cranked Arrow", or the ogee delta used on the Anglo-French Concorde Mach 2 airliner, connected another much more highly swept piece of the delta wing to the forward root section of the main one, to create the high-lift vortex in a more controlled fashion, reduce the drag and thereby allow for landing the delta at acceptably slow speed.

As the performance of jet engines grew, fighters with other planforms could perform as well as deltas, and do so while maneuvering much harder and at a wider range of altitudes. Today a remnant of the compound delta can be found on most fighter aircraft, in the form of leading edge extensions. These are effectively very small delta wings placed so they remain parallel to the airflow in cruising flight, but start to generate a vortex at high angles of attack. The vortex is then captured on the top of the wing to provide additional lift, thereby combining the delta's high-alpha performance with a conventional highly efficient wing planform. Many modern fighter aircraft, such as the JAS 39 Gripen and the Eurofighter Typhoon use a combination of canards and a delta wing.

7. Leading-Edge Extensions

Leading edge extensions or LEX (also referred to as leading edge root extensions or LERX or strakes or chines) are fillets added to the front of a modern fighter aircraft's wings in order to provide usable airflow at high angles of attack. They are typically roughly triangular in shape, running from the leading edge of the wing root to a point near the cockpit along the fuselage. They tend to be fairly small in span, extending out less than a metre. In effect, they are small delta wings grafted onto the front of the normal wings. In cruising flight the effect of the LEX is minimal. However: when the angle of attack increases, as in a dog fight, the LEX starts to generate a high-speed vortex that remains attached to the top of the wing. Due to the effects described by Bernoulli's principle the wing therefore has a low pressure zone on top, and continues to generate lift past the normal stall point. The F/A-18 Hornet has especially large examples of LEX, as does the Sukhoi Su-27. Early prototypes of the Su-27 crashed due to poorly designed LEX, causing it to freeze at angles of attack above 5 degrees. This has since been overcome. In fact, the LEX help in making advanced maneuvers such as the Pugachev's Cobra, the Cobra Turn and the Kulbit possible.

Topic Objective:

At the end of this topic student will be able to understand:

- Thermodynamic Concepts
- Adiabatic Flow in a Variable-Area Streamtube
- Shock Waves

Definition/Overview:

Laws of thermodynamics, which postulate that energy, can be exchanged between physical systems as heat or work.

Key Points:

1. Thermodynamic Concepts

- Describes processes that involve changes in temperature, transformation of energy, relationships between heat and work.

- It is a science, and more importantly an engineering tool, that is necessary for describing the performance of propulsion systems, power generation systems, refrigerators, fluid flow, combustion, ...
- Generalization of extensive empirical evidence (however most thermodynamic principles and can be derived from kinetic theory)
- Examples of heat engines

- Concept of a thermodynamic system
  - A quantity of matter of fixed identity, boundaries may be fixed or movable, can transfer heat and work across boundary but not mass
  - Identifiable volume with steady flow in and out, a control volume. Often more useful way to view devices such as engines

2. Adiabatic Flow

To begin the quantitative study of compressible flow, we first consider those flows that are steady, inviscid, and adiabatic. A quick review of these three terms will provide some clarification about the type of flow we are considering.

- Steady - a flow in which the properties at a given point do not change in time
- Inviscid - effects of friction are neglected
- Adiabatic - a process in which no heat is added or taken away from the flow

These approximations suit many flows well.

The energy equation for steady, inviscid adiabatic flow can be written

where ho is the stagnation or total enthalpy. The equation above states that the stagnation or total enthalpy (h_o) is a constant and represents the sum of the enthalpy (h) and the kinetic energy per unit mass ($V^2/2$). Therefore, if the flow velocity (u) increases, then the enthalpy (h) must decrease and vice-versa.

For a calorically perfect gas, the above equation can be expressed as a function of temperature. In its rearranged form,

where T_o is the stagnation or total temperature and T is the static temperature where the local Mach number is M. This important equation shows that the ratio of the total temperature to the local, static temperature at any point in the flow is dependent only upon the Mach number (M) for a fixed ratio of specific heats ( ). This equation holds for all steady, adiabatic flows. This means it is valid for reversible (isentropic) and non-reversible flows.

3. Shock Waves

A shock wave (also called shock front or simply "shock") is a type of propagating disturbance. Like an ordinary wave, it carries energy and can propagate through a medium (solid, liquid or gas) or in some cases in the absence of a material medium, through a field such as the electromagnetic field. Shock waves are characterized by an abrupt, nearly discontinuous change in the characteristics of the medium. Across a shock there is always an extremely rapid rise in pressure, temperature and density of the flow. In supersonic flows, expansion is achieved through an expansion fan. A shock wave travels through most media at a higher speed than an ordinary wave. Unlike solitons (another kind of nonlinear wave), the energy of a shock wave dissipates relatively quickly with distance. Also, the accompanying expansion wave approaches and eventually merges with the shock wave, partially cancelling it out. Thus the sonic boom associated with the passage of a supersonic aircraft is the sound wave resulting from the degradation and merging of the shock wave and the expansion wave produced by the aircraft.

When a shock wave passes through matter, the total energy is preserved but the energy which can be extracted as work decreases and entropy increases. This for example creates additional drag force on aircraft with shocks.

Topic Objective:

At the end of this topic student will be able to understand:

- Compressible, Subsonic Flow
- Transonic Flow Past Unswept Airfoils
- Swept Wings at Transonic Speeds
- Forward Swept Wing
- Transonic Aircraft

Definition/Overview:

A flow in which the Mach number is less than one everywhere is known as subsonic flow. Typically, if the freestream Mach number is less than about 0.8 then the flow is usually subsonic everywhere. However in some situations like flow over a highly cambered airfoil, the Mach number may locally exceed unity due to acceleration of the fluid in the curved regions. Subsonic flows are characterized by the absence of shocks which appear only in supersonic flows.

Transonic is an aeronautics term referring to a range of velocities just below and above the speed of sound (about mach 0.81.2). It is defined as the range of speeds between the critical mach number, when some parts of the airflow over an aircraft become supersonic, and a higher speed, typically near Mach 1.2, when all of the airflow is supersonic. Between these speeds some of the airflow is supersonic, and some is not.

Most modern jet powered aircraft spend a considerable amount of time in the transonic state. This is particularly important due to an effect known as wave drag, which is prevalent in these speed ranges. Attempts to combat wave drag can be seen on all high-speed aircraft; most notable is the use of swept wings, but another common form is a wasp-waist fuselage as a side effect of the Whitcomb area rule.

Severe instability can occur at transonic speeds. Shock waves move through the air at the speed of sound. When an object such as an aircraft also moves at the speed of sound, these shock waves build up in front of it to form a single, very large shock wave. During transonic flight, the plane must pass through this large shock wave, as well as contending with the instability caused by air moving faster than sound over parts of the wing and slower in other parts. The difference in speed is due to Bernoulli's principle.

Transonic speeds can also occur at the tips of rotor blades of helicopters and aircraft. However, as this puts severe, unequal stresses on the rotor blade, it is avoided and may lead to dangerous accidents if it occurs. It is one of the limiting factors to the size of rotors, and also to the forward speeds of helicopters (as this speed is added to the forward-sweeping (leading) side of the rotor, thus possibly causing localized transonics).

Key Points:

1. Compressible, Subsonic Flow

In fluid dynamics, a flow is considered to be a compressible flow if the density of the fluid changes with respect to pressure. A flow in which the Mach number is less than one everywhere is known as subsonic flow. Typically, if the freestream Mach number is less than about 0.8 then the flow is usually subsonic everywhere. However in some situations like flow over a highly cambered airfoil, the Mach number may locally exceed unity due to acceleration of the fluid in the curved regions. Subsonic flows are characterized by the absence of shocks which appear only in supersonic flows. In general, this is the case where the Mach number (defined as the ratio of the flow speed to the local speed of sound) of the flow exceeds 0.3. The Mach 0.3 value is rather arbitrary, but it is used because gas flows with a Mach number below that value introduce less than 5% change in density. Furthermore, the maximum density change occurs at the stagnation points and the density change in the rest of the flow field will be significantly lower. The factor that distinguishes a flow from being compressible or incompressible is the fact that in compressible flow the changes in the velocity of the flow can lead to changes in the temperature which are not negligible. On the other hand in case of incompressible flow, the changes in the internal energy (i.e. temperature) are negligible even if the entire kinetic energy of the flow is converted to internal energy (i.e. the flow is brought to rest). These definitions, though they seem to be inconsistent, are all saying one and the same thing: the Mach number of the flow is high enough so that the effects of compressibility can no longer be neglected. For subsonic compressible flows, it is sometimes possible to model the flow by applying a correction factor to the answers derived from incompressible calculations or modelling - for example, the Prandtl-Glauert rule:

(a_c is compressible lift curve slope, a_i is the incompressible lift curve slope, and M is the Mach number). Note that this correction only yields acceptable results over a range of approximately 0.3<M<0.7. For many other flows, their nature is qualitatively different to subsonic flows. A flow where the local Mach number reaches or exceeds 1 will usually contain shock waves. A shock is an abrupt change in the velocity, pressure and temperature in a flow; the thickness of a shock scales with the molecular mean free path in the fluid (typically a few micrometers). Shocks form because information about conditions downstream of a point of sonic or supersonic flow cannot propagate back upstream past the sonic point. The behaviour of a fluid changes radically as it starts to move above the speed of sound (in that fluid), ie. When the Mach number is greater than 1. For example, in subsonic flow, a stream tube in an accelerating flow contracts. But in a supersonic flow, a stream tube in an accelerating flow expands. To interpret this in another way, consider steady flow in a tube that has a sudden expansion: the tube's cross section suddenly widens, so the cross-sectional area increases. In subsonic flow, the fluid speed drops after the expansion (as expected). In supersonic flow, the fluid speed increases. This sounds like a contradiction, but it isn't: the mass flux is conserved but because supersonic flow allows the density to change, the volume flux is not constant. This effect is utilized in De Laval nozzles.

2. Transonic Flow past Unswept Airfoils

When an airplane is in motion at subsonic speeds, the air is treated as though it was incompressible. As airplane speed increases, however, the air loses its assumed incompressibility and the error in estimating, for example, drags, becomes greater and greater. The question arises as to how fast an airplane must be moving before one must take into account compressibility.

A disturbance in the air will send pressure pulses or waves out into the air at the speed of sound. Consider the instance of cannon fired at sea level. An observer situated some distance from the cannon will see the flash almost instantaneously, but the sound wave is heard (or the pressure wave is felt) some time later. The observer can easily compute the speed of sound by dividing the distance between him and the cannon by the time it takes the sound to reach him. The disturbance propagates out away from the cannon in an expanding hemispherical shell.

The speed of sound varies with altitude. More precisely, it depends upon the square root of the absolute temperature. At sea level under standard conditions (T_o = 288.15 K (degrees Kelvin)), the speed of sound is 340.3 meters per second (761.2 miles per hour), but at an altitude of 15 kilometers (9.3 miles or 49,212 feet) where the temperature is down to 216.7 K, the speed of sound is only 295.1 meters per second (660.2 miles per hour). This difference indicates that an airplane flying at this altitude encounters the speed of sound at a slower speed, and, therefore, comes up against compressibility effects sooner.

An airplane flying well below the speed of sound creates a disturbance in the air and sends out pressure pulses in all directions. Air ahead of the airplane receives these "messages" before the airplane arrives and the flow separates around the airplane. But as the plane approaches the speed of sound, the pressure pulses merge closer and closer together in front of the airplane and little time elapses between the time the air gets a warning of the plane's approach and the plane's actual arrival time at the speed of sound, the pressure pulses move at the same speed as the plane. They merge ahead of the airplane into a "shock wave" that is an almost instantaneous line of change in pressure, temperature, and density. The air has no warning of the approach of the airplane and abruptly passes through the shock system. There is a tendency for the air to break away from the airplane and not flow smoothly about it; as a result, there is a change in the aerodynamic forces from those experienced at low incompressible flow speeds.

The Mach number is a measure of the ratio of the airplane speed to the speed of sound. In other words, it is a number that may relate the degree of warning that air may have to an airplane's approach. The Mach number is named after Ernst Mach, an Austrian professor. For Mach numbers less than one, one has subsonic flow, for Mach numbers greater than one, supersonic flow, and for Mach numbers greater than 5, the name is hypersonic flow. Additionally, transonic flow pertains to the range of speeds in which flow patterns change from subsonic to supersonic or vice versa, about Mach 0.8 to 1.2. Transonic flow presents a special problem area as neither equations describing subsonic flow nor those describing supersonic flow may be accurately applied to the regime. At subsonic speeds, drag was composed of three main componentsskin-friction drag, pressure drag, and induced drag (or drag due to lift). At transonic and supersonic speeds, there is a substantial increase in the total drag of the airplane due to fundamental changes in the pressure distribution.

This drag increase encountered at these high speeds is called *wave drag*. The drag of the airplane wing, or for that matter, any part of the airplane rises sharply, and large increases in thrust are necessary to obtain further increases in speed. This wave drag is due to the unstable formation of shock waves that transforms a considerable part of the available propulsive energy into heat, and to the induced separation of the flow from the airplane surfaces. Throughout the transonic range, the drag coefficient of the airplane is greater than in the supersonic range because of the erratic shock formation and general flow instabilities. Once a supersonic flow has been established, however, the flow stabilizes and the drag coefficient is reduced.

The total drag at transonic and supersonic speeds can be divided into two categories: (1) zero-lift drag composed of skin-friction drag and wave (or pressure-related) drag of zero lift and (2) lift-dependent drag composed of induced drag (drag due to lift) and wave (or pressure-related) drag due to lift. In the early days of transonic flight, the sound barrier represented a real barrier to higher speeds. Once past the transonic regime, the drag coefficient and the drag decrease, and less thrust is required to fly supersonically. However, as it proceeds toward higher supersonic speeds, the drag increases (even though the drag coefficient may show a decrease).

It is a large loss in propulsive energy due to the formation of shocks that causes wave drag. Up to a free-stream Mach number of about 0.7 to 0.8, compressibility effects have only minor effects on the flow pattern and drag. The flow is subsonic everywhere. As the flow must speed up as it proceeds about the airfoil, the local Mach number at the airfoil surface will be higher than the free-stream Mach number. There eventually occurs a free-stream Mach number called the critical Mach number at which a supersonic point appears somewhere on the airfoil surface, usually near the point of maximum thickness, and indicates that the flow at that point has reached Mach 1. As the free-stream Mach number is increased beyond the critical Mach number and approaches Mach 1, larger and larger regions of supersonic flow appear on the airfoil surface. In order for this supersonic flow to return to subsonic flow, it must pass through a shock (pressure discontinuity). This loss of velocity is accompanied by an increase in temperature, that is, a production of heat. This heat represents an expenditure of propulsive energy that may be presented as wave drag. These shocks appear anywhere on the airplane (wing, fuselage, engine nacelles, etc.) where, due to curvature and thickness, the localized Mach number exceeds 1.0 and the airflow must decelerate below the speed of sound. For transonic flow, the wave drag increase is greater than would be estimated from a loss of energy through the shock. In fact, the shock wave interacts with the boundary layer so that a separation of the boundary layer occurs immediately behind the shock. This condition accounts for a large increase in drag that is known as shock-induced (boundary-layer) separation.

The free-stream Mach number at which the drag of the airplane increases markedly is called the *drag-divergence Mach number*. Large increases in thrust are required to produce any further increases in airplane speed. If an airplane has an engine of insufficient thrust, its speed will be limited by the drag-divergence Mach number. The prototype Convair F- 102A was originally designed as a supersonic interceptor but early flight tests indicated that because of high drag, it would never achieve this goal. It later achieved its goal through a redesign.

At a free-stream Mach number greater than 1, a bow shock appears around the airfoil nose. Most of the airfoil is in supersonic flow. The flow begins to realign itself parallel to the body surface and stabilize, and the shock-induced separation is reduced.
This condition results in lower drag coefficients. Supersonic flow is better behaved than transonic flow and there are adequate theories that can predict the aerodynamic forces and moments present. Often, in transonic flow, the flow is unsteady, and the shock waves on the body surface may jump back and forth along the surface, thus disrupting and separating the flow over the wing surface. This sends pulsing, unsteady flow back to the tail surfaces of the airplane. The result is that the pilot feels a buffeting and vibration of both wing and tail controls. This condition occurred especially in the first airplane types to probe the sound barrier. With proper design, however, airplane configurations gradually evolved to the point where flying through the transonic region posed little or no difficulty in terms of wing buffeting or loss of lift.

The question arises as to whether one may delay the drag-divergence Mach number to a value closer to 1 so as to impart the ability to fly at near-sonic velocities with the same available engine thrust before encountering large wave drag. There are a number of ways of delaying the transonic wave drag rise (or equivalently, increasing the drag-divergence Mach number closer to 1). These include:

- Use of thin airfoils;
- Use of a forward or backward swept wing;
- Low-aspect-ratio wing;
- Removal of boundary layer and vortex generators; and
- Supercritical and area-rule technology.

3. Swept Wings at Transonic Speeds

In the transonic the swept wing also sweeps the shock which is at the top rear of the wing. Only the velocity component perpendicular to the shock is affected and suffers an entropy increase. As an aircraft enters the transonic speeds just below the speed of sound, an effect known as wave drag starts to appear. Using conservation of momentum principles in the direction normal to surface curvature, airflow accelerates around curved surfaces, and near the speed of sound the acceleration can cause the airflow to reach supersonic speeds. When this occurs, an oblique shock wave is generated at the point where the flow goes supersonic. Since this occurs on curved areas, they are normally associated with the upper surfaces of the wing, the cockpit canopy, and the nose cone of the aircraft, areas with the highest local curvature.

Shock waves require energy to form. This energy is taken out of the aircraft, which has to supply extra thrust to make up for this energy loss. Thus the shocks are seen as a form of drag. Since the shocks form when the local air velocity reaches supersonic speeds over various features of the aircraft, there is a certain "critical mach" speed (or drag divergence Mach number) where this effect becomes noticeable. This is normally when the shocks start generating over the wing, which on most aircraft is the largest continually curved surface, and therefore the largest contributor to this effect.

Since these shock waves are generated at areas of curvature, the obvious way to reduce their effect is to reduce the curvature. In the case of the fuselage, this suggests long, thin designs that are pointed at the ends. Such designs are common on high speed aircraft, the Concorde being one example, and are referred to as having a high "fineness ratio".

This applies to the wing as well, which suggests that wings should have as little curvature as possible, be as thin as possible, and have a long chord. Examples of this sort of wing planform can be found on the F-104 Starfighter for instance, which is highly optimized for high-speed performance. However, these same characteristics make a wing have a very low lift coefficient, and poor performance at slow speeds. The Starfighter has had a large number of landing accidents caused by its very high landing speed that was needed to keep the wing generating enough lift to fly.

Swept wings essentially "fool" the airflow at high speeds into thinking the wing has a longer and flatter profile than it has as measured "head on" to the wing. At low angles of attack airflow over the wing travels almost directly front to back, so a wing swept at 45 degrees would see an effective chord 1.4 times the actual chord. This reduces the effects of wave drag, making transonic flight much more economical. Early experiments demonstrated that the peak drag was lowered by as much as four times compared to a straight wing.

4. Forward Swept Wing

The forward-swept wing is a high-performance aircraft configuration, first proposed in 1936 by German aircraft designers. Perceived benefits of a forward-swept wing design include:

- mounting the wings further back on the fuselage, allowing for an unobstructed cabin or bomb bay, and
- Increased maneuverability at transonic speeds, due to airflow from wing tip to wing root preventing a stall of the wing tips and ailerons at high angle of attack.

5. Transonic Aircraft

As an aircraft enters the transonic speeds just below the speed of sound, an effect known as wave drag starts to appear. Using conservation of momentum principles in the direction normal to surface curvature, airflow accelerates around curved surfaces, and near the speed of sound the acceleration can cause the airflow to reach supersonic speeds. When this occurs, an oblique shock wave is generated at the point where the flow goes supersonic. Since this occurs on curved areas, they are normally associated with the upper surfaces of the wing, the cockpit canopy, and the nose cone of the aircraft, areas with the highest local curvature.

For aircraft speeds which are very near the speed of sound, the aircraft is said to be transonic. Typical speeds for transonic aircraft are greater than 250 mph but less than 760 mph, and the Mach number M is nearly equal to one, M ~= 1 . While the aircraft itself may be traveling less than the speed of sound, the air going around the aircraft exceeds the speed of sound at some locations on the aircraft. In the regions where the local airspeed is near or greater than the speed of sound, we encounter compressibility effects and the air density may vary because of local shock waves, expansions, or flow choking.

The first powered aircraft to explore this regime were the high performance fighters of World War II. These aircraft seemed to encounter a sound barrier at which drag was increasing faster than thrust. There was speculation in the mid-1940's that manned flight was not possible at speeds faster than the speed of sound, even though the muzzle velocity of rifle bullets is supersonic. Of course, the flight of the X-1A in 1947 proved that people could fly faster than sound and, until the recent retirement of the Concorde, any person with enough money can fly supersonic. As mentioned above, even though modern airliners typically fly at about M = .85, the flow over the wings is transonic or supersonic. Drag increases dramatically as an aircraft approaches Mach 1, so airliners use high thrust gas turbine propulsion systems. On the slide we show a DC-8 airliner which is powered by four turbofan engines. The wings of airliners are typically swept in planform to reduce the transonic drag. For Mach numbers less than 2.0, the frictional heating of the airframe is low enough that light weight aluminum is used for the structure.

- In Section 4 of this course you will cover these topics:
- Two-Dimensional Supersonic Flows Around Thin Airfoil
- Supersonic Flows Over Wings And Airplane Configurations
- Hypersonic Flows

Topic Objective:

At the end of this topic student will be able to understand:

- Linear Theory
- Second-Order Theory (Busemann's Theory)
- Shock-Expansion Technique

Definition/Overview:

The term supersonic is used to define a speed that is over the speed of sound (Mach 1). At a typical temperature like 21 C (70 F), the threshold value required for an object to be traveling at a supersonic speed is approximately 344 m/s, (1,129 ft/s, 761 mph or 1,238 km/h). Speeds greater than 5 times the speed of sound are often referred to as hypersonic. Speeds where only some parts of the air around an object (such as the ends of rotor blades) reach supersonic speeds are labeled transonic (typically somewhere between Mach 0.8 and Mach 1.2).

Sounds are travelling vibrations (pressure waves) in an elastic medium. In gases sound travels longitudinally at different speeds, mostly depending on the molecular mass and temperature of the gas; (pressure has little effect). Since air temperature and composition varies significantly with altitude, Mach numbers for aircraft can change without airspeed varying. In water at room temperature supersonic can be considered as any speed greater than 1,440 m/s (4,724 ft/s). In solids, sound waves can be longitudinal or transverse and have even higher velocities. Supersonic fracture is crack motion faster than the speed of sound in a brittle material.

Key Points:

1. Linear Theory

Characterizing the complete input-output properties of a system by exhaustive measurement is usually impossible. When a system qualifies as a *linear system*, it is possible to use the responses to a small set of inputs to predict the response to any possible input. This can save the scientist enormous amounts of work, and makes it possible to characterize the system completely. These notes explain the following ideas related to linear systems theory:

- The challenge of characterizing a complex systems
- Simple linear systems
    1. Homogeneity
    2. Additivity
    3. Superposition
- Shift-invariance
    1. Decomposing a signal into a set of shifted and scaled impulses
    2. The impulse response function
    3. Use of sinusoids in analyzing shift-invariant linear systems
    4. Decomposing stimuli into sinusoids via Fourier Series
    5. Characterizing a shift-invariant system using sinusoids

- Examples
  1. Stereo
  2. Swinging pendulum

2. Second-Order Theory (Busemann's Theory)

Busemann's Biplane is a conceptual airframe design invented by Adolf Busemann which inherently prohibits the formation of N-type shock waves and thus does not create a sonic boom.

It consists of two triangular cross-section plates a certain distance apart, with flat sides top and bottom parallel to the fluid flow. The spacing between the plates is sufficiently large that the flow does not choke and supersonic flow is maintained between them.

Usually with supersonic flow a positive pressure shock wave is generated at the front and a negative pressure shock wave at the rear. With the biplane, the high pressure shock wave created is only on the inside between the two plates and reflects between the two plates so that it cancels/fills in the expansion fan forming at the rear, leaving no external shock waves to propagate to infinity. The flat sides at top and bottom generate no shock waves as the flow is parallel. The lack of external shock waves means that the Busemann Biplane does not suffer from any wave drag.

Although it has been shown to work in wind tunnel testing, and it has been successfully tested for ammunition, nobody has as yet been able to suggest a practical implementation of the concept for aircraft, as it generates no lift.

If we retain terms of the order of in our expression for , This gives better results for , , than the linearized theory, but worse results for .

3. Shock-Expansion Technique

The pressures on the airfoil surface can also be calculated using oblique shocks and P-M expansions.

Note: at the trailing edge, there are a few conditions to be met.

- same flow direction
- same static pressure on both sides of the slip line

This is the most exact way of calculating forces and moments. It is also the most tedious but it is necessary for angles > 5.

Topic Objective:

At the end of this topic student will be able to understand:

- General Remarks about Lift and Drag
- General Remarks about Supersonic Wings
- Governing Equation and Boundary Conditions
- Conical-Flow Method
- Design Considerations for Supersonic Aircraft. Some Comments
- About the Design of the SST and of the HSCT

Definition/Overview:

Airflow at supersonic speeds generates lift through the formation of shock waves, as opposed to the patterns of airflow over and under the wing. These shock waves, as in the transonic case, generate large amounts of drag. One of these shock waves is created by the leading edge of the wing, but contributes little to the lift. In order to minimize the strength of this shock it needs to remain "attached" to the front of the wing, which demands a very sharp leading edge. To better shape the shocks that will contribute to lift, the rest of an ideal supersonic airfoil is roughly diamond-shaped in cross-section. For low-speed lift these same airfoils are very inefficient, leading to poor handling and very high landing speeds.

One way to avoid the need for a dedicated supersonic wing is to use a highly swept subsonic design. Airflow behind the shock waves of a moving body are reduced to subsonic speeds. This effect is used within the intakes of engines meant to operate in the supersonic, as jet engines are generally incapable of ingesting supersonic air directly. This can also be used to reduce the speed of the air as seen by the wing, using the shocks generated by the nose of the aircraft. As long as the wing lies behind the cone-shaped shock wave, it will "see" subsonic airflow and work as normal. The angle needed to lie behind the cone increases with increasing speed, at Mach 1.3 the angle is about 45 degrees, at Mach 2.0 it is 60 degrees. For instance, at Mach 1.3 the angle of the Mach cone formed off the body of the aircraft will be at about $\sin\mu = 1/M$ (μ is the sweep angle of the Mach cone)

Generally it is not possible to arrange the wing so it will lie entirely outside the supersonic airflow and still have good subsonic performance. Some aircraft, like the English Electric Lightning or F-106 Delta Dart are tuned entirely for high-speed flight and feature highly-swept planforms without regard to the low-speed problems this creates. In other cases the use of variable geometry wings, as on the F-14 Tomcat, allows an aircraft to move the wing to keep it at the most efficient angle regardless of speed, although the cost in complexity and weight makes this a rare feature.

Most high-speed aircraft have a wing that spends at least some of its time in the supersonic airflow. But since the shock cone moves towards the fuselage with increased speed (that is, the cone becomes narrower), the portion of the wing in the supersonic flow also changes with speed. Since these wings are swept, as the shock cone moves inward, the lift vector moves forward as the outer, rearward, portions of the wing are generating less lift. This results in powerful pitching moments and their associated required trim changes.

Key Points:

1. General Remarks about Lift and Drag

In aerodynamics, lift-induced drag, induced drag, vortex drag, or sometimes drags due to lift, is a drag force that occurs whenever a moving object redirects the airflow coming at it. This drag force occurs in airplanes due to wings or a lifting body redirecting air to cause lift and also in cars with airfoil wings that redirect air to cause a down force. With other parameters remaining the same, as the angle of attack increases, induced drag increases.

2. General Remarks about Supersonic Wings

Sweep may be used to produce subsonic characteristics for a wing, even in supersonic flow. At some point, though, sweep is no longer very effective in delaying the effects of compressibility. That is, the difficulties associated with sweep outweigh the advantages as the required sweep angle gets very large. When the Mach number normal to the leading edge becomes greater than 1, the airfoil sections behave according to linear supersonic theory, with the associated wave drag.

For a double wedge: $C_d = C_l^2 (M^2-1)^{0.5}/4 + 4 (t/c)^2 / (M^2-1)^{0.5}$

For a parabolic section: $C_d = C_l^2 (M^2-1)^{0.5}/4 + 16/3 (t/c)^2 / (M^2-1)^{0.5}$

As in 2D, such supersonic wings are more easily analyzed than their subsonic counterparts, though. Consider the point (A) on the wing shown below. Its effect on the flow cannot propagate upstream because disturbances travel at the speed of sound and the freestream is traveling faster than this. This fact is called the law of forbidden signals and implies that disturbances originating at (A) can only affect the darker shaded area. Similarly, points outside the forward-going Mach cone (lightly shaded area) cannot affect the flow at point A.

This means that points on the tips of a supersonic wing can only affect a small part of the wing. The rest of the wing behaves as if it did not know about the wing tips and (except for the effects of sweep and taper) the rest of the wing may be treated as a set of 2-D sections. More detailed analysis shows that in the tip regions behave very much like 2-D sections with their lift curve slope reduced by 50%.

To avoid this loss of lift, the tip sections of supersonic wings are sometimes truncated so that no part of the wing is affected by the tips:

Sections with supersonic leading edges generally have more wave drag than sections with subsonic leading edges which can develop leading edge suction. For wings with sufficient sweep an important part of the design problem is to properly distribute the lift and volume over the length and span. The applet below shows some of the considerations involved in doing this.

3. Governing Equation and Boundary Conditions

The structure of Lecture 7 has as follows: In paragraph 3.0 we introduce the concept of inviscid fluid and formulate the governing equations and boundary conditions for an ideal fluid flow. In paragraph 3.1 we introduce the concept of circulation and state Kelvins theorem (a conservation law for angular momentum). In paragraph 3.2 we introduce the concept of vorticity.

4. Conical-Flow Method

In fluid dynamics, conical flow is an inviscid flow field in which properties depend solely on angle. Therefore no characteristic length is involved in conical flow. The most intuitive example is the inviscid flow over an infinite cone at zero incidences, where flow properties are function of semivertex angle only. Conical flow serves as a convenient approximation for supersonic flow analysis.

5. Design Considerations for Supersonic Aircraft. Some Comments

At supersonic speeds the shape and dimensions of the fuselage have a strong effect on the aircraft drag. Supersonic wave drag increases quickly as the fuselage volume increases and the fineness ratio is reduced. For this reason, the cabin diameter is kept as small as possible and the cabin length increased. The figure below shows a Aerospatiale design for the fuselage of a Mach 2.0 transport (Avion de Transport Supersonique Futur, ATSF).

Note that the diameter and seat layout is similar to the MD-80, but the fuselage is much longer. The Concorde diameter of 113 inches is very small because of the strong impact of fuselage diameter on wave drag. The requirement for a high overall fineness ratio is reflected in the fuselage geometries shown below.

Note that the Boeing design has a fuselage whose diameter varies over the cabin section. This is done to reduce the interference wave drag between wing and fuselage. This was not done on the Concorde as it was felt that the increase in production costs would be too high. Indeed the variable cross-section introduces many difficulties and affects the seating arrangement as shown below.

The supersonic business jet represents a somewhat less ambitious entry into commercial supersonic flight. Since supersonic wave drag depends on volume, the motivation for a smaller cabin cross-section is greater, and high fineness ratios are required. The drawings below illustrate the fuselage and cabin design for a supersonic business jet by Reno Aeronautical Corporation.

6. About the Design of the SST and of the HSCT

Jet engine design differs significantly between supersonic and subsonic aircraft. Jet engines, as a class, can supply *increased* fuel efficiency at supersonic speeds, even though their specific fuel consumption is greater at higher speeds. Because their speed over the ground is greater, this decrease in efficiency is less than proportional to speed until well above Mach 2, and the consumption per mile is lower.

When Concorde was being designed by Arospatiale-BAC, high bypass jet engines had not yet been deployed on subsonic aircraft, and Concorde would have been more competitive. When these high bypass jet engines reached commercial service in the 1960s, subsonic jet engines immediately became much more efficient, closer to the efficiency of turbojets at supersonic speeds. A bypass design is more fuel efficient at subsonic speeds, as they can reduce the jet exhaust speed to better match that of the aircraft. This capability would not improve efficiency, indeed would reduce it, during supersonic cruise, where the smaller size of turbojet engines gives low drag and better net efficiency.For example the early TU-144S was fitted with a low bypass jet engine which was much less efficient than Concorde's turbojets. The later TU-144D featured a turbojet engine and was comparable. Modern jet engines employ a high overall pressure ratio wherever possible, which, for fundamental reasons, yields better fuel efficiency and it may be that a more modern design would give even better fuel efficiency than Concorde's engines.

The High Speed Civil Transport (HSCT) was a NASA project to design a supersonic transport. It was to be a future Supersonic Passenger Aircraft, able to fly Mach 2, or twice the speed of sound. The project started in 1990 and ended during 1999. The goal was to employ up-to-date technologies. It was intended to cross the Atlantic or the Pacific Ocean in half the time of a non-supersonic aircraft. It was to be fuel efficient, carry 300 passengers, and it would have allowed customers to buy tickets at a much lower price than that of a ticket on a Concorde. The goal for its maiden flight was within 20 years.

Topic Objective:

At the end of this topic student will be able to understand:

- Newtonian Flow Model
- Stagnation Region Flow-Field Properties
- Modified Newtonian Flow
- Aerodynamic Heating
- Importance of Interrelating CFD, Ground-Test Data, and Flight-Test Data

Definition/Overview:

In aerodynamics, hypersonic speeds are speeds that are highly supersonic. Since the 1970s, the term has generally been assumed to refer to speeds of Mach 5 (5 times the speed of sound) and above. The hypersonic regime is a subset of the supersonic regime.

Supersonic airflow is very different from subsonic flow. Nearly everything about the way an aircraft flies changes dramatically as it accelerates to supersonic speeds. Even with this strong demarcation, there is still some debate as to the definition of "supersonic". One definition is that the aircraft, as a whole, is traveling at Mach 1 or greater. More technical definitions state that it is only supersonic if the airflow over the entire aircraft is supersonic, which occurs around Mach 1.2 on typical designs. The range Mach 0.75 to 1.2 is therefore considered transonic.

Considering the problems with this simple definition, the precise Mach number at which a craft can be said to be fully hypersonic is even more elusive, especially since physical changes in the airflow (molecular dissociation, ionization) occur at quite different speeds. Generally, a combination of effects become important "as a whole" around Mach 5. The hypersonic regime is often defined as speeds where ramjets do not produce net thrust. This is a nebulous definition in itself, as there exist a proposed change to allow them to operate in the hypersonic regime (the Scramjet).

Key Points:

1. Newtonian Flow Model

Under Newtonian flow the viscosity of a fluid is defined as the tangential force per unit area that will provide unit relative velocity between two parallel plates in the fluid unit distance apart. When the shear force exceeds a value characteristic of the material plastic flow can occur but this rarely happens in chromatography. However, if the flow rapidly changes direction then turbulent flow is introduced which is characterized by an increase in solute diffusiity and Newtonian flow no longer exists. The introduction of turbulent flow into connecting tubes (tubes between column and detector) reduces peak dispersion and, thus, maintains column resolution.

2. Stagnation Region Flow-Field Properties

Transient heat transfer problems involving melting or solidification are important in many engineering applications involving processes such as casting, welding and spray forming. The problem of solid-liquid phase change belongs to the class of moving boundary problems because of the existence of a moving interface. The rate of propagation of this boundary into the liquid region (solidification) or into the solid region (melting) depends on the thermal properties of the solid and liquid regions, and in addition, in the cases where there exists motion in the liquid phase, such as metal droplet solidification in spray processes, it also depends on the fluid and flow properties of the liquid region. In our study, the effect of the liquid motion on its solidification behavior has been investigated by considering the two-dimensional stagnation flow onto a cold substrate. By coupling liquid-phase momentum equation and the conductive-convective liquid energy equation with the heat conduction equation in the solid region as well as the energy balance equation at the interface, we set up the mathematical model of the half space convective Stefan solidification problem. An instantaneous-similarity method and a quasi-steady method, as well as the finite-difference method have been applied to solve the time-dependent system of equations. Parametric studies such as the effect of Prandtl number, Stefan number, the stagnation-flow flow strain rate, and the ratio of the liquid and solid phase thermal diffusivity, as well as the initial substrate and liquid phase temperatures on the solidification behavior have been conducted. The solution provides a more reasonable model for the solidification behavior of the liquid in motion, and provides better insight into situations such as those encountered during the deformation and solidification of a droplet impinging on a cold substrate.

3. Modified Newtonian Flow

In physics, Modified Newtonian dynamics (MOND) is a theory that proposes a modification of Newton's Second Law of Dynamics (F = ma) to explain the galaxy rotation problem. When the uniform velocity of rotation of galaxies was first observed, it was unexpected because Newtonian theory of gravity predicts that objects that are farther out will have lower velocities. For example, planets in the Solar System orbit with velocities that decrease as their distance from the Sun increases. MOND theory posits that acceleration is not linearly proportional to force at low values. The galaxy rotation problem may be understood without MOND if a halo of dark matter provides an overall mass distribution different from the observed distribution of normal matter. MOND was proposed by Mordehai Milgrom in 1981 to model the observed uniform velocity data without the dark matter assumption. He noted that Newton's Second Law for gravitational force has only been verified when gravitational acceleration is large.

4. Aerodynamic Heating

Aerodynamic heating is the heating of a solid body produced by the passage of fluid (such as air) over a body such as a meteor, missile, or airplane. It is a form of forced convection in that the flow field is created by forces beyond those associated with the thermal processes. The heat transfer essentially occurs at vehicle surface where aerodynamic friction ensures that the flow is at zero speed relative to the body for a very small layer of molecules. Because the flow has slowed to zero speed at this point a significant amount of its kinetic energy from the free-field is converted to heat. The total thermal energy at the surface is less than the stagnation temperature because the true process is not isentropic. The actual temperature is referred to as the recovery temperature. Heat then conducts into the surface material from the higher temperature air. The result is an increase in the temperature of the material and a loss of energy from the flow. The forced convection ensures that other material replenishes the gases that have cooled to continue the process. The stagnation and the recovery temperature of a flow increase with the speed of the flow and are greater at high speeds. The total thermal loading of the structure is a function of both the recovery temperature and the mass flow rate of the flow. Aerodynamic heating is greatest at high speed and in the lower atmosphere where the density is greater. Aerodynamic heating increases with the speed of the vehicle and is continuous from zero speed. It produces much less heating at subsonic speeds but becomes more important at supersonic speeds. At these speeds it can induce temperatures that begin to weaken the materials that compose the object. The heating effects are greatest at leading edges. Aerodynamic heating is dealt with by the use of high temperature alloys for metals, the addition of insulation of the exterior of the vehicle, or the use of ablative material.

5. Importance of Interrelating CFD, Ground-Test Data, and Flight-Test Data

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. Even with simplified equations and high-speed supercomputers, only approximate solutions can be achieved in many cases. Ongoing research, however, may yield software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is often performed using a wind tunnel with the final validation coming in flight test.

Flight test is a branch of aeronautical engineering that develops and gathers data during flight of an aircraft and then analyses the data to evaluate the flight characteristics of the aircraft and validate its design, including safety aspects. The flight test phase accomplishes two major tasks: 1) finding and fixing any aircraft design problems and then 2) verifying and documenting the aircraft capabilities for government certification or customer acceptance. The flight test phase can range from the test of a single new system for an existing aircraft to the complete development and certification of a new aircraft. Therefore the duration of a flight test program can vary from a few weeks to several years.

When the aircraft is completely assembled and instrumented, it typically conducts many hours of ground testing before its first/maiden flight. This ground testing will verify basic aircraft systems operations, measure engine performance, evaluate dynamic systems stability, and provide a first look at structural loads. Flight controls will also be checked out. Once all required ground tests are completed, the aircraft is ready for the first flight. First/maiden flight is a major milestone in any aircraft development program and is undertaken with the utmost caution.

There are several aspects to a flight test program: handling qualities, performance, aero-elastic/flutter stability, avionics/systems capabilities, weapons delivery, and structural loads. Handling qualities evaluates the aircraft's controllability and response to pilot inputs throughout the range of flight. Performance testing evaluates aircraft in relation to its projected abilities, such as speed, range, power available, drag, airflow characteristics, and so forth. Aero-elastic stability evaluates the dynamic response of the aircraft controls and structure to aerodynamic (i.e. air-induced) loads. Structural tests measure the stresses on the airframe, dynamic components, and controls to verify structural integrity in all flight regimes. Avionics/systems testing verifies all electronic systems (navigation, communications, radars, sensors, etc.) perform as designed. Weapons delivery looks at the pilots ability to acquire the target using on-board systems and accurately deliver the ordnance on target. Weapons delivery testing also evaluates the separation of the ordnance as it leaves the aircraft to ensure there are no safety issues. Other military unique tests are: air-to-air refueling, radar/infrared signature measurement, and aircraft carrier operations. Emergency situations are evaluated as a normal part of all flight test program. Examples are: engine failure during various phases of flight (takeoff, cruise, and landing), systems failures, and controls degradation. The overall operations envelope (allowable gross weights, centers-of-gravity, altitude, max/min airspeeds, maneuvers, etc.) is established and verified during flight testing. Aircraft are always demonstrated to be safe beyond the limits allowed for normal operations in the Flight Manual.

Because the primary goal of a flight test program is to gather accurate engineering data, often on a design that is not fully proven, piloting a flight test aircraft requires a high degree of training and skill, so such programs are typically flown by a specially trained test pilot, and the data is gathered by a flight test engineer, and often visually displayed to the test pilot and/or flight test engineer using flight test instrumentation.

- In Section 5 of this course you will cover these topics:

Aerodynamic Design Considerations

Tools For Defining The Aerodynamic Environment

Topic Objective:

At the end of this topic student will be able to understand:

- High-Lift Configurations
- Circulation Control Wing
- Design Considerations for Tactical Military Aircraft
- Drag Reduction
- Considerations for Wing/Canard and Tailless Configurations
- Comments on the F-15 Design
- The Design of the F-22

Definition/Overview:

Aerodynamics is important in a number of applications other than aerospace engineering. It is a significant factor in any type of vehicle design, including automobiles. It is important in the prediction of forces and moments in sailing. It is used in the design of large components such as hard drive heads. Structural engineers also use aerodynamics, and particularly aeroelasticity, to calculate wind loads in the design of large buildings and bridges. Urban aerodynamics seeks to help town planners and designers improve comfort in outdoor spaces, create urban microclimates and reduce the effects of urban pollution. The field of environmental aerodynamics studies the ways atmospheric circulation and flight mechanics affect ecosystems. The aerodynamics of internal passages is important in heating/ventilation, gas piping, and in automotive engines where detailed flow patterns strongly affect the performance of the engine.

Key Points:

1. High-Lift Configurations

Short take-off and landing lengths for high altitude air strips require meticulous design of high lift airfoil configurations. Precise relative positions and orientation of airfoil elements are essential to evolve landing configuration for high Clmax. The requirement gets compounded for finding take-off configuration which generates a low drag.
The current design process goes through numerical analysis of a large number of configurations, and their selection, before wind tunnel test is carried out. Though geometric parameterization is usually simple, the complexity lies in high angle of physics with accurate stall prediction. Our experience for a typical Reynolds number of 5 millions shows requirement of the near wall mesh to be orthogonal layer of 10 microns thickness. Though structured mutli-block mesh is best, and probably the only reliable option, time is wasted making it again-and-again for multiple configurations, combined by the need for multiple angle of attack runs per configuration for stall prediction.
Zeus Numerix delivered a customized package with a purpose to ensure rapid turn-around time with minimal user intervention. The following figure shows the flexibility that it offers in modifying geometric parameter.

The flexibility allowed for introduction of optimization module through which profile with required Clmax was obtained.

Apart from that in aircraft design, high-lift devices are a variety of mechanisms intended to add lift during certain portions of flight. They include common devices such as flaps and slats, as well as less common devices such as leading edge extensions and blown flaps. Generally they are divided into two classes by engineers, powered and unpowered.

All aircraft designs include a number of compromises intended to maximize performance for a particular role. One of the most fundamental of these is the size of the wing; a larger wing will provide more lift and make takeoffs and landings shorter and easier, but increase drag during cruising and thereby lead to lower fuel economy. High-lift devices are used to smooth out the differences between the two goals, allowing the use of an efficient cruising wing, and adding lift for takeoff and landing.

The most common high-lift device is the flap, a movable portion of the rear wing that can be bent down into the airflow to produce extra lift. The effect of the flap is not as obvious as it may seem; the real goal is to re-shape the wing as a whole into one that has more camber as well as being longer. In general wings with more camber and chord will produce more lift for any given amount of drag. It is the second goal, making the wing longer, that results in the complex flap arrangements found on many modern aircraft. The first "travelling flaps" that moved rearward were starting to be used just before World War II due to efforts at Arado, and have been followed by increasingly complex systems made up of several parts, known as slots. Large modern airliners make use of triple-slotted flaps to produce the massive lift required during takeoff.

Another common high-lift device is the slat, what appears to be a flap at the front of the wing. In fact the action of the slat is very different than the flap, as it does not directly produce extra lift. Instead the slat re-directs the airflow at the front of the wing, allowing it to flow more smoothly over the surface while at a high angle of attack. This allows the wing to be operated effectively at higher angles, which produce more lift. The original slats were patented by Handley-Page in 1919, and by the 1930s had developed into a system that operated automatically when the airflow over the wing reduced pressure on the leading edge, small springs would then push the slat out. More modern systems, like modern flaps, are more complex and typically operated hydraulically.

Although not as common, another high-lift device is the leading edge extension, or LEX. LEX systems typically consist of a thin delta wing mounted in front of the main wing, which normally generates little lift. At higher angles of attack, however, the LEX generates a vortex that is positioned to lie on the upper surface of the main wing. This reduces the pressure over the wing, leading to greater lift. LEX systems are notable for their potentially huge angles in which they operate, and are commonly found on modern fighter aircraft.

Powered high-lift systems generally use airflow from the engine to shape the flow of air over the wing, replacing or modifying the action of the flaps. Blown flaps use "bleed air" from the jet engine's compressor which is blown over the rear of the wing and flap, adding airflow and allowing the air to remain attached at higher angles of attack. In effect the airflow acts as a sort of slat for the flaps. A more advanced version of the Blown flap is the Circulation control wing; a mechanism that tangentially ejects air over a specially designed airfoil to create lift through the Coanda effect.

2. Circulation Control Wing

A more common system uses the airflow from the engines directly, by placing a flap directly in the path of the exhaust. This is not a trivial exercise due to the power of modern engines, and most aircraft deliberately "split" the flap so the portions behind the engines do not move into the airflow. However, if the flaps can be made strong enough, the effects can be enormous. Oddly the effect is most pronounced if the engines are mounted above the wing due to the Coandă effect, which led to a number of aircraft such as the Boeing YC-14 and Antonov An-72 with high-mounted engines. The C-17 Globemaster III uses a more conventional low-mounted engine design based on the same general idea.

3. Design Considerations for Tactical Military Aircraft

Several factors come into play during landing and takeoff that place requirements on the condition and size of airfield surfaces. These include an aircraft's landing gear and structural integrity, as well as its controllability and engines; these factors must receive consideration during aircraft design.

The capability of landing gear is of critical importance because it determines the aircraft's allowable sink rate (vertical descent) and the types of surfaces (roughness and strength) the aircraft can safely operate on. Structural integrity is also important. Theoretically, an aircraft moving over the irregular surface of a repaired runway could reach its natural resonance frequency and suffer serious structural damage.

Controlling the aircraft can be a problem when a pilot tries to land on a short, narrow MOS. The speed of the aircraft during final approach (generally in excess of 140 knots) and crosswinds are critical factors affecting aircraft control, especially if the pilot is trying to land on a MOS that is not parallel with the existing runway and markings (as may be the case during a contingency situation). Speed and crosswinds also become critical if the MOS's threshold is displaced significantly (several thousand feet) from the runway's threshold. Even for "normal" landings, the difficulty of maintaining control is evident in the fact that the number of accidents involving land-based aircraft increases in proportion to the square of the approach speed.

A related controllability factor is the effect of flaring. Because of the low sink rates that current Air Force tactical aircraft are capable of, the pilot must flare the aircraft before touchdown.

Flaring involves decreasing the aircraft's rate of descent just before it touches down. Flaring, as well as the previously mentioned landing-related factors, causes both lateral and longitudinal dispersion of the touchdown point, both of which can cause significant problems-especially when aircraft land on a MOS. Longitudinal dispersion reduces the usable length of a MOS (already short), just as lateral dispersion reduces the usable width of the 50-foot-wide operating surface.

Finally, critical systems such as engines affect the operating-surface requirements for takeoff and landing. On the one hand, aircraft engines providing short takeoff and vertical landing (STOVL) capabilities or engines equipped with thrust reversers can shorten the length of airfield surface that is required to take off or land safely. On the other hand, engines may be damaged by the ingestion of debris or other loose matter. This leads to foreign object damage (FOD)-a likely occurrence in the extremely dirty environment of a damaged airfield.

Thus, aircraft requirements directly affect the time and effort CE must expend to provide an adequate MOS for aircraft to operate from:

[The] extreme sensitivities to runway surface roughness of virtually all current tactical aircraft implicitly demand very high-quality, smooth, level repair of runway damage, which greatly extends the time required to restore runway operations after attack.[30]

That is, the higher the quality of surface repair required by aircraft, the longer the time needed to make the repair. However, by decreasing aircraft requirements (e.g., by using a more durable landing gear or by shortening the required MOS length), CE could reduce its repair time and aircraft could launch sooner.

Aircraft requirements can also affect enemy operations. For example, "the number of [enemy] sorties required to close a runway increases exponentially with a linear decrease in required runway length." An enemy has to cut an 8,000-to 10,000-foot runway only once or twice to deny an undamaged MOS (assuming MOS criteria dictated by current aircraft) but would need four to six successful sorties if our aircraft required only a 1,500-to 2,000-foot MOS. Finally, as the runway-length requirements of tactical aircraft decrease, the number of available runways within a given theater that are usable by those aircraft is sure to increase.

One must note that some people have the *perception* that improving aircraft systems to facilitate operations from repaired airfields would diminish an aircraft's performance. For example, thrust reversers, STOVL capability, better landing gear, and improvements to structural integrity can increase the aircraft's weight-the dread of tactical-aircraft designers. (The increase in weight due to improved landing, gear may not be as great as previously thought.) Extra weight can decrease an aircraft's payload, range, and maneuverability, thereby diminishing, its performance. Such improvements also increase manufacturing costs. Perhaps for these reasons, demands for such changes have not been forthcoming, despite the fact that much of the technology exists to implement them.

4. Drag Reduction

Drag reduction polymers function by delaying the onset of energy inefficient turbulent flow by maintaining energy efficient laminar flow:

- At the same flow rates pumping pressure can be reduced by 80%.
- At the same pumping pressure flow rate can be increased by 30 to 40%.

Drag reduction polymers are utilised where hardware or pumping constraints do not allow an alternative mechanical solution. The drag reduction effect is achieved using just a few parts per million (ppm) polymers in small pipe applications and around 50ppm in large pipe applications. Aqueous polymeric Drag Reduction was discovered using Ciba products. Polymers properties have been improved to give fast dissolution and ease of application.

5. Considerations for Wing/Canard and Tailless Configurations

In aeronautics, canard (French for duck) is an airframe configuration of fixed-wing aircraft in which the tailplane is ahead of the main wing, rather than behind them as in conventional aircraft empennage. The earliest airplanes, such as the Wright Flyer and the Santos-Dumont 14-bis, due to their tail-first configuration were seen by observers to resemble a flying duck hence the name.

5.1. Classes of canards

The canard wing exists in two classes: the control-canard and the lifting-canard.

5.1.1. Control-canard

In the control-canard, most of the weight of the aircraft is carried by the main wing and the canard wing is used primarily for longitudinal control during maneuvering. A control-canard mostly operates at zero angle of attack. Combat aircraft that have the canard configuration typically have a control-canard. In combat aircraft, the canard is usually driven by a computerized flight control system.

One benefit obtainable from a control-canard is avoidance of pitch-up. An all-moving canard capable of a significant nose-down deflection will protect against pitch-up. As a result, the aspect ratio and wing-sweep of the main wing can be optimized without having to guard against pitchup.

5.1.2. Lifting-canard

In the lifting-canard, the weight of the aircraft is always shared between the main wing and the canard wing. A lifting-canard generates an upload, in contrast to a conventional aft-tail which mostly generates a download that must be counteracted by extra lift on the main wing. The lifting-canard configuration is therefore more efficient than a conventional aft-tail from the perspective of induced drag. The lift generated by the canard wing is significant, so in order to minimise induced drag on the canard, it is usually of higher aspect ratio and greater airfoil camber than a control-canard.

With a lifting-canard, the main wing must be located further aft of the center of gravity range than with a conventional aft tail, and this increases the pitching moment caused by trailing-edge flaps. Aircraft with lifting canards cannot readily be designed with sophisticated trailing-edge flaps.

Tailless aircraft and Wing/Tail: tailless aircraft (often tail-less) traditionally has all its horizontal control surfaces on its main wing surface. It has no horizontal stabilizer - either tailplane or canard foreplane (nor does it have a second wing in tandem arrangement). A 'tailless' type usually still has a vertical stabilising fin (vertical stabilizer) and control surface (rudder). However, NASA has recently adopted the 'tailless' description for the novel X-36 research aircraft which has a canard foreplane but no vertical fin.

The most successful tailless configuration has been the tailless delta, especially for combat aircraft.

6. Comments on the F-15 Design

The F-15's maneuverability is derived from low wing loading (weight to wing area ratio) with a high thrust-to-weight ratio enabling the aircraft to turn tightly without losing airspeed. The weapons and flight control systems are designed so one person can safely and effectively perform air-to-air combat.

A multi-mission avionics system includes a Head-Up Display, advanced radar, inertial navigation system, flight instruments, ultra high frequency communications, tactical navigation system and Instrument Landing System. It also has an internally mounted, tactical electronic-warfare system, "identification friend or foe" system, electronic countermeasures set and a central digital computer.

The head-up display projects through a combiner, all essential flight information gathered by the integrated avionics system. This display, visible in any light condition, provides the pilot information necessary to track and destroy an enemy aircraft without having to look down at cockpit instruments.

The F-15's versatile APG-63/70 pulse-Doppler radar system can look up at high-flying targets and down at low-flying targets without being confused by ground clutter. It can detect and track aircraft and small high-speed targets at distances beyond visual range down to close range, and at altitudes down to treetop level. The radar feeds target information into the central computer for effective weapons delivery. For close-in dogfights, the radar automatically acquires enemy aircraft, and this information is projected on the head-up display. The F-15's electronic warfare system provides both threat warning and automatic countermeasures against selected threats. Because of the advanced electronics deployed on the F-15, the aircraft was given the nickname "Starship" by users.

A variety of air-to-air weaponry can be carried by the F-15. An automated weapon system enables the pilot to perform aerial combat safely and effectively, using the head-up display and the avionics and weapons controls located on the engine throttles or control stick. When the pilot changes from one weapon system to another, visual guidance for the required weapon automatically appears on the head-up display.

The Eagle can be armed with combinations of four different air-to-air weapons: AIM-7F/M Sparrow missiles or AIM-120 AMRAAM advanced medium range air-to-air missiles on its lower fuselage corners, AIM-9L/M Sidewinder or AIM-120 missiles on two pylons under the wings, and an internal 20 mm Gatling gun in the right wing root.

Low-drag, Conformal Fuel Tanks were especially developed for the F-15C and D models. Conformal fuel tanks can be attached to the sides of the engine air intake trunks under each wing and are designed to the same load factors and airspeed limits as the basic aircraft. Each conformal fuel tank contains about 114 cu. ft. (3,200 L) of usable space. These tanks reduce the need for in-flight refueling on global missions and increase time in the combat area. All external stations for munitions remain available with the tanks in use. Sparrow or AMRAAM missiles, moreover, can be attached to the corners of the conformal fuel tanks. Because the CFTs degrade performance (although not as much as normal external tanks), and cannot be jettisoned in-flight (unlike normal external tanks) air combat F-15s (A/B/C/D) typically fly without them, while the F-15E typically flies with them.

The F-15E Strike Eagle is a two-seat, dual-role, totally integrated fighter for all-weather, air-to-air and deep interdiction missions. The rear cockpit is upgraded to include four multi-purpose CRT displays for aircraft systems and weapons management. The digital, triple-redundant Lear Siegler flight control system permits coupled automatic terrain following, enhanced by a ring-laser gyro inertial navigation system.

For low-altitude, high-speed penetration and precision attack on tactical targets at night or in adverse weather, the F-15E carries a high-resolution APG-70 radar and low-altitude navigation and targeting infrared for night pods.

7. The Design of the F-22

7.1. Characteristics

The F-22 is a fifth generation fighter that is considered a fourth-generation stealth aircraft by the USAF. Its dual afterburning Pratt & Whitney F119-PW-100 turbofans incorporate pitch axis thrust vectoring, with a range of 20 degrees. The maximum thrust is classified, though most sources place it at about 35,000 lbf (156 kN) per engine. Maximum speed, without external weapons, is estimated to be Mach 1.82 in super-cruise mode; as demonstrated by General John P. Jumper, former US Air Force Chief of Staff, when his Raptor exceeded Mach 1.7 without afterburners on 13 January 2005. With afterburners, it is "greater than Mach 2.0" (1,317 mph, 2,120 km/h), according to Lockheed Martin; however, the Raptor can easily exceed its design speed limits, particularly at low altitudes, with max-speed alerts to help prevent the pilot from exceeding them.

Former Lockheed Raptor chief test pilot Paul Metz stated that the Raptor has a fixed inlet; but while the absence of variable intake ramps may theoretically make speeds greater than Mach 2.0 unreachable; there is no evidence to prove this. Such ramps would be used to prevent engine surge resulting in a compressor stall, but the intake itself may be designed to prevent this. Metz has also stated that the F-22 has a top speed greater than 1,600 mph (Mach 2.42) and its climb rate is faster than the F-15 Eagle due to advances in engine technology, despite the F-15's thrust-to-weight ratio of about 1.2:1, with the F-22 having a ratio closer to 1:1. The US Air Force claims that the F-22A cannot be matched by any known or projected fighter.

The true top-speed of the F-22 is unknown to the general public. The ability of the airframe to withstand the stress and heat from friction is a further, key factor, especially in an aircraft using as many polymers as the F-22. However, while some aircraft are faster on paper, the internal carriage of its standard combat load allows the aircraft to reach comparatively higher performance with a heavy load over other modern aircraft due to its lack of drag from external stores. It is one of only a handful of aircraft that can sustain supersonic flight without the use of afterburner augmented thrust (and its associated high fuel usage). This ability is called super-cruise.

The F-22 is highly maneuverable, at both supersonic and subsonic speeds. It is extremely departure-resistant, enabling it to remain controllable at extreme pilot inputs. The F-22's thrust vectoring nozzles allow the aircraft to turn tightly, and perform extremely high alpha (angle of attack) maneuvers such as the Herbst maneuver (or J-turn), Pugachev's Cobra, and the Kulbit, though the J-Turn is more useful in combat. The F-22 is also capable of maintaining a constant angle of attack of over 60, yet still having some control of roll. During June 2006 exercises in Alaska, F-22 pilots demonstrated that cruise altitude has a significant effect on combat performance, and routinely attributed their altitude advantage as a major factor in achieving an unblemished kill ratio against other US fighters and 4th/4.5th generation fighters.

7.2. Avionics

The F-22's avionics include BAE Systems E&IS radar warning receiver (RWR) AN/ALR-94, AN/AAR 56 Infra-Red and Ultra-Violet MAWS (Missile Approach Warning System) and the Northrop Grumman AN/APG-77 Active Electronically Scanned Array (AESA) radar. The AN/APG-77 has both long-range target acquisition and low probability of interception of its own signals by enemy aircraft.

The AN/ALR-94 is a passive receiver system capable of detecting the radar signals in the environment. Composed of more than 30 antennas smoothly blended into the wings and fuselage, it is described by the former head of the F-22 program at Lockheed Martin Tom Burbage as "the most technically complex piece of equipment on the aircraft." With greater range (250+ nmi) than the radar, it enables the F-22 to limit its own radar emission which might otherwise compromise its stealth. As the target approaches, AN/ALR-94 can cue the AN/APG-77 radar to keep track of its motion with a narrow beam, which can be as focused as 2 by 2 in azimuth and elevation.

The AN/APG-77 AESA radar, designed for air-superiority and strike operations, features a low-observable, active-aperture, electronically-scanned array that can track multiple targets in all kinds of weather. The AN/APG-77 changes frequencies more than 1,000 times per second to reduce the chance of being intercepted. The radar can also focus its emissions to overload enemy sensors, giving the aircraft an electronic-attack capability.

The radars information is processed by two Raytheon Common Integrated Processor (CIP)s. Each CIP operates at 10.5 billion instructions per second and has 300 megabytes of memory. Information can be gathered from the radar and other onboard and offboard systems, filtered by the CIP, and offered in easy-to-digest ways on several cockpit displays, enabling the pilot to remain on top of complicated situations. The Raptors software is composed of over 1.7 million lines of code, most of which concerns processing data from the radar. The radar has an estimated range of 125-150 miles, though planned upgrades will allow a range of 250 miles (400 km) or more in narrow beams.

The F-22 has several unique functions for an aircraft of its size and role. For instance, it has threat detection and identification capability along the lines of that available on the RC-135 Rivet Joint. While the F-22's equipment isn't as powerful or sophisticated, because of its stealth, it can be typically hundreds of miles closer to the battlefield, which often compensates for the reduced capability.

The F-22 is capable of functioning as a "mini-AWACS." Though reduced in capability compared to dedicated airframes such as the E-3 Sentry, as with its threat identification capability, the F-22's forward presence is often of benefit. The system allows the F-22 to designate targets for cooperating F-15s and F-16s, and even determine if two friendly aircraft are targeting the same enemy aircraft, thus enabling one of them to choose a different target. It is often able to identify targets "sometimes many times quicker than the AWACS."

The F-22's low probability of intercept radar is being given a high-bandwidth data transmission capability, to allow it to be used in a "broadband" role to permit high-speed relaying of data between friendly transmitters and receivers in the area. The F-22 can already pass data to other F-22s, resulting in considerably reduced radio "chatter".

The IEEE-1394B data bus, developed for the F-22, was derived from the commercial IEEE-1394 "FireWire" bus system, often used on personal computers. The same data bus is employed by the subsequent F-35 Lightning II fighter.

7.3. Cockpit

The F-22 cockpit is a glass cockpit design without any traditional analog flight instruments and represents a marked improvement on the cockpit design of previous advanced aircraft. The leading features of the F-22 cockpit include simple and rapid start-up, highly developed HMI, light helmet, large anthropometric accommodation and highly integrated warning system. Other main features include the large single-piece canopy and improved life support systems.

7.4. Airframe

Several small design changes were made from the YF-22A prototype to the production F-22A. The swept-back angle on the wing's leading edge was decreased from 48 degrees to 42 degrees, while the vertical stabilizer area was decreased 20%. To improve pilot visibility, the canopy was moved forward 7 inches (178 mm) and the engine intakes were moved rearward 14 inches (356 mm). The shape of the wing and stabilator trailing edges was refined to improve aerodynamics, strength, and stealth characteristics. The Airframe also features three internal weapons bays on the bottom and sides of the fuselage.

7.5. Armament

The Raptor is designed to carry air-to-air missiles in internal weapons bays, both to avoid disrupting its stealth capability and to reduce drag resulting in higher top speeds and longer combat ranges. Launching missiles requires opening the weapons bay doors for less than a second, while the missiles are pushed clear of the airframe by hydraulic arms. The aircraft can also carry bombs compatible with the Joint Direct Attack Munition (JDAM) guidance system, and the new Small-Diameter Bomb (SDB). The Raptor carries an M61A2 Vulcan 20 mm rotary cannon, also with a trap door, in the right wing root. The M61A2 is a last ditch weapon, and carries only 480 rounds; enough ammunition for approximately five seconds of sustained fire. Despite this, the F-22 has been able to use its gun in dogfighting without being detected, which can be necessary when missiles are depleted.

The Raptor's very high sustained cruise speed and operational altitude add significantly to the effective range of both air-to-air and air-to-ground munitions. These factors may be the rationale behind the USAF's decision not to pursue long-range, high-energy air-to-air missiles such as the MBDA Meteor. However, the USAF plans to procure the AIM-120D AMRAAM, which will have a significant increase in range compared to the AIM-120C. The Raptor launch platform provides additional energy to the missile which helps improve the range of air-to-ground ordnance. While specific figures remain classified, it is expected that JDAMs employed by F-22s will have twice or more the effective range of munitions dropped by legacy platforms. In testing, a Raptor dropped a 1,000 lb (450 kg) JDAM from 50,000 feet (15,000 m), while cruising at Mach 1.5, striking a moving target 24 miles (39 km) away. The SDB, as employed from the F-22, should see even greater increases in effective range, due to the improved lift to drag ratio of these weapons.

While in its air-superiority configuration, the F-22 carries its weapons internally, though it is not limited to this option. The wings include four hardpoints, each rated to handle 5,000 lb (2,300 kg). Each hardpoint has a pylon that can carry a detachable 600 gallon fuel tank or a rail launcher that holds two air-air missiles. However, use of external stores compromises the F-22's stealth, and has a detrimental effect on maneuverability, speed, and range. The two inner hardpoints are "plumbed" for external fuel tanks. The hardpoints allow detaching of the mounting pylons in flight so the fighter can regain its stealth after exhausting external stores. Research is currently being conducted to develop a stealth ordnance pod and hardpoints for it. Such a pod would comprise a stealth shape and carry its weapons internally, then would split open when launching a missile or dropping a bomb. Both the pod and hardpoints could be detached when no longer needed. This system would allow the F-22 to carry its maximum ordnance load while remaining stealthy, albeit at a loss of maneuverability.

7.6. Stealth

Although several recent Western fighter aircraft are less detectable on radar than previous designs using techniques such as radar absorbent material-coated S-shaped intake ducts that shield the compressor fan from reflecting radar waves, the F-22 design placed a much higher degree of importance on low observance throughout the entire spectrum of sensors including radar signature, visual, infrared, acoustic, and radio frequency.

The stealth of the F-22 is due to a combination of factors, including the overall shape of the aircraft, the use of radar absorbent material (RAM), and attention to detail such as hinges and pilot helmets that could provide radar return. However, reduced radar cross section is only one of five facets that designers addressed to create a stealth design in the F-22. The F-22 has also been designed to disguise its infrared emissions to make it harder to detect by infrared homing ("heat seeking") surface-to-air or air-to-air missiles. Designers also made the aircraft less visible to the naked eye, and controlled radio and noise emissions. The Raptor has an under bay carrier made for hiding heat from missile threats, like surface-to-air missiles.

The F-22 apparently relies less on maintenance-intensive radar absorbent material and coatings than previous stealth designs like the F-117. These materials caused deployment problems due to their susceptibility to adverse weather conditions. Unlike the B-2, which requires climate-controlled hangars, the F-22 can undergo repairs on the flight line or in a normal hangar. Furthermore, the F-22 has a warning system (called "Signature Assessment System" or "SAS") which presents warning indicators when routine wearand-tear have degraded the aircraft's radar signature to the point of requiring more substantial repairs. The exact radar cross section of the F-22 remains classified.

7.7. External lighting

The aircraft has integral position and anti-collision lighting (including strobes) on the wings, compatible with stealth requirements, supplied by Goodrich Corporation. The low voltage electroluminescent formation lights are located on the aircraft at critical positions for night flight operations (on both sides of the forward fuselage under the chin, on the tip of the upper left and right wings, and on the outside of both vertical stabilizers). There are similar airs refueling lights on the butterfly doors that cover the air refueling receptacle.

7.8. Operational history

Intended to be the leading American advanced tactical fighter in the early part of the 21st century, the Raptor is an expensive fighter with an incremental cost of about US$138 million per unit. The number of aircraft to be built has dropped to 183, down from the initial requirement of 750. Part of the reason for the decrease in the requirement is that the F-35 Lightning II uses technology from the F-22, but at a much more affordable price. To a large extent the cost of these technologies is lower for the F-35 only because they have already been developed for the F-22.

7.9. YF-22 "Lightning II"

The prototype YF-22 won a fly-off competition against the Northrop/McDonnell-Douglas YF-23 for the Advanced Tactical Fighter contract. In April 1992 during flight testing after contract award, test pilot Tom Morgenfeld escaped without injury when the first YF-22 prototype that he was flying crashed while landing at Edwards Air Force Base in California. The cause of the crash was found to be a flight control software error that failed to prevent a pilot-induced oscillation.

The YF-22 was a developmental aircraft that led to the F-22; however, there are significant differences between the YF-22 and the F-22. Relocation of cockpit, structural changes, and many other smaller changes exist between the two types. The two are sometimes confused in pictures, often at angles where it is difficult to see certain features. For example, there are some F-22 with pilot booms which some think are only found on the YF-22.

The YF-22 was originally given the unofficial name *"Lightning II"*, after the World War II fighter P-38, by Lockheed, which persisted until the mid-1990s when the USAF officially named the aircraft "Raptor". For a short while, the aircraft was also dubbed "SuperStar" and "Rapier". The F-35 later received the *Lightning II* name on 7 July 2006.

7.10. F-22 Raptor to F/A-22 and back again

The production model was formally named F-22 "Raptor" when the first production-representative aircraft was unveiled on 9 April 1997 at Lockheed-Georgia Co., Marietta, Georgia. First flight occurred on 7 September 1997.

In September 2002, Air Force leaders changed the Raptors designation to F/A-22. The new designation, which mimicked that of the Navys F/A-18 Hornet, was intended to highlight plans to give the Raptor a ground-attack capability amid intense debate over the relevance of the expensive air-superiority jet. This was later changed back to simply F-22 on 12 December 2005. On 15 December 2005, the F-22A entered service.

7.11. Testing

Testing of the F-22 began in 1997 and has been curtailed to save program costs, but risks hiding flaws until a point at which fixing flaws become unaffordable. The U.S. General Accounting Office cautioned, "Moreover, engine and stealthiness problems already disclosed by the DoD, and the potential for avionics and software problems, underscore the need to demonstrate the weapon systems performance through flight testing before significant commitments are made to production."

Raptor 4001 was retired and sent to Wright-Patterson AFB to be fired at for testing the fighter's survivability. Usable parts of 4001 would be used to make a new F-22. Another engineering and manufacturing development (EMD) F-22 was also retired and likely to be sent to be rebuilt. A testing aircraft was converted to a maintenance trainer at Tyndall AFB.

On 3 May 2006, a report was released detailing a problem with a forward titanium boom on the aircraft that was not properly heat treated. Officials are still investigating the problem which was caused by the boom portion not being subjected to high temperatures in the factory for long enough, causing the boom to be less ductile than specified and potentially shortening the lives of the first 80 or so F-22s. Work is underway to restore them to full life expectancy.
The F-22 fleet underwent modifications at Hill AFB, and at Edwards AFB near Palmdale, California.

7.12. Recent developments

In 2006, the Raptor's development team, composed of Lockheed Martin and over 1,000 other companies, plus the United States Air Force, won the Collier Trophy, American aviation's most prestigious award. The U.S. Air Force will acquire F-22s that are to be divided among seven active duty combat squadrons, and jointly flown and maintained by three integrated Air Force Reserve Command and Air National Guard fighter squadrons.

During Exercise Northern Edge in Alaska in June 2006, 12 F-22s of the 94th FS downed 108 adversaries with no losses in simulated combat exercises. In two weeks of exercises, the Raptor-led Blue Force amassed 241 kills against two losses in air-to-air combat, and neither Blue Force loss was an F-22.

This was followed with the Raptor's first participation in a Red Flag exercise. 14 F-22s of the 94th FS supported attacking Blue Force strike packages as well as engaging in close air support sorties themselves in Red Flag 07-1 between 3 February and 16 February 2007. Against designed superior numbers of Red Force Aggressor F-15s and F-16s, it established air dominance using eight aircraft during day missions and six at night, reportedly defeating the Aggressors quickly and efficiently, even though the exercise rules of engagement allowed for four to five Red Force regenerations of losses but none to Blue Force. Further, no sorties were missed because of maintenance or other failures, and only one Raptor was adjudged lost against the virtual annihilation of the defending force. When their ordnance was expended, the F-22s remained in the exercise area providing electronic surveillance to the Blue Forces.

While attempting its first overseas deployment to the Kadena Air Base in Okinawa, Japan, on 11 February 2007, a group of six Raptors flying from Hickam AFB, Hawaii experienced multiple computer crashes coincident with their crossing of the 180th meridian of longitude (the International Date Line). The computer failures included at least navigation (completely lost) and communication. The fighters were able to return to Hawaii by following their tankers in good weather. The error was fixed within 48 hours and the F-22s continued their journey to Kadena.

In 2007, tests carried out by Northrop Grumman, Lockheed Martin, and L-3 Communications enabled the AESA system of a Raptor to act like a WiFi access point, able to transmit data at 548 Megabit/sec and receive at Gigabit speed; far faster than the current Link 16 system used by US and allied aircraft, which transfers data at just over 1 Megabit/sec.

F-22A Raptors of the 90th Fighter Squadron performed their first intercept of two Russian Tu-95MS 'Bear-H' bombers in Alaska, on 22 November 2007. This was the first time that F-22s had been called to support a NORAD mission.
On 12 December 2007, General John D.W. Corley, USAF, Commander of Air Combat Command, officially declared the F-22s of the integrated active duty 1st Fighter Wing and Virginia Air National Guard 192d Fighter Wing fully operational; three years after the first Raptor arrived at Langley Air Force Base, Virginia. This was followed from 13 April to 19 April 2008 by an Operational Readiness Inspection (ORI) of the integrated wing in which it received an "excellent" rating in all categories while scoring a simulated kill-ratio of 221-0. The first pair of Raptors assigned to the 49th Fighter Wing became operational at Holloman Air Force Base, New Mexico, on 2 June.

In July 2008, F-22s were to be showcased in the 2008 Royal International Air Tattoo air show at RAF Fairford, but did not perform after the show was canceled due to bad weather. An F-22, however, performed on the first day of the Farnborough Airshow on 14 July 2008.

On 28 August, 2008, an F-22 from the 411th Flight Test Squadron performed in the first ever air-to-air refueling of an aircraft using synthetic jet fuel. The test was a part of the wider USAF effort to qualify all of its aircraft to use the fuel, a 50/50 mix of JP-8 and a Fischer-Tropsch process-produced, and natural gas-based fuel. For the tests, no modifications were made to the F-22 or the KC-135 Stratotanker which performed the refueling.

Topic Objective:

At the end of this topic student will be able to understand:

- CFD Tools
- Establishing the Credibility of CFD Simulations
- Ground-Based Test Programs
- Flight-Test Programs

Definition/Overview:

A simple and rapid tool for calculating the aerodynamic characteristics of almost arbitrary wing configurations with sufficient accuracy for preliminary design is presented. The main emphasis is put on simple validation examples for subsonic and transonic cruise flight cases. An approach for polar transformation to consider transonic effects of swept wings is investigated in detail.

Key Points:

1. CFD Tools

"CADD" and "CAD" redirect here. For other uses, see CADD (disambiguation) and CAD (disambiguation). "ECAD" redirects here. For other uses, see ECAD (disambiguation).

Computer-aided design (CAD) is the use of computer technology to aid in the design and particularly the drafting (technical drawing and engineering drawing) of a part or product, including entire buildings. It is both a visual (or drawing) and symbol-based method of communication whose conventions are particular to a specific technical field.

Drafting can be done in two dimensions ("2D") and three dimensions ("3D"). Drafting is the integral communication of technical or engineering drawings and is the industrial arts sub-discipline that underlies all involved technical endeavors. In representing complex, three-dimensional objects in two-dimensional drawings, these objects have traditionally been represented by three projected views at right angles.

Current CAD software packages range from 2D vector-based drafting systems to 3D solid and surface modelers. Modern CAD packages can also frequently allow rotations in three dimensions, allowing viewing of a designed object from any desired angle, even from the inside looking out. Some CAD software is capable of dynamic mathematic modeling, in which case it may be marketed as CADD *computer-aided design and drafting*.

CAD is used in the design of tools and machinery used in the manufacture of components, and in the drafting and design of all types of buildings, from small residential types (houses) to the largest commercial and industrial structures (hospitals and factories).

CAD is mainly used for detailed engineering of 3D models and/or 2D drawings of physical components, but it is also used throughout the engineering process from conceptual design and layout of products, through strength and dynamic analysis of assemblies to definition of manufacturing methods of components.

CAD has become an especially important technology within the scope of computer-aided technologies, with benefits such as lower product development costs and a greatly shortened design cycle. CAD enables designers to lay out and develop work on screen, print it out and save it for future editing, saving time on their drawings.

2. Establishing the Credibility of CFD Simulations

Essential steps toward establishing credibility in computational fluid dynamics (CFD) simulations are outlined, and a vision for the process of systematic collaborative validation that is open to public scrutiny via the Internet is suggested. It begins with an exposition of the elements of CFD simulations and reviews protocols useful for establishing credibility. The various sources of uncertainty in CFD, which include the skills of the user, are presented. Lessons learned from collective verification and validation exercises done in the past are surveyed and lead to our suggestion for a systematic validation process that requires the creation and use of a detailed flow taxonomy for the given application field. The code validator uses the taxonomy and an electronic database to carry out the validation process. This database archives but also gives easy access to trustworthy data and allows full public discussion and scrutiny of the information, comparisons, and hypotheses so that judgments and conclusions about the validation may be accepted or rejected by the scientific community at large.

3. Ground-Based Test Programs

Ground-Based Midcourse Defense (GMD) is a component of the national missile defense strategy of the United States administered by the U.S. Missile Defense Agency. Previously known as National Missile Defense (NMD), the name was changed in 2002 to differentiate it from other missile defense programs, such as space-based and sea-based intercept programs, and defense targeting the boost phase and the reentry phase (see flight phases).

4. Flight-Test Programs

There are typically two categories of flight test programs commercial and military. Commercial flight testing is conducted to certify that the aircraft meets all applicable safety and performance requirements of the government certifying agency. In the US, this is the Federal Aviation Administration (FAA); in Canada, Transport Canada (TC); in the United Kingdom (UK), the Civil Aviation Authority; and in the European Union, the Joint Aviation Authorities (JAA). Since commercial aircraft development is normally funded by the aircraft manufacturer and/or private investors, the certifying agency does not have a stake in the commercial success of the aircraft. These civil agencies are concerned with the aircrafts safety and that the pilots flight manual accurately reports the aircrafts performance. The market will determine the aircrafts suitability to operators. Normally, the civil certification agency does not get involved in flight testing until the manufacturer has found and fixed any development issues and is ready to seek certification.

4.1. Civil Aircraft Flight Test

There are typically two categories of flight test programs commercial and military. Commercial flight testing is conducted to certify that the aircraft meets all applicable safety and performance requirements of the government certifying agency. In the US, this is the Federal Aviation Administration (FAA); in Canada, Transport Canada (TC); in the United Kingdom (UK), the Civil Aviation Authority; and in the European Union, the Joint Aviation Authorities (JAA). Since commercial aircraft development is normally funded by the aircraft manufacturer and/or private investors, the certifying agency does not have a stake in the commercial success of the aircraft. These civil agencies are concerned with the aircrafts safety and that the pilots flight manual accurately reports the aircrafts performance. The market will determine the aircrafts suitability to operators. Normally, the civil certification agency does not get involved in flight testing until the manufacturer has found and fixed any development issues and is ready to seek certification.

4.2. Military aircraft Flight Test

Military programs differ from commercial in that the government contracts with the aircraft manufacturer to design and build an aircraft to meet specific mission capabilities. These performance requirements are documented to the manufacturer in the Aircraft Specification and the details of the flight test program (among many other program requirements) are spelled out in the Statement of Work. In this case, the government is the customer and has a direct stake in the aircrafts ability to perform the mission. Since the government is funding the program, it is more involved in the aircraft design and testing from early-on. Often military test pilots and engineers are integrated as part of the manufacturers flight test team, even before first flight. The final phase of the military aircraft flight test is the Operational Test (OT). OT is conducted by a government-only test team with the dictate to certify that the aircraft is suitable and effective to carry out the intended mission. Flight testing of military aircraft is often conducted at military flight test facilities. The US Navy tests aircraft at Naval Air Station Patuxent River, MD (a.k.a. Pax River) and the US Air Force at Edwards Air Force Base, CA. The U.S. Air Force Test Pilot School and the U.S. Naval Test Pilot School are the programs designed to teach military test personnel. In the UK most military flight testing is conducted by three organisations, the RAF, BAE Systems and Qinetiq. For minor upgrades the testing may be conducted by one of these three organisations in isolation, but major programs are normally conducted by a joint trials team (JTT), with all three organisations working together under the umbrella of an Integrated Project Team (IPT)

4.3. Flight Test Processes

Flight Testing is highly expensive and potentially very risky. Unforeseen problems can lead to damage to aircraft and loss of life, both of aircrew and people on the ground. For these reasons modern flight testing is probably one of the most safety conscious professions today. Flight trials can be divided into 3 sections, planning, execution and analysis and reporting.

4.4. Preparation

For both commercial and military aircraft, flight test preparation begins well before the aircraft is ready to fly. Initially requirements for flight testing must be defined, from which the Flight Test Engineers prepare the test plan(s). These will include the aircraft configuration, data requirements and maneuvers to be flown or systems to be exercised. A full certification/qualification flight test program for a new aircraft will require testing for many aircraft systems and in-flight regimes; each is typically documented in a separate test plan. During the actual flight testing, similar maneuvers from all test plans are combined and the data collected on the same flights, where practical. This allows the required data to be acquired in the minimum number of flight hours.

Once the flight test data requirements are established, the aircraft is instrumented to record that data for analysis. Typical instrumentation parameters recorded during a flight test are: temperatures, pressures, structural loads, vibration/accelerations, noise levels (interior and exterior), aircraft performance parameters (airspeed, altitude, etc.), aircraft controls positions (stick/yoke position, rudder pedal position, throttle position, etc.), engine performance parameters, and atmospheric conditions. During selected phases of flight test, especially during early development of a new aircraft, many parameters are transmitted to the ground during the flight and monitored by the Flight Test Engineer and test support engineers. This provides for safety monitoring and allows real-time analysis of the data being acquired.