INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal, Hyderabad -500 043

AERONAUTICAL ENGINEERING

Course Name	**AEROSPACE PROPULSION AND COMBUSTION**
Course Code	**AAE551**
Programme	B. Tech
Year	2019-20
Semester	VI
Course Coordinator	Mr. M Vijay Kumar, Assistant Professor, AE
Course Faculty	Mr. M Vijay Kumar, Assistant Professor, AE

COURSE OBJECTIVES:

I. Demonstrate with an overview of various aerospace propulsion systems and a sound foundation in the fundamentals of thermodynamics.
II. Distinguish the elementary principles of thermodynamic cycles as applied to propulsion analysis.
III. Prioritize an introduction to combustion& gas kinetic theory.
IV. Discover the knowledge of working knowledge of and the tools to measure various flight propulsion systems such as turbojets, turbofans, ramjets, rockets, air turbo-rockets and nuclear/electric propulsion systems.

COURSE OUTCOMES (COs):

I. Gain knowledge about power plants and aircraft engines performance
II. Assess the importance of various types engine components used in the aircraft
III. Obtain an insight in the concept of propellers, inlets and various nozzles in aircraft
IV. Assess the significance of combustion inside the engines and its performance
V. Estimate the flammability limits, premixed flames and their significance in the combustion zone

COURSE LEARNING OUTCOMES (CLOs):

1. Apply knowledge and understand the essential facts, concepts and principles of
2. Understand the basic function of all aircraft engine components and how they work.Analyze the engine performance parameters and parameters influencing them..
3. Analyze classification of aircraft propulsion.Demonstrate different types aircraft engine operating principle.
4. Demonstrate different type’s aircraft engine operating principle.Understand steps involved in performance analysis of all aircraft engine.
5. Understand step by step procedure of engine parametric cycle analysis.Analyze diffuser performance, losses in it and their impact on engine performance.
6. Describe principle of operation of axial and centrifugal compressor.Understand different types of combustion chamber and functions of all the components.
7. Understand different design of compressor and limitations of each method.Describe theory of flow in isentropic nozzle and physics behind nozzle operation.
8. Analyze performance characteristics of axial and centrifugal turbines.Describe principle of operation of axial and centrifugal compressor.
9. Analyze propeller performance and its types and explain their impact on engine

performance.Analyze performance characteristics of axial and centrifugal compressor.
10. Describe operational modes of subsonic inlets and parameters influencing it.
11. Describe theory of flow in isentropic nozzle and physics behind nozzle operation.
12. Understand different nozzle operating conditions for convergent and divergent nozzle.
13. Understand different types of combustion chamber and functions of all the components.
14. Describe the effect of operating variables on performance.
15. Analyze combustion chamber performance and parameters influencing them.
16. Describe the effect of flame tube cooling and its applications.
17. Understand different types of premixed flames.
18. Explain the significance of flammability limits during combustion process.
19. Describe theory of droplet combustion and turbulent combustion.
20. Analyze the numerical methods of LNS & DNS and explain the parameters influencing them.

SYLLABUS

UNIT-I	ELEMENTS OF AIRCRAFT PROPULSION	Classes: 10
Classification of power plants, methods of aircraft propulsion, propulsive efficiency, specific fuel consumption, thrust and power, factors affecting thrust and power, illustration of working of gas turbine engine, characteristics of turboprop, turbofan and turbojet engines and performance.		
UNIT-II	**COMPONENTS OF JET ENGINES**	**Classes: 08**
Ram jet, scram jet engines construction and nomenclature, theory and performance, methods of thrust augmentation, atmospheric properties, introduction to compressors, turbines, combustors and after burners for aircraft engines.		
UNIT-III	**INLETS, NOZZLES AND PROPERLLER THEROY**	**Classes: 10**
Propeller performance parameters, negative thrust, prop fans, ducted propellers, propeller noise, propeller selection, propeller charts. Subsonic and supersonic inlets, relation between minimum area ratio and external deceleration ratio. Starting problem in supersonic inlets, modes of inlet operation, jet nozzle, efficiencies, over expanded, under and optimum expansion in nozzles, thrust reversal.		
UNIT-IV	**THERMODYNAMICS OF REACTING SYSTEMS**	**Classes: 09**
Classification of combustion chambers, combustion chamber performance, flame tube cooling, flame stabilization, effect of operating variables on performance.		
UNIT-V	**PREMIXED FLAMES**	**Classes: 08**
Rankinehugoniot relations, theories of laminar premixed flame propagation, quenching and flammability limits; Diffusion flames: Burke-Schumann theory, laminar jet diffusion flame, droplet combustion, turbulent combustion, closure problem, premixed and non-premixed turbulent combustion, introduction to DNS and LES.		
Text Books:		
1. Stephen R. Turns, "An Introduction to Combustion", McGraw-Hill, 3rd Edition, 2012. 2. Thomas A. Ward, "Aerospace Propulsion Systems", John Wiley and Sons, 1st Edition, 2010.		
Reference Books:		
1. M. H. Sadd, "Elasticity: Theory, Applications, and Numerics", Academic Press, 2nd Edition, 2009. 2. R. G. Budynas "Advanced Strength and Applied Stress Analysis", McGraw-Hill, 2nd Edition, 1999. 3. A.P. Boresi, R.J. Schmidt, "Advanced Mechanics of Materials", John Willey & Sons, 5th Edition, 2003.		

ELEMENTS OF AIRCRAFT PROPULSION

UNIT-I

Propulsion (Lat. pro-pellere, push forward) is making a body to move (against natural forces), fighting against the natural tendency of relative-motion to decay. Motion is relative to an environment. Sometimes, propulsion is identified with thrust, the force pushing a body to move against natural forces, and one might say that propulsion is thrust (but thrust not necessary implies motion, as when pushing against a wall; on the other hand, propulsion implies thrust).

Sometimes a distinction is made from propulsion (pushing) to traction (pulling), but, leaving aside internal stresses in the system (compression in the first case and tension in the latter), push and pull motions produces the same effect: making a body to move against natural forces. In other occasions 'traction' is restricted to propulsion by shear forces on solid surfaces.

The special case of creating aerodynamic thrust to just balance gravitational attraction of a vehicle without solid contact (e.g. hovering helicopters, hovering VTOL-aircraft, hovering rockets, hovering hovercraft...), is often included as propulsion (it is usually based on the same engine), though its propulsion efficiency iszero.

Operational envelopes

Each engine type will operate only within a certain range of altitudes and Mach numbers (velocities). Similar limitations in velocity and altitude exist for airframes. It is necessary, therefore, to match airframe and propulsion system capabilities. Figure1.1 shows the approximate velocity and altitude limits, or corridor of flight, within which airlift vehicles can operate. The corridor is bounded by a lift limit, a temperature limit, ·and an aerodynamic force limit. The lift limit is determined by the maximum level-flight altitude at a given velocity. The temperature limit is set by the structural thermal limits of the material used in construction of the · aircraft. At any given altitude, the maximum velocity attained is temperature-limited by aerodynamic heating effects. At lower altitudes, velocity is limited by aerodynamic force loads rather than bytemperature.

The operating regions of all aircraft lie within the flight corridor. The operating region of a particular aircraft within the corridor is determined by aircraft design, but it is a very small portion of the overall corridor. Superimposed on "the flight corridor in Fig.1.1 is the operational envelopes of various powered aircraft. The operational limits of each propulsion system are determined by limitations of the components of the propulsion system and are shown in Fig. 1.2.

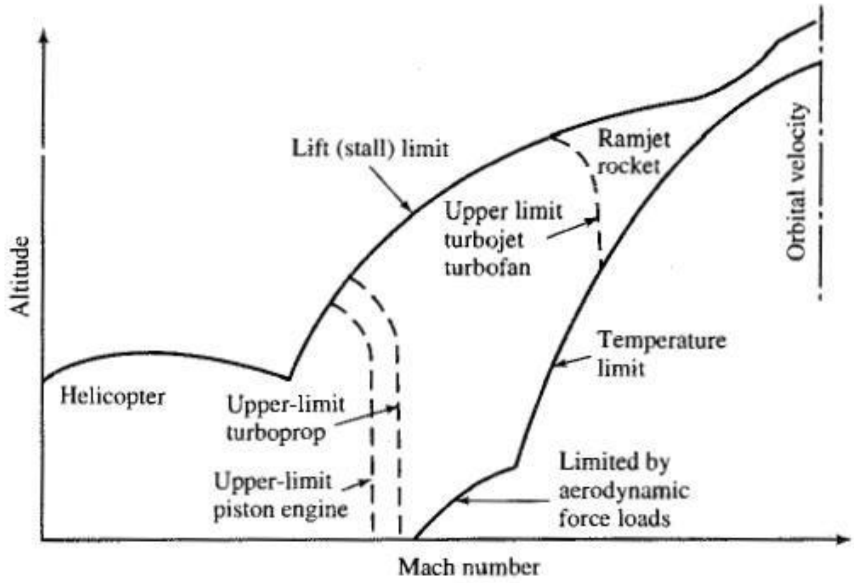


Fig.1.1: Flight limits.

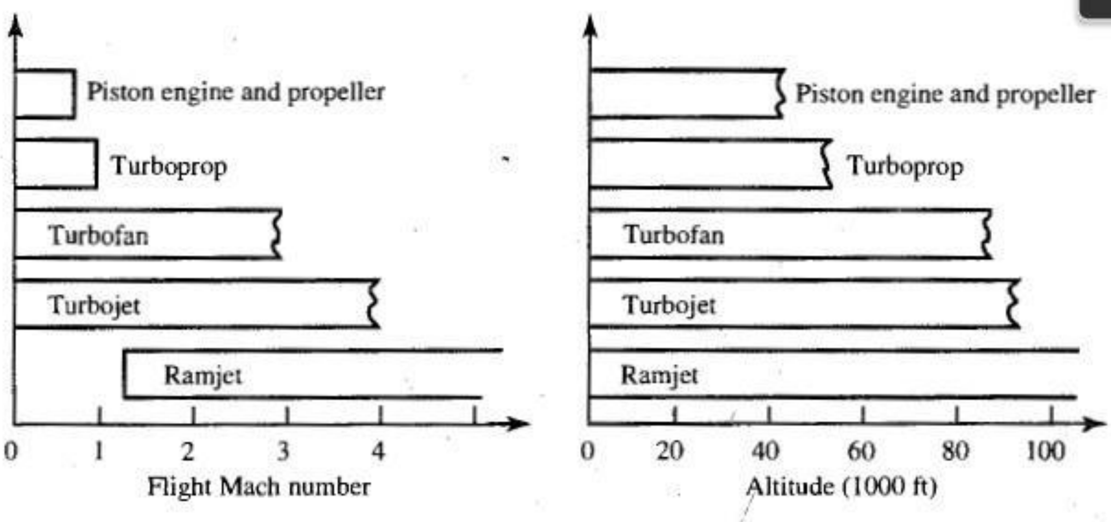


Fig.1.2: Engine operational limits.

Classification of air breathing engine

The air breathing engine classified into following types:

1. turbojet,
2. turbofan,
3. turboprop,
4. ramjet
5. Scramjet

Gas Generator

The "heart" of a gas turbine type of engine is the gas generator. A schematic diagram of a gas generator is shown in Fig. 1.3. The compressor, combustor, and turbine are the major components of the gas generator which is common to the turbojet, turbofan, turboprop, and turbo shaft engines. The purpose of a gas generator is to supply high-temperature and high-pressure gas.

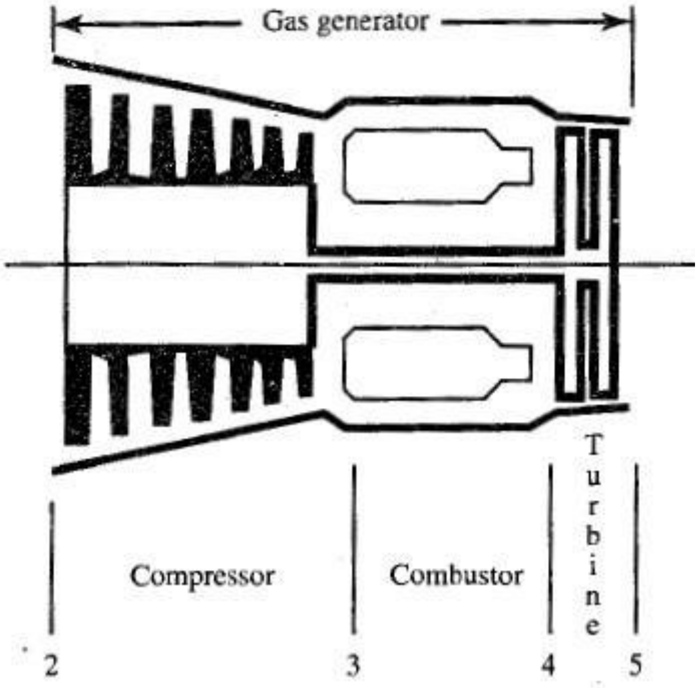


Fig.1.3: Schematic diagram of gas generator.

Turbojet engine

By adding an inlet and a nozzle to the gas generator, a turbojet engine can be constructed. A schematic diagram of a afterburner is shown Fig. 1.4.The thrust of a turbojet is developed by compressing air in the inlet and compressor, mixing the air with fuel and burning in the combustor, and expanding the gas stream through the turbine and nozzle. The expansion of gas through the turbine supplies the power to turn the compressor. The net thrust delivered by the engine is the result of converting internal energy to kineticenergy.

The different processes in a turbojet cycle are the following:

A-1: Air from far upstream is brought to the air intake (diffuser) with some acceleration/deceleration

1-2: Air is decelerated as is passes through the diffuser

2-3: Air is compressed in a compressor (axial or centrifugal)

3-4: The air is heated using a combustion chamber/burner

4-5: The air is expanded in a turbine to obtain power to drive the compressor

5-6 : The air may or may not be further heated in an afterburner by adding further fuel
6-7: The air is accelerated and exhausted through the nozzle to producethrust.

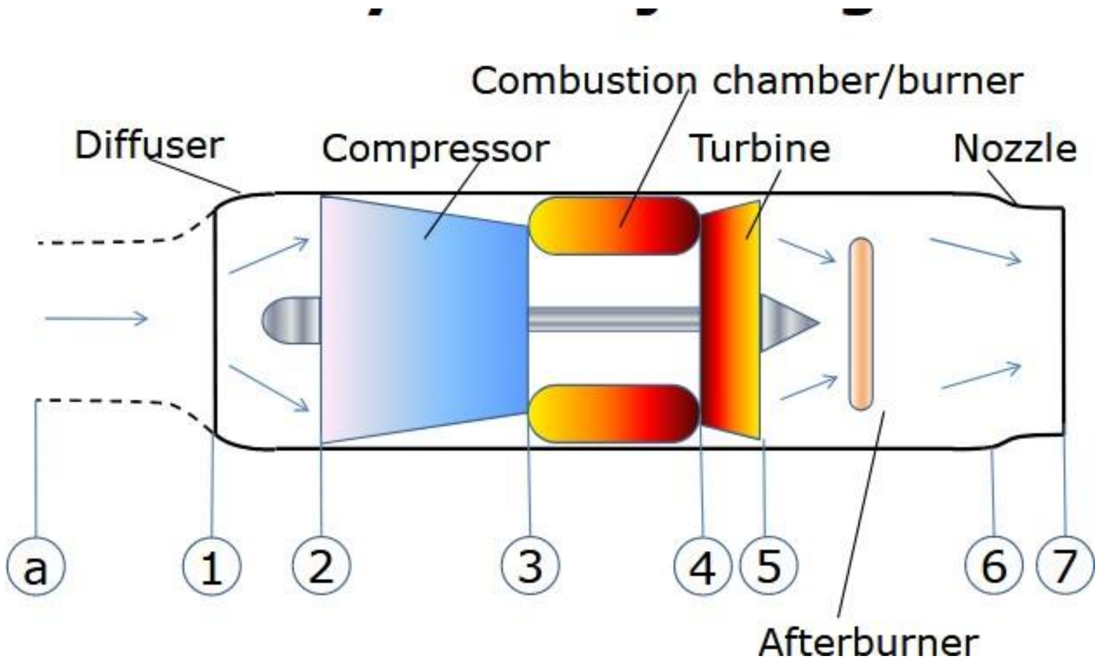


Fig.1.4: Turbo jet engine with after burner

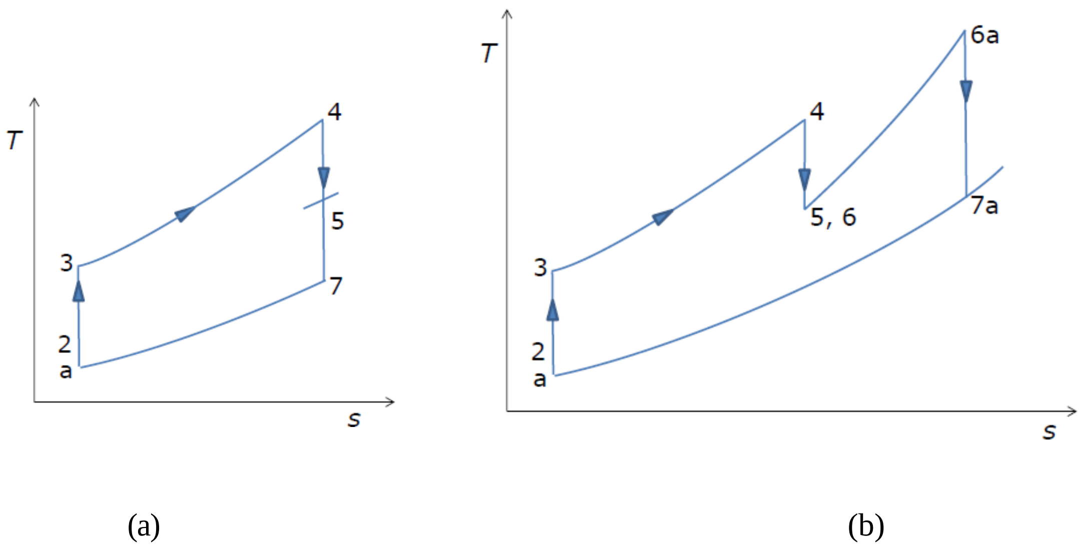


Fig.1.5: (a) Ideal turbojet cycle (without afterburning). (b) Ideal turbojet cycle with afterburning

Afterburning: used when the aircraft needs a substantial increment in thrust. fore.g.to accelerate to and cruise at supersonic speeds. Since the air-fuel ratio in gas turbine engines are much greater than the stoichiometric values, there is sufficient amount of air available for combustion at the turbine exit. There are no rotating components like a turbine in the afterburner; the temperatures can be taken too much higher values than that at turbine entry.

Variation of flow properties across jet engine

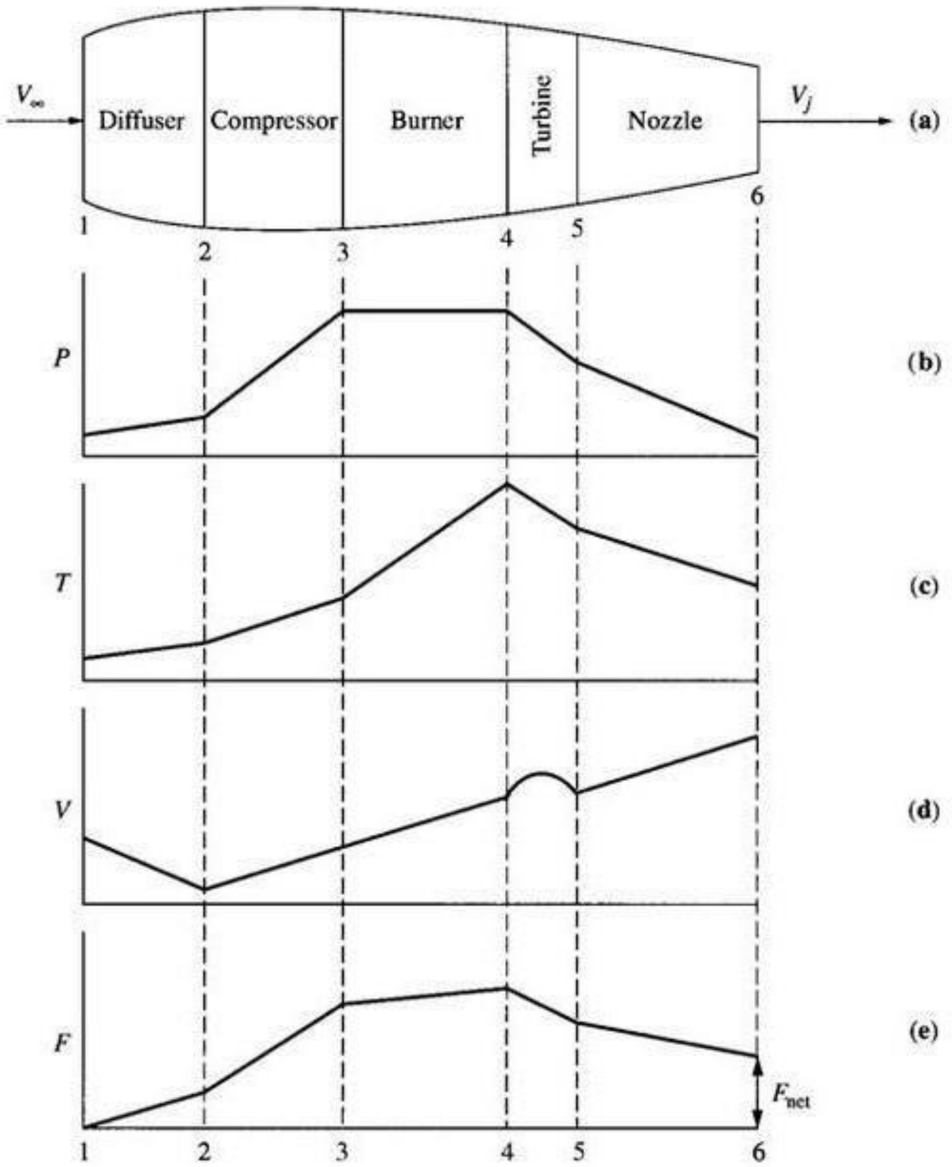


Fig.1.6: Variation of flow properties across turbojet engine

The pressure, temperature, and velocity variations through a jet engine are shown in Fig. 1.6. In the compressor section, the pressure and temperature increase as a result of work being done on the air. The temperature of the gas is further increased by burning in the combustor. In the turbine section, energy is being removed from the gas stream and converted to shaft power to turn the compressor. The energy is removed by an expansion process which results in a decrease of temperature and pressure. In the nozzle, the gas stream is further expanded to produce a high exit kinetic energy. All the sections of the engine must operate in such a way as to efficiently produce the greatest amount of thrust for a minimum of weight.

Turbo fan engine

The turbofan engine consists of an inlet, fan, gas generator, and nozzle. A schematic diagram of a turbofan is shown in Fig. 17. In the turbofan, a portion of the turbine work is used to supply power to the fan. Generally the turbofan engine is more economical and efficient than the turbojet engine in" a limited realm of flight. The thrust specific fuel consumption (TSFC or fuel mass flow rate per unit thrust) is lower for turbofans and indicates a more economical operation. The turbofan also accelerates a larger mass of air to a lower velocity than a turbojet for a higher propulsive efficiency. The frontal area of a turbofan is quite large compared to that of a turbojet, and for this reason more drag and more weight result. The fan diameter is also limited aerodynamically when compressibility -effectsoccur.

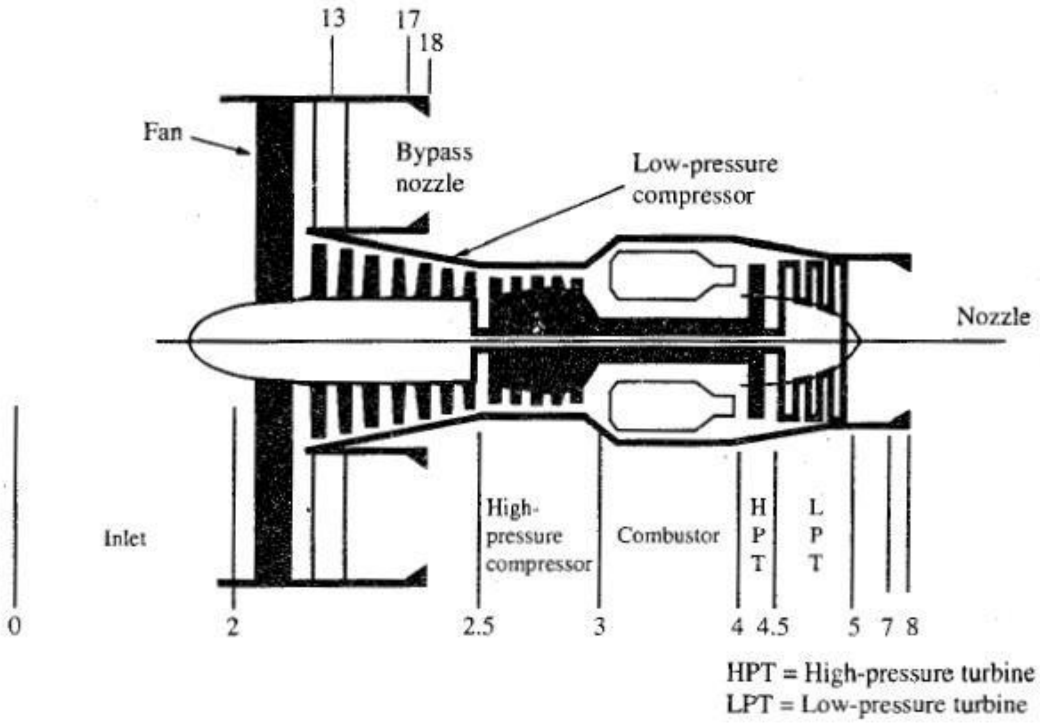


Fig.1.7: Schematic diagram of a high-bypass-ratio turbofan.

Bypass ratio

The bypass ratio (BPR) of a turbofan engine is the ratio between the mass flow rates of the bypass stream to the mass flow rate entering the core. A 10:1 bypass ratio, for example, means that 10 kg of air passes through the bypass duct for every 1 kg of air passing through the core.

The Turboprop and Turbo shaft engine

A gas generator that drives a propeller is a turboprop engine. The expansion of gas through the turbine supplies the energy required to turn the propeller. A schematic diagram of the turboprop is shown in Fig. 1.8. The turbo shaft engine is similar to the turboprop except that power is supplied to a shaft rather than a propeller. The turbo shaft engine is used quite extensively for supplying power for helicopters. The turboprop engine may find applicationin VTOL (vertical takeoff and landing) transporters. The limitations and advantages of the turboprop are those of the propeller. For low-speed flight and, short-field takeoff, the propeller has a performance advantage. At speeds approaching the speed of sound, compressibility effects set in and the propeller loses its aerodynamic efficiency. Due to the rotation of the propeller, the propeller tip will approach the speed of sound before the vehicle approaches the speed of sound. This compressibility effect when one approaches the speed of sound limits the design of helicopter rotors and propellers. At high subsonic speeds, the turbofan engine will have a better aerodynamic performance than the turboprop since the turbofan is essentially a ducted turboprop. Putting a duct or shroud around a propeller increases its aerodynamicperformance.

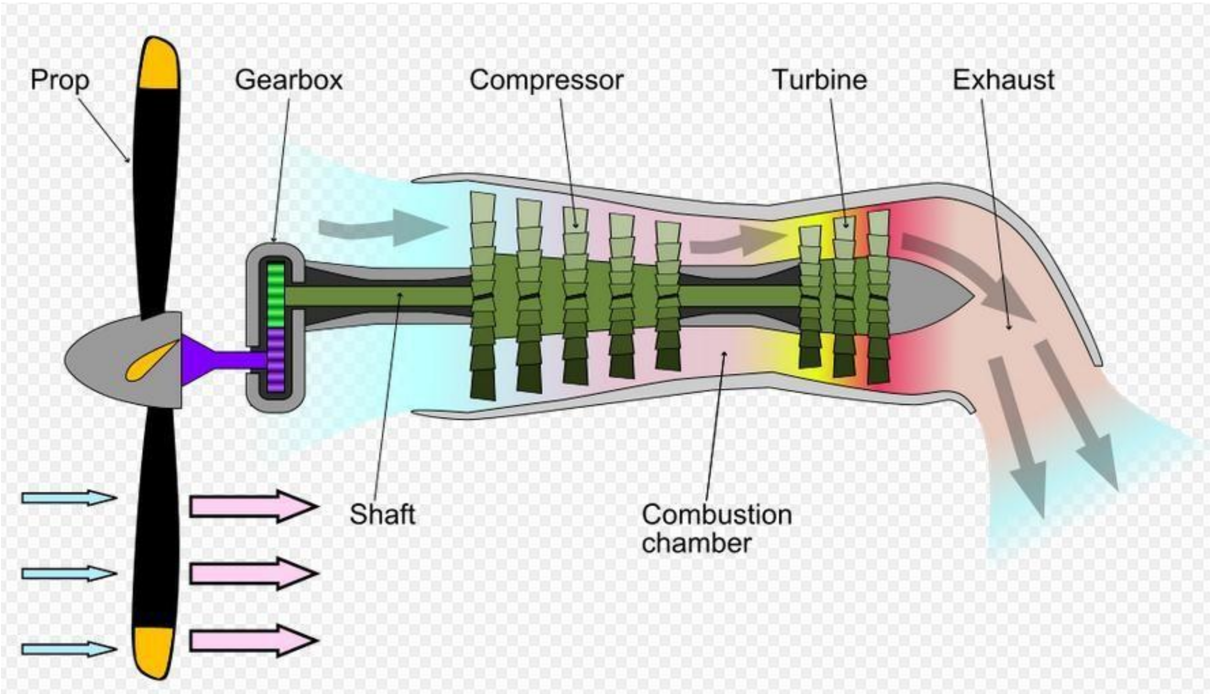


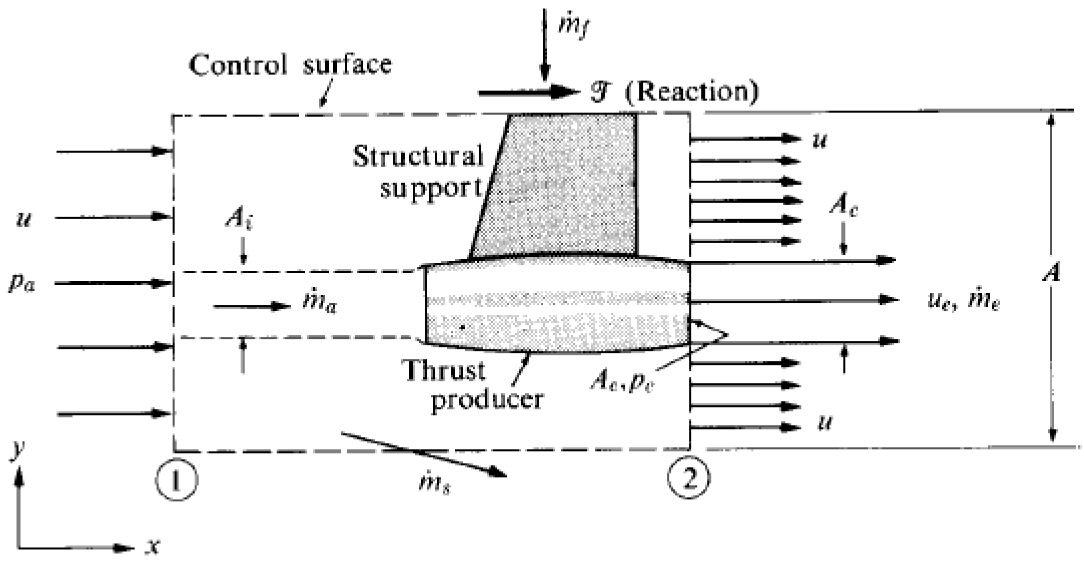
Fig.1.8: Schematic diagram of a turboprop.

The thrust equation

The following assumptions are made:

1. The flow is steady within the control volume; thus all the properties within the control do not change withtime.

2. The external flow is reversible; thus the pressures and velocities are constant over the control surface except over the exhaust area Pe of theengine**.**



$$\mathcal{T} = \dot{m}_a[(1 + f)u_e - u] + (p_e - p_a)A_e.$$

Fig.1.12: Generalized thrust equation

Engine performance parameters

The engine performance is described by different efficiency definitions, thrust and the fuel consumption. The efficiency definitions that we shall now be discussing are applicable to an engine with a single propellant stream (turbojets or ramjets). For other types of jet engines (turbofan, turboprop) the equations need to be appropriately modified**.**

Thermal efficiency: The ratio of the rate of production of propellant kinetic energy to the total energy consumption rate.

$$\eta_{th} = \frac{\dot{m}_a\left[(1+f)(u_e^2/2) - u^2/2\right]}{\dot{m}_f Q_R} = \frac{\left[(1+f)(u_e^2/2) - u^2/2\right]}{f Q_R}$$

For a turboprop or turbo shaft engine, the output is largely shaft power. In this case,

$$\eta_{th} = \frac{P_s}{\dot{m}_f Q_R}$$

Propulsion efficiency: The ratio of thrust power to the rate of production of propellant kinetic energy. The propulsive efficiency ηp can be defined as the ratio of the useful propulsive energy or thrust power (F. u) to the sum of that energy and the unused kinetic energy of thejet.

$$\eta_P = \frac{\Im u}{\dot{m}_a\left[(1+f)(u_e^2/2) - u^2/2\right]}$$

If we assume that f«1 and the pressure thrust term is negligible

$$\eta_P = \frac{(u_e - u)u}{u_e^2/2 - u^2/2} = \frac{2u/u_e}{1+u/u_e}$$

Overall efficiency: The product of thermal efficiency and propulsion efficiency.

$$\eta_o = \eta_p \eta_{th}$$

Specific Thrust

Specific thrust is a term used in gas turbine engineering to show the relative thrust per air mass flow rate of a jet engine (e.g. turbojet, turbofan, etc.) and is defined as the ratio: net thrust/total intake airflow.

Why are we interested in specific thrust?

First, it is an indication of engine efficiency. Two different engines have different values of specific thrust. The engine with the higher value of specific thrust is more efficient because it produces more thrust for the same amount of airflow.

It gives us an easy way to "size" an engine during preliminary analysis. The result of our thermodynamic analysis is a certain value of specific thrust. The aircraft drag defines the required value of thrust. Dividing the thrust required by the specific thrust tells us how much airflow our engine must produce and this determines the physical size of the engine.

Specific impulse

Specific impulse (usually abbreviated Isp) is a measure of how effectively a rocket uses propellant or jetengine uses fuel. By definition, it is the total impulse (or change in momentum) delivered per unit of propellant consumed.

$$I_{\mathrm{sp}} \equiv \frac{F}{\dot{m}_{\mathrm{p}} g}$$

$$TSFC = \frac{\dot{m}_f}{\Im} \approx \frac{\dot{m}_f}{\dot{m}_a\left[(1+f)u_e - u\right]}$$

Thrust Specific fuel consumptionTSFC or SFC for thrust engines (e.g. turbojets, turbofans, ramjets, rocket engines, etc.) is the mass of fuel needed to provide the net thrust for a given period.SFC varies with throttle setting, altitude and climate. For jet engines, flight speed also has a significant effect upon SFC; SFC is roughly proportional to air speed.

Problem 1: A gas turbine operating at a pressure ratio of 11.314 produces zero net work output when 473.35 kJ of heat is added per kg of air. If the inlet air temperature is 300 K and the turbine efficiency if 71%, find the compressor efficiency.

Since the net work output is zero

$$w_c = w_t$$

$$\text{or, } T_{02} - T_{01} = T_{03} - T_{04}$$

$$T_{03} - T_{02} = T_{04} - T_{01}$$

$$\frac{T_{02s}}{T_{01}} = \left(\frac{P_{02}}{P_{01}}\right)^{(\gamma-1)/\gamma} = 11.314^{0.4/1.4}$$

$$T_{02s} = 300 \times 11.314^{0.4/1.4} = 600K$$

Given that heat added $= 476.35$ kJ/kg

$$c_p(T_{03} - T_{02}) = 476.354$$

$$or,\ T_{03} - T_{02} = 474K$$

We know that

$$T_{04} = T_{01} + (T_{03} - T_{02}) = 300 + 474 = 774K$$

The turbine efficiency is 71%

$$0.71 = \frac{T_{03}(1 - T_{04}/T_{03})}{T_{04s}(T_{03}/T_{04s} - 1)} \text{ and } T_{03}/T_{04s} = 11.314^{0.4/1.4}$$

$$\therefore \frac{T_{04}}{T_{03}} = 1 - \frac{0.71}{2} = 0.645$$

$$or, T_{03} = 774/0.645 = 1200K \text{ and } T_{02} = 1200 - 474 = 726K$$

$$\therefore \eta = \frac{T_{02s} - T_{01}}{T_{02} - T_{01}} = \frac{600 - 300}{726 - 300} = 0.704 \ or\ 70.4\%$$

Problem 2: An aircraft flies at a Mach number of 0.75 ingesting airflow of 80 kg/s at an altitude where the ambient temperature and pressure are 222 K and 10 kPa, respectively. The inlet design is such that the Mach number at the entry to the inlet is 0.60 and that at the compressor face is 0.40. The inlet has an isentropic efficiency of 0.95. Find (a) the area of theinlet entry (b) the inlet pressure recovery (c) the compressor facediameter.

Mach number is 0.75, hence, the flight speed is

$$u_a = M_a\sqrt{\gamma R T_a} = 0.75\sqrt{1.4\times 287\times 222} = 224\ m/s$$

$$\rho_a = P_a / RT_a = 0.1569\ kg/m^3$$

The total temperature, $T_{0a} = T_a\left(1+\frac{\gamma-1}{2}M_a^2\right) = 246\,K$

Total pressure, $P_{0a} = P_a\left(1+\frac{\gamma-1}{2}M_a^2\right)^{\gamma/(\gamma-1)} = 14522.8\ Pa$

$\therefore$ Static temperature at inlet entry,

$$T_1 = T_{0a} / \left(1+\frac{\gamma-1}{2}M_a^2\right) = 230.4\ K$$

Static pressure at inlet entry,

$$P_1 = P_{0a} / \left(1+\frac{\gamma-1}{2}M_a^2\right)^{\gamma/(\gamma-1)} = 11386\ Pa$$

$$\rho_1 = P_1 / RT_1 = 0.1722\ kg/m^3$$

Therefore, area at the inlet entry, $A_1 = \frac{\dot{m}}{u_1\rho_1} = \frac{\dot{m}}{M_1\sqrt{\gamma R T_1}\rho_1}$

$$= \frac{80}{0.6\sqrt{1.4\times 287\times 230.4}\times 0.1722}$$

$$= 2.54\ m^2$$

Now, $T_{02} = T_{0a}$

Diffuser efficiency, $\eta_d = \frac{T_{02s} - T_a}{T_{0a} - T_a}$

Substituting the values, $T_{02s} = 245.75K$

$\therefore$ Pressure recovery, $\frac{P_{02}}{P_{01}} = \left(\frac{T_{02s}}{T_{01}}\right)^{\gamma/(\gamma-1)} = 0.982$

The static pressure at the compressor face, T_2

$$T_2 = T_{02} / \left[1+\frac{\gamma-1}{2}M_2^2\right] = 239.3\,K$$

$$P_2 = P_1\left(T_2/T_1\right)^{\gamma/(\gamma-1)} = 13001\ Pa$$

$$\rho_2 = P_2 / RT_2 = 0.1893\, kg / m^3$$

Velocity at the compressor face, $u_2 = M_2 \sqrt{\gamma R T_2}$

$$= 124.03\ m / s$$

Area of the compressor face, $A_2 = \dot{m} / u_2 \rho_2$

$$= 3.407\, m^2$$

$\therefore$ the diameter, d = 2.08 m

Problem 3: Aturbojet engine operates at an altitude where the ambient temperature and pressure are 216.7 K and 24.444 kPa, respectively. The flight Mach number is 0.9 and the inlet conditions to the convergent nozzle are 1000 K and 60 kPa. If the nozzle efficiency is 0.98, the ratio of specific heat is 1.33, determine whether the nozzle is operating under choked condition or not. Determine the nozzle exitpressure.

The nozzle efficiency is defined as

$$\eta_n = \frac{h_{06} - h_7}{h_{06} - h_{7s}} = \frac{T_{06} - T_7}{T_{06} - T_{7s}} = \frac{1 - T_7 / T_{06}}{1 - T_{7s} / T_{06}} = \frac{1 - T_7 / T_{07}}{1 - T_{7s} / T_{06}}$$

Under choked condition, M = 1,

$$\therefore \eta_n = \frac{1 - (2 / (\gamma + 1))}{1 - (P_c / P_{06})^{(\gamma - 1)/\gamma}}$$

$$or, \frac{P_{06}}{P_c} = \frac{1}{(1 - (1/\eta_n)((\gamma - 1)/(\gamma + 1)))^{\gamma/(\gamma - 1)}}$$

Substituting the values,

$$\frac{P_{06}}{P_c} = 1.878$$

$$\text{Also,} \frac{P_{06}}{P_a} = \frac{60}{24.444} = 2.45 Pa$$

We can see that $P_c > P_a$

Therefore, the nozzle is operating under choked condition.

The exit pressure would therefore be equal to

$$P_e = \frac{60}{1.878} = 31.95 kPa$$

UNIT-II
COMPONENTS OF JET ENGINES

The Ramjet

The ramjet engine consists of an inlet, a combustion zone, and a nozzle. A schematic diagram of a ramjet is shown in Fig. 1.9. The ramjet does not have the compressor and turbine as the turbojet does. Air enters the inlet where it is compressed and then enters the combustion zone where it is mixed with the fuel and burned. The hot gases are then expelled through the nozzle, developing thrust. The operation of the ramjet depends upon the inlet to decelerate the incoming air to raise the pressure in the combustion zone. The pressure rise makes it possible for the ramjet to operate. The higher the velocity of the incoming air, the greater the pressure rise. It is for this reason that the ramjet operates best at high supersonic velocities. At subsonic velocities, the ramjet is inefficient, and to start the ramjet, air at a relatively higher velocity must enter theinlet.

The combustion process in an ordinary ramjet takes place at low subsonic velocities. At high supersonic flight velocities, a very large pressure rise is developed that is more than sufficient to support operation of the ramjet. Also, if the inlet has to decelerate a supersonic high-velocity airstream to a subsonic velocity, large pressure losses can result. The deceleration process also produces a temperature rise, and at some limiting flight speed, the temperature will approach the limit set by the wall materials and cooling methods. Thus when the temperature increase due to deceleration reaches the limit, it may not be possible to burn fuel in the airstream.

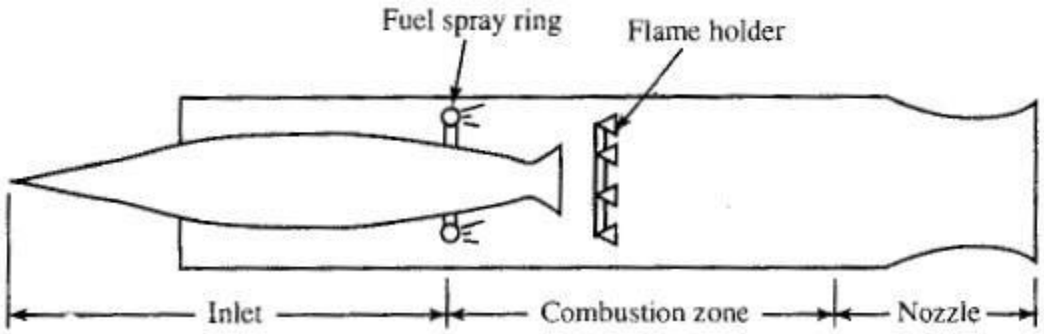


Fig.1.9: Schematic diagram of a ramjet.

SCRAMJET

The ramjet with supersonic combustion is known as the scramjet (supersonic combustion ramjet). By using a supersonic combustion process, the temperature rise and pressure loss due to deceleration in the inlet can be reduced.

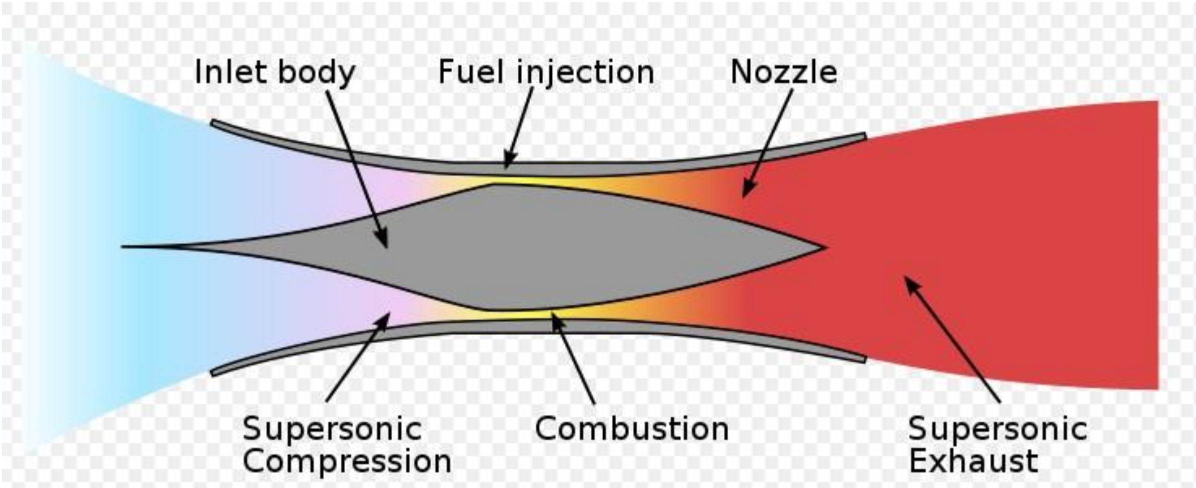


Fig.1.10: Schematic diagram of a Scramjet.

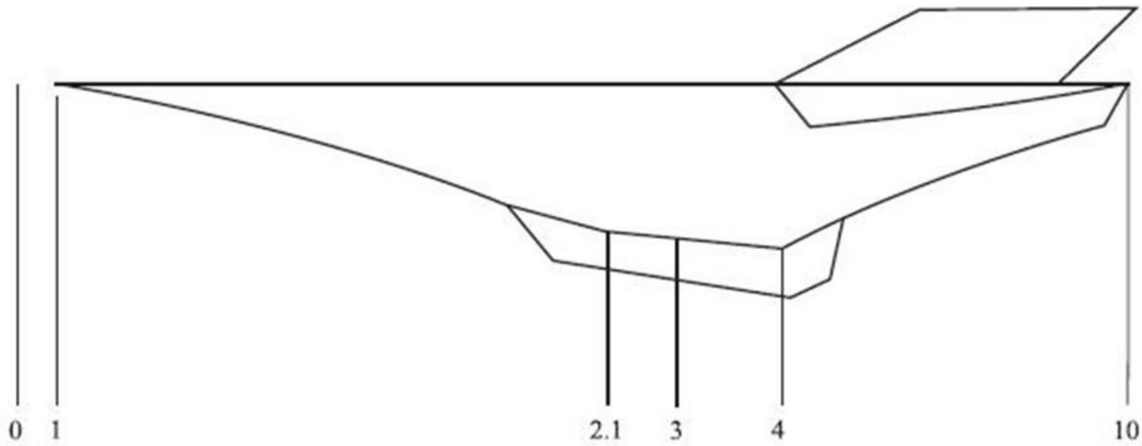


Fig.1.11: Schematic diagram of a Scramjet with station number.

The scramjet engine belongs to the family of Brayton cycles, which consist of two adiabatic and two constant-pressure processes. A simplified schematic of a scramjet equipped vehicles shown above Fig.1.10.

Station 0 represents the free-stream condition.

Station 1 represents the beginning of the compression process. Hypersonic shock-wave angles are small, resulting in long compression ramps (or spikes if an ax symmetric configuration is used) that, in many of the suggested configurations, begin at the vehicle‘s leading edge. Additional compression takes place inside the inletduct.

Station 2.1 represents the entrance into the isolator section. The role of the isolator is to separate the inlet from the adverse effects of a pressure rise that is due to combustion in the combustion chamber. The presence of a shock train in the isolator further compresses the air before arriving at the combustion chamber. Thermodynamically the isolator is not a desirable component, because it is a source of additional pressure losses, increases the engine cooling loads, and adds to the engine weight. However, operationally it is needed to include a shock train that adjusts such that it fulfills the role just described.

Station 3 is the combustion chamber entrance. Unlike the turbojet engine cycle, in which the air compression ratio is controlled by the compressor settings, in a fixed-geometry scramjet the pressure at the combustion chamber entrance varies over a large range.

Station 4 is the combustion chamber exit and the beginning of expansion.

Station 10 is the exit from the nozzle; because of the large expansion ratios the entire aft part of the vehicle may be part of the engine nozzle.

Components performance
Air intake performance

Inlet losses arise due to wall friction and shock waves (in a supersonic inlet).

•These result in a reduction in totalpressure.

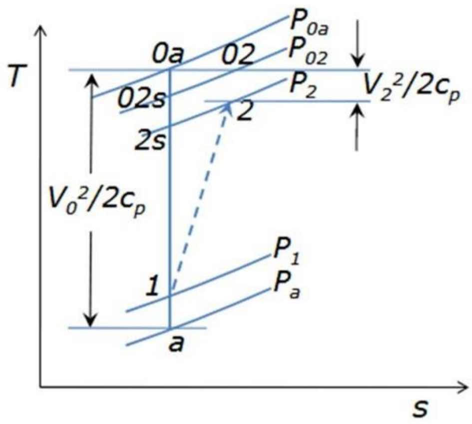
•The flow is usually adiabatic as it flows through theintake.

•Performance of intakes ischaracterized using total pressure ratio and isentropicefficiency. Isentropic efficiency, ηd, of the diffuseris

$$\eta_d = \frac{h_{02s} - h_a}{h_{0a} - h_a} \cong \frac{T_{02s} - T_a}{T_{0a} - T_a}$$

This efficiency can be related to the total pressure ratio (πd) and Mach number

$$\eta_d = \frac{\left(1 + \frac{\gamma - 1}{2} M^2\right) \pi_d^{(\gamma-1)/\gamma} - 1}{[(\gamma - 1)/2] M^2}$$

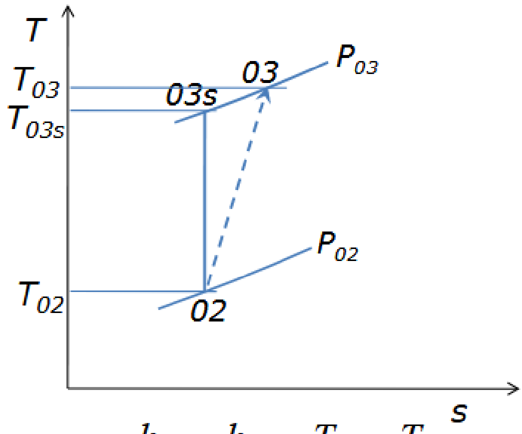


Compressor/fan performance

Compressors are to a high degree of approximation, adiabatic. Compressor performance is evaluated using the isentropic efficiency.

$$\eta_C = \frac{\text{Ideal work of compression for given pressure ratio}}{\text{Actual work of compression for given pressure ratio}}$$

$$= \frac{w_{ci}}{w_c} = \frac{h_{03s} - h_{02}}{h_{03} - h_{02}}$$



$$\eta_C = \frac{h_{03s} - h_{02}}{h_{03} - h_{02}} \simeq \frac{T_{03s} - T_{02}}{T_{03} - T_{02}}$$

$$= \frac{T_{03s}/T_{02} - 1}{T_{03}/T_{02} - 1} = \frac{(P_{03}/P_{02})^{(\gamma-1)/\gamma} - 1}{\tau_C - 1}$$

$$= \frac{(\pi_C)^{(\gamma-1)/\gamma} - 1}{\tau_C - 1}$$

The isentropic efficiency is thus a function of the total pressure ratio and the total temperature ratio.

Combustion chamberperformance

In a combustion chamber (or burner), there are two possibilities of losses, incomplete combustion and total pressure losses.

•Combustion efficiency can be defined by carrying out an energy balance across the combustor.

•Two different values of specific heat at constant pressure: one for fluid upstream of the combustor and the other for fluid downstream of the combustor

Combustion efficiency, η_b

$$\eta_b = \frac{(\dot{m}+\dot{m}_f)h_{04} - \dot{m}h_{03}}{\dot{m}_f \dot{Q}_f} = \frac{(\dot{m}+\dot{m}_f)c_{p4}T_{04} - \dot{m}c_{p3}T_{03}}{\dot{m}_f \dot{Q}_f}$$

$$= \frac{(\dot{m}+\dot{m}_f)c_{pg}T_{04} - \dot{m}c_{pa}T_{03}}{\dot{m}_f \dot{Q}_f}$$

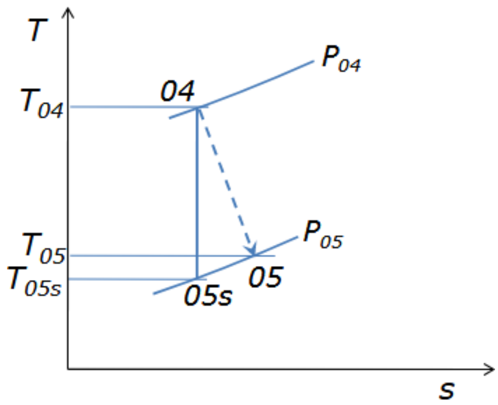
Total pressure losses arise from two effects:

–viscous losses in the combustion chamber

–total pressure loss due to combustion at finite Mach number.

Turbine performance

The flow in a turbine is also assumed to be adiabatic, though in actual engines there could be turbine blade cooling. Isentropic efficiency of the turbine is defined in a manner similar to that of the compressor.

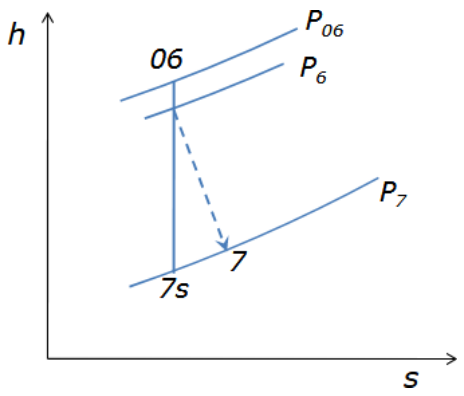


$$\eta_t = \frac{\text{Actual work of compression for given pressure ratio}}{\text{Ideal work of compression for given pressure ratio}}$$

$$= \frac{w_t}{w_{ti}} = \frac{h_{04} - h_{05}}{h_{04} - h_{05s}} = \frac{1-\tau_t}{1-\pi_t^{(\gamma-1)/\gamma}}$$

Nozzle performance

The flow in the nozzle is also adiabatic. However losses in a nozzle could occur due to incomplete expansion (under or over-expansion). Friction may reduce the isentropic efficiency.



The efficiency is defined by

$$\eta_n = \frac{h_{06} - h_7}{h_{06} - h_{7s}}$$

Afterburner performance

Afterburner is thermodynamically similar to a combustion chamber. The performance parameters for an afterburner are thus the combustion efficiency and the total pressure loss. In case of engines with afterburning, the corresponding performance parameters for an afterburner need to be taken intoaccount.

Centrifugal compressor Principle of operation

The centrifugal compressor consists essentially of a stationary casing containing a rotating impeller which imparts a high velocity to the air, and a number of fixed diverging passages in which the air is decelerated with a consequent rise in static pressure. The latter process is one of diffusion, and consequently the part of the compressor containing the diverging passages is known as the diffuser.

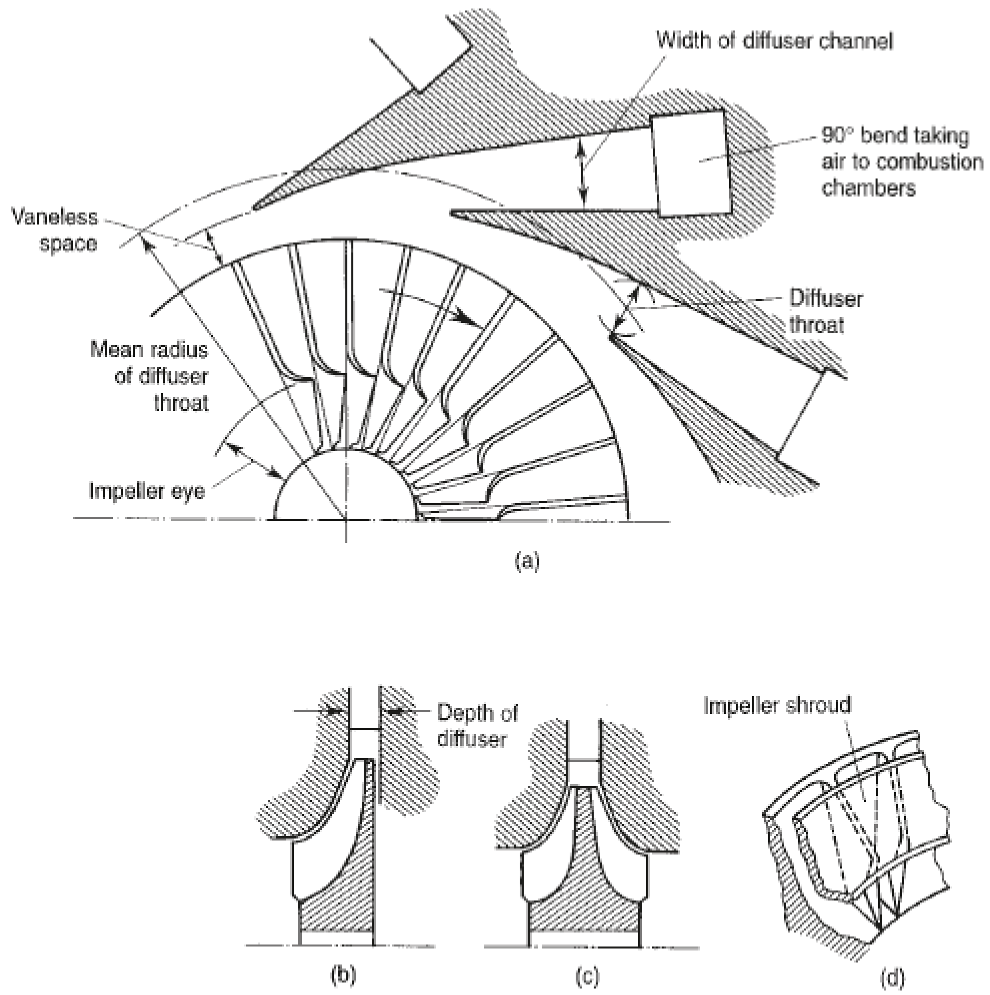


Fig. 4.1 Diagrammatic sketches of centrifugalcompressors

Figure 4.1(a) is a diagrammatic sketch of a centrifugal compressor. The impeller may be single- or double-sided as in Fig. 4.1(b) or (c), but the fundamental theory is the same for both. The double-sided impeller was required in early aero engines because of the relatively small flow capacity of the centrifugal compressor for a given overalldiameter.

Air is sucked into the impeller eye and whirled round at high speed by the vanes on the impeller disc. At any point in the flow of air through the impeller, the centripetal acceleration is obtained by a pressure head, so that the static pressure of the air increases from the eye to the tip of the impeller. The remainder of the static pressure rise is obtained in the diffuser, where the very high velocity of the air leaving the impeller tip is reduced to somewhere in the region of the velocity with which the air enters the impeller eye; it should be appreciated that friction in the diffuser will cause some loss in stagnation pressure. The normal practice is to design the compressor so that about half the pressure rise occurs in the impeller and half in the diffuser.

It will be appreciated that owing to the action of the vanes in carrying the air around with the impeller, there will be a slightly higher static pressure on the forward face of a vane than on the trailing face. The air will thus tend to flow round the edges of the vanes in the clearance space between the impeller and the casing. This naturally results in a loss of efficiency, and the clearance must be kept as small as possible. A shroud attached to the vanes, Fig. 4.1(d), would eliminate such a loss, but the manufacturing difficulties are vastly increased and there would be a disc friction or windage loss associated with the shroud. Although shrouds have been used on superchargers and process compressors, they are not used on impellers for gas turbines.

Work done and pressure rise

Since no work is done on the air in the diffuser, the energy absorbed by the compressor will be determined by the conditions of the air at the inlet and outlet of the impeller. Figure 4.2 shows the nomenclature employed.

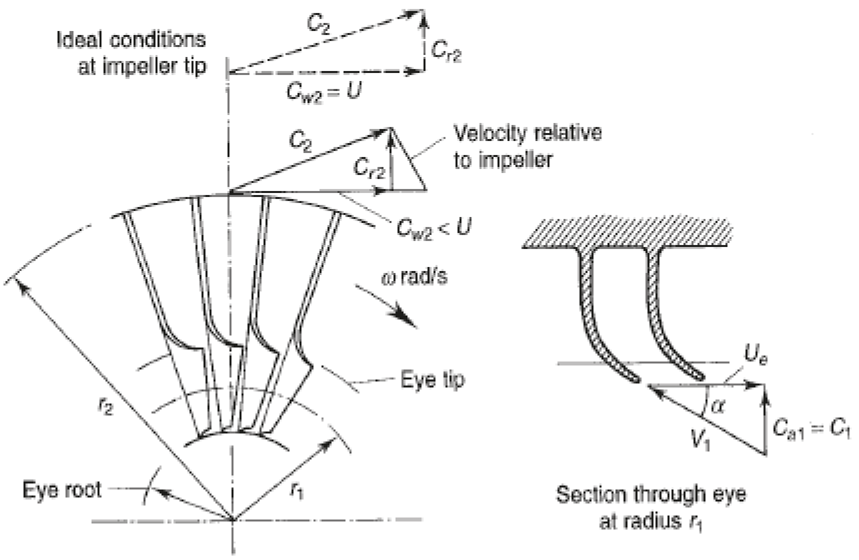


Fig. 4.2 Nomenclature

In the first instance it will be assumed that the air enters the impeller eye in the axial direction, so that the initial angular momentum of the air is zero. The axial portion of the vanes must be curved so that the air can pass smoothly into the eye. The angle which the leading edge of a vane makes with the tangential direction a will be given by the direction of the relative velocity of the air at inlet, V1, as shown in Fig.4.2.

If the air leaves the impeller tip with an absolute velocity C2, it will have a tangential or whirl component Cw2 and a comparatively small radial component Cr2. Under ideal conditions C2 would be such that the whirl component is equal to the impeller tip speed U, as shown by the velocity triangle at the top of Fig. 4.2. Owing to its inertia, the air trapped between the impeller vanes is reluctant to move round with the impeller, and we have already noted that this results in a higher static pressure on the leading face of a vane than on the trailing face. It also prevents the air from acquiring a whirl velocity equal to the impeller speed.This effect is known as slip. How far the whirl velocity at the impeller tip falls short of the tip speed depends largely upon the number of vanes on the impeller. The greater the number of vanes, the smaller the slip, i.e. the more nearly Cw2 approaches U. It is necessary in design to assume a value for the slip factor s, where s is defined as the ratio Cw2>U. Various approximate analyses of the flow in an impeller channel have led to formulae for s: the one appropriate to radial vaned impellers which seems to agree best with experiment is that due to Stanitz.

s = 1 - 0.63p/n

Where n is the number of vanes

Considering unit mass flow of air, this torque is given by

Theoretical torque=Cw2r2 (4.1)

If ω is the angular velocity, the work done on the air willbe

Theoretical work done = Cw_2r_2 ω = Cw2U

Or, introducing the slip factor,

Theoretical work done= sU^2 (4.2)

Owing to friction between the casing and the air carried round by the vanes, and other losses which have a braking effect such as disc friction or ‗windage, the applied torque and therefore the actual work input is greater than this theoretical value. A power input factor c can be introduced to take account of this, so that the actual work done on the air becomes

Work done = $\psi\sigma U^2$ (4.3)

If (T03 - T01) is the stagnation temperature rise across the whole compressor then, since no energy is added in the diffuser, this must be equal to the stagnation temperature rise (T02 - T01) across the impeller alone. It will therefore be equal to the temperature equivalent of the work done on the air given by equation (4.3), namely where cp is the mean specific heat over this temperature range. Typical values for the power input factor liein the region of 1.03591.04.

T03 - T01 = $\psi\sigma U^2/c_p$ (4.4)

So far we have merely considered the work which must be put into the compressor. If a value for the overall isentropic efficiency $\eta_c$is assumed, then it is known how much of the work is usefully employed in raising the pressure of the air. The overallstagnation pressure ratio follows as

$$\frac{p_{03}}{p_{01}} = \left(\frac{T'_{03}}{T_{01}}\right)^{\gamma/(\gamma-1)} = \left[1 + \frac{\eta_c(T_{03} - T_{01})}{T_{01}}\right]^{\gamma/(\gamma-1)} = \left[1 + \frac{\eta_c\psi\sigma U^2}{c_pT_{01}}\right]^{\gamma/(\gamma-1)} \quad (4.5)$$

Axial Flow Compressors

The basic components of an axial flow compressor are a rotor and stator, the former carrying the moving blades and the latter the stationary rows of blades. The stationary blades convert the kinetic energy of the fluid into pressure energy, and also redirect the flow into an angle suitable for entry to the next row of moving blades. Each stage will consist of one rotor row followed by a stator row, but it is usual to provide a row of so called inlet guide vanes. This is an additional stator row upstream of the first stage in the compressor and serves to direct the axially approaching flow correctly into the first row of rotating blades. For a compressor, a row of rotor blades followed by a row of stator blades is called a stage. Two forms of rotor have been taken up, namely drum type and disk type. A disk type rotor illustrated in Figure 4.3 The disk type is used where consideration of low weight is most important. There is a contraction of the flow annulus from the low to the high pressure end of the compressor. This is necessary to maintain the axial velocity at a reasonably constant level throughout the length of the compressor despite the increase in density of air. Figure 4.4 illustrate flow through compressor stages. In an axial compressor, the flow rate tends to be high and pressure rise per stage is low. It also maintains fairly high efficiency.

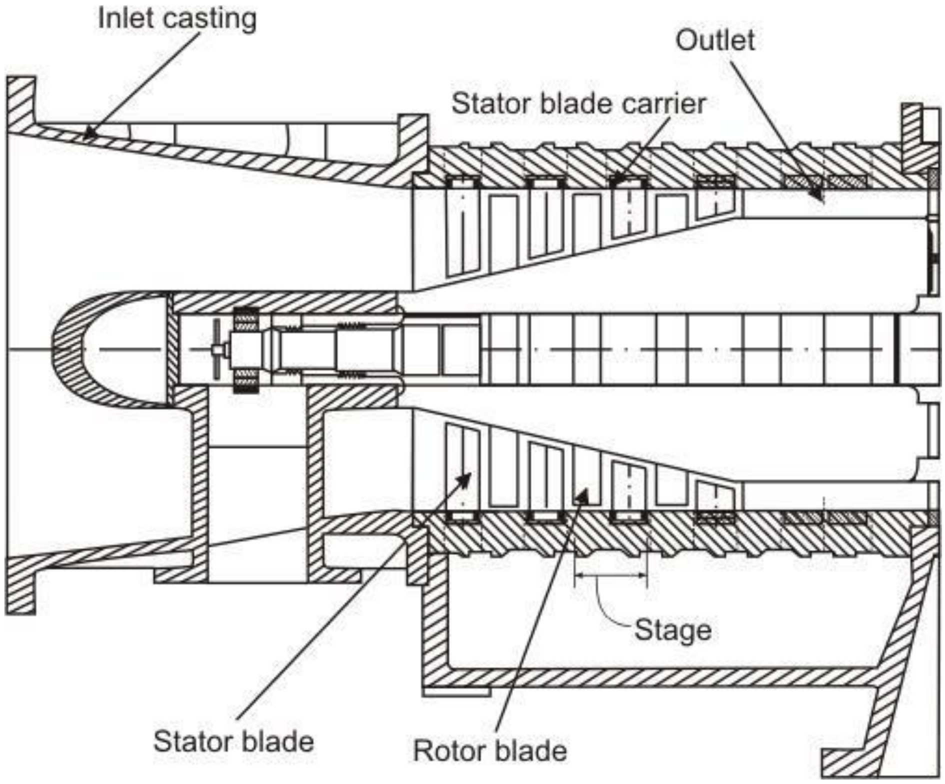


Figure 4.3 Disk type axial flow compressor

The basic principle of acceleration of the working fluid, followed by diffusion to convert acquired kinetic energy into a pressure rise, is applied in the axial compressor. The flow is considered as occurring in a tangential plane at the mean blade height where the blade peripheral velocity is U. This two dimensional approach means that in general the flowvelocity will have two components, one axial and one peripheral denoted by subscript w , implying a whirl velocity. It is first assumed that the air approaches the rotor blades with an absolute velocity, V_1, atand angle α_1 to the axial direction. In combination with the peripheral velocity U of the blades, its relative velocity will be V_{r1} at and angle β_1 as shown in the upper velocity triangle (Figure 9.3). After passing through the diverging passages formed between the rotor blades which do work on the air and increase its absolute velocity, the air will emerge with the relative velocity of V_{r2} at angle β_2 which is less than β_1. This turning of air towards the axial direction is, as previously mentioned, necessary to provide an increase in the effective flow area and isbrought about by the camber of theblades.Since V_{r2} is less than due to diffusion, some pressure rise has been accomplished in the rotor. The velocityin combinationwith U gives the absolute velocity V_2 at the exit from the

rotorat an angle to the axial direction. The air then passes through the passages formed by the stator blades where it is further diffused to velocity V_3 at an angle α_3 which in most designs equals to α_1 so that it is prepared for entry to next stage. Here again, the turning of the air towards the axial direction is brought about by the camber of theblades.

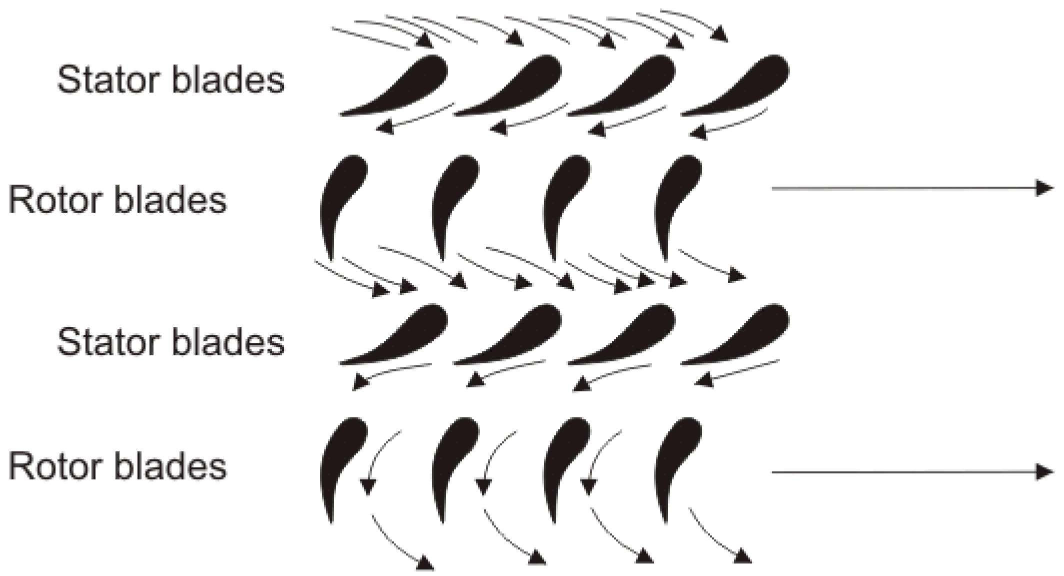


Figure 4.4 Flow through stages

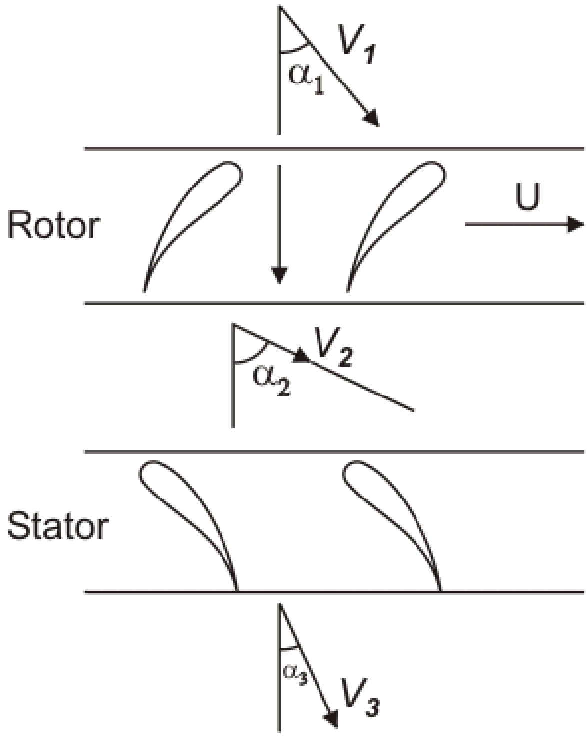
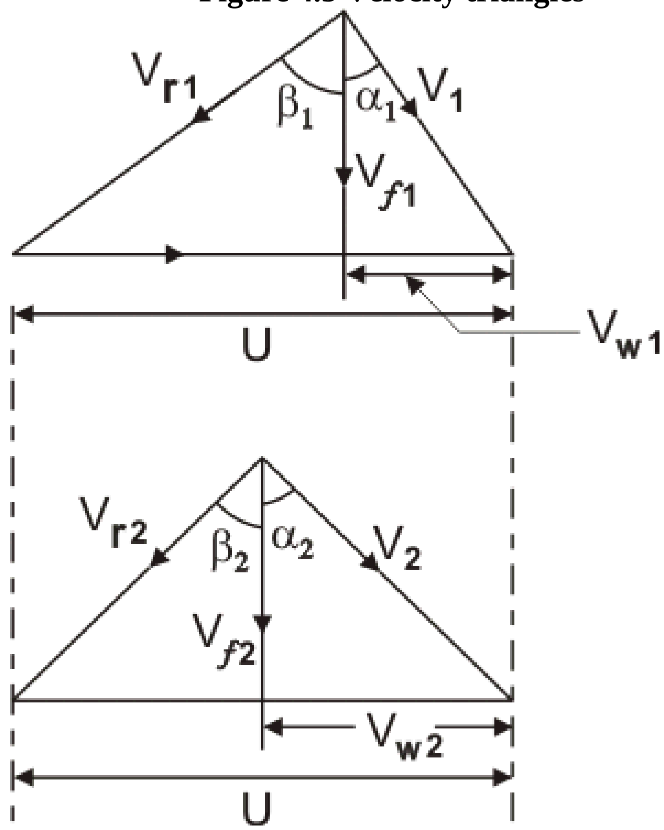


Figure 4.5 Velocity triangles



Two basic equations follow immediately from the geometry of the velocity triangles. These are:

$$\frac{U}{V_f} = \tan\alpha_1 + \tan\beta_1 \qquad (4.6)$$

$$\frac{U}{V_f} = \tan\alpha_2 + \tan\beta_2 \qquad (4.7)$$

In which $V_f = V_{f1} = V_{f2}$ is the axial velocity, assumed constant through the stage. The work done per unit mass or specific work input, w being givenby

$$w = U(V_{w2} - V_{w1}) \qquad (4.8)$$

This expression can be put in terms of the axial velocity and air angles to give

$$w = UV_f(\tan\alpha_2 - \tan\alpha_1) \qquad (4.9)$$

or by using Eqs. (4.6) and (4.7)

$$w = UV_f(\tan\beta_1 - \tan\beta_2) \qquad (4.10)$$

This input energy will be absorbed usefully in raising the pressure and velocity of the air. A part of it will be spent in overcoming various frictional losses. Regardless of the losses, the input will reveal itself as a rise in the stagnation temperature of the air ΔT_0. If the absolute velocity of the air leaving thestage V_3 is made equalto that at the entry. V_1, the stagnation temperature rise ΔT_0 will also bethe static temperature rise of the stage, ΔT_s ,so that

$$\Delta T_0 = \Delta T_s = \frac{UV_f}{c_p}(\tan \beta_1 - \tan \beta_2) \qquad (4.11)$$

In fact, the stage temperature rise will be less than that given in Eq. (4.11) owing to three dimensional effects in the compressor annulus. Experiments show that it is necessary to multiply the right hand side of Eq. (4.11) by a work-done factor λ which is a number less than unity. This is a measure of the ratio of actual work-absorbing capacity of the stage to its ideal value.

The radial distribution of axial velocity is not constant across the annulus but becomes increasingly peaky (Figure. 4.9) as the flow proceeds, settling down to a fixed profile at about the fourth stage. Equation (4.10) can be written with the help of Eq. (4.6) as

$$\begin{aligned} w &= U[(U - V_f \tan \alpha_1) - V_f \tan \beta_2] \\ &= U(U - V_f (\tan \alpha_1 + \tan \beta_2)) \end{aligned} \qquad (4.12)$$

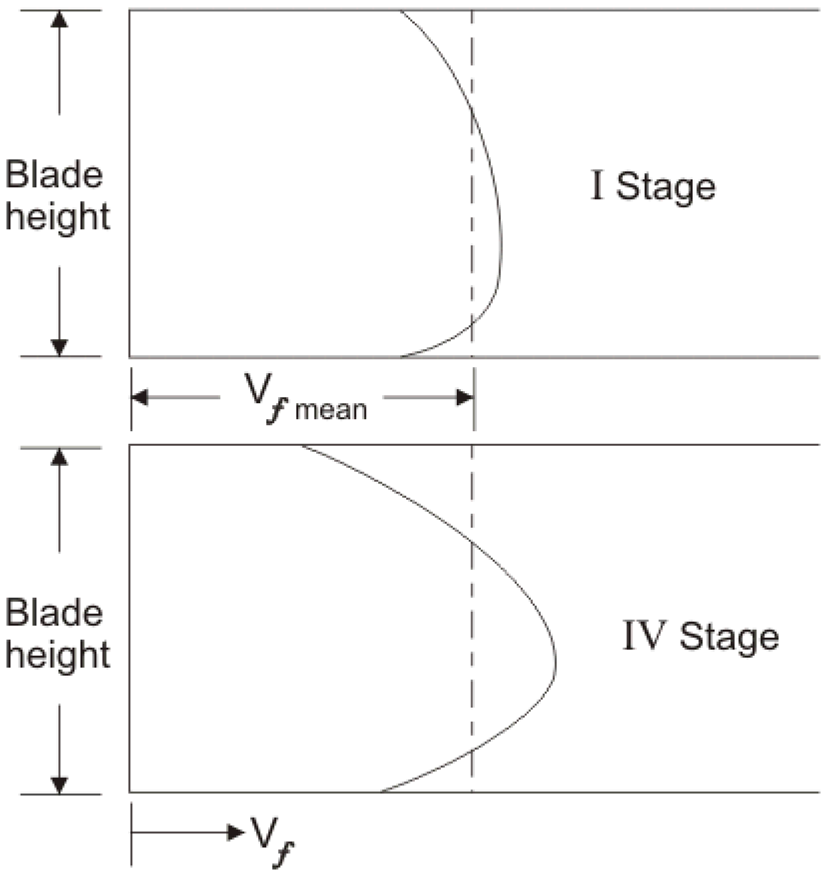


Figure 4.6 Axial velocity distributions

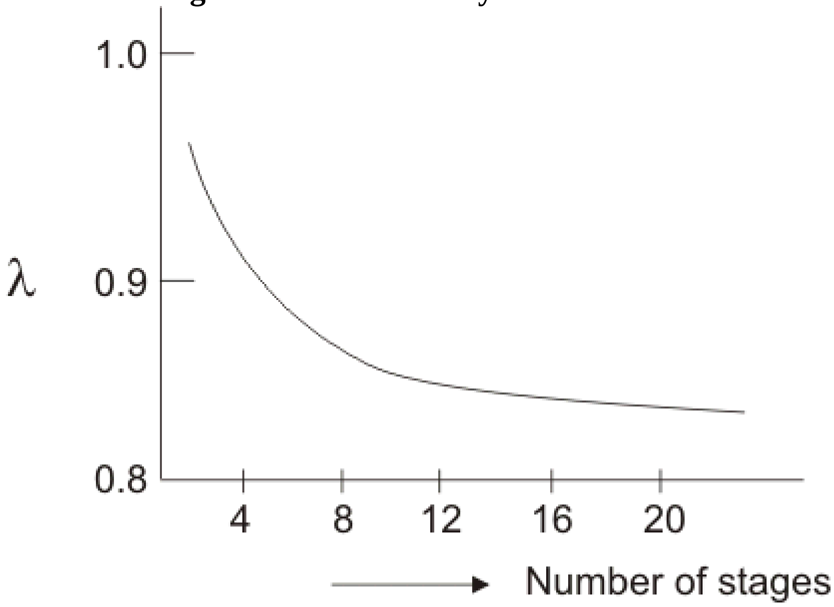


Figure 4.7 Variation of work-done factor with number of stages

Since the outlet angles of the stator and the rotor blades fix the value of α_1 and β_2 hence the value of $(\tan \alpha_1 + \tan \beta_2)$. Any increase in V_f will result in a decrease in and vice ϖ- versa. If the compressor is designed for constant radial distribution of V_f as shown by the dotted line in Figure (4.6), the effect of an increase in V_f in the central region of the annulus will be to reduce the work capacity of blading in that area. However this reduction is somewhat compensated by an increase in ϖ in the regions of the root and tip of the blading because of the reduction of V_f at these parts of the annulus. The net result is a loss in total work capacity because of the adverse effects of blade tip clearance and boundary layers on the annulus walls. This effect becomes more pronounced as the number of stages is increased and the way in which the mean value varies with the number of stages. Thevariationof λ with the number of stages is shown in Figure. 4.7. Care should be taken to avoid confusion of the work done factor with the idea of efficiency. If ϖ is the expression for the specific work input (Equation. 4.8), then $\lambda\varpi$ is the actual amount of work which can be supplied to the stage. The application of an isentropic efficiency to the resulting temperature rise will yield the equivalent isentropic temperature rise from which the stage pressure ratio may be calculated. Thus, the actual stage temperature rise is givenby

$$\Delta T_0 = \frac{\lambda U V_f}{c_p}(\tan \beta_1 - \tan \beta_2) \qquad (4.13)$$

and the pressure ratio R_s by

$$R_s = \left[1 + \frac{\eta_s \Delta T_0}{T_{0_1}}\right]^{\frac{\gamma}{\gamma-1}} \qquad (4.14)$$

Where, T_{0_1} is the inlet stagnation temperature and η_s is the stage isentropicefficiency.

Degree of Reaction

A certain amount of distribution of pressure (a rise in static pressure) takes place as the air passes through the rotor as well as the stator; the rise in pressure through the stage is in general, attributed to both the blade rows. The term degree of reaction is a measure of the extent to which the rotor itself contributes to the increase in the static head of fluid. It is defined as the ratio of the static enthalpy rise in the rotor to that in the whole stage. Variation of c_p over the relevant temperature range will be negligibly small and hence this ratio of enthalpy rise will be equal to the corresponding temperaturerise.

It is useful to obtain a formula for the degree of reaction in terms of the various velocities and air angles associated with the stage. This will be done for the most common case in which it is assumed that the air leaves the stage with the same velocity (absolute) with which it enters ($V_1 = V_3$).

Thisleadsto $\Delta T_s = \Delta T_0$ .If ΔT_A and ΔT_B are the static temperature rises in the rotor and the statorrespectively,

Then from Eqs (4.9),(4.10),(4.11),

$$\begin{aligned} w &= c_p(\Delta T_A + \Delta T_B) = c_p \Delta T_s \\ &= UV_f (\tan \beta_1 - \tan \beta_2) \qquad (4.15) \\ &= UV_f (\tan \alpha_2 - \tan \alpha_1) \end{aligned}$$

Since all the work input to the stage is transferred to air by means of the rotor, the steady flow energy equation yields,

$$w = c_p \Delta T_A + \frac{1}{2}(V_2^2 - V_1^2)$$

With the help of Eq. (4.15), it becomes

$$c_p \Delta T_A = UV_f (\tan \alpha_2 - \tan \alpha_1) - \frac{1}{2}(V_2^2 - V_1^2)$$

But $V_2 = V_f \sec \alpha_2$ and $V_1 = V_f \sec \alpha_1$, andhence

$$c_p\Delta T_A = UV_f(\tan\alpha_2 - \tan\alpha_1) - \frac{1}{2}V_f^2(\sec^2\alpha_2 - \sec^2\alpha_1)$$
$$= UV_f(\tan\alpha_2 - \tan\alpha_1) - \frac{1}{2}V_f^2(\tan^2\alpha_2 - \tan^2\alpha_1) \qquad (4.16)$$

The degree of reaction

$$\Lambda = \frac{\Delta T_A}{\Delta T_A + \Delta T_B} \qquad (4.17)$$

With the help of Eq. (10.2), it becomes

$$\Lambda = \frac{UV_f(\tan\alpha_2 - \tan\alpha_1) - \frac{1}{2}V_f^2(\tan^2\alpha_2 - \tan^2\alpha_1)}{UV_f(\tan\alpha_2 - \tan\alpha_1)}$$

and

$$\Lambda = 1 - \frac{V_f}{2U}(\tan\alpha_2 + \tan\alpha_1)$$

By adding up Eq. (4.6) and Eq. (4.7) we get

$$\frac{2U}{V_f} = \tan\alpha_1 + \tan\beta_1 + \tan\alpha_2 + \tan\beta_2$$

Replacing α_1 and α_2 in theexpressionfor Λ with β_1 and β_2,

$$\Lambda = \frac{V_f}{2U}(\tan\beta_1 + \tan\beta_2) \qquad (4.18)$$

As the case of 50% reaction blading is important in design, it is of interest to see the result for Λ =0.5,

$$\tan\beta_1 + \tan\beta_2 = \frac{U}{V_f}$$

and it follows from Eqs. (4.6) and (4.7) that

$$\tan\alpha_1 = \tan\beta_2, \text{ i.e. } \alpha_1 = \beta_2 \qquad (4.19a)$$

$$\tan\beta_1 = \tan\alpha_2, \text{ i.e. } \beta_1 = \alpha_2 \qquad (4.20b)$$

Furthermore since V_f is constant through thestage.

$$V_f = V_1 \cos \alpha_1 = V_3 \cos \alpha_3$$

And sincewe haveinitially assumedthat $V_3 = V_1$, it followsthat $\alpha_1 = \alpha_3$. Because ofthis equality ofangles,namely, $\alpha_1 = \beta_2 = \alpha_3$ and $\beta_1 = \alpha_2$, blading designed on this basis is sometimes referred to as symmetrical blading .The 50% reaction stage is called a repeating stage.

It is to be remembered that in deriving Eq. (4.18) for Λ, we have implicitly assumed a work done factor λ of unity in making use of Eq. (4.16). A stage designed with symmetrical blading is referred to as 50% reaction stage, although Λ will differ slightly for λ.

Types of Whirl Distribution

The whirl (vortex) distributions normally used in compressor design practice are:

- Freevortex $r\,C_\theta$ =constant
- Forcedvortex C_θ / r =constant
- Constantreaction R =constant
- Exponential $C_{\theta 1} = a - b/r$ (afterstator)
- Free vortex whirl distribution results in highly twisted blades and is not advisable for blades of smallheight.
- The current design practice for transonic compressors is to use constant pressure ratio across thespan.

Elementary Cascade Theory

The previous module dealt with the axial flow compressors, where all the analyses were based on the flow conditions at inlet to and exit from the impeller following kinematics of flow expressed in terms of velocity triangles. However, nothing has been mentioned about layout and design of blades, which are aerofoil sections. In the development of the highly efficient modern axial flow compressor or turbine, the study of the two-dimensional flow through a cascade of aerofoils has played an important part. An array of blades representing the blade ring of actual turbo machinery is called the cascade. Figure 4.8 shows a compressor blade cascade tunnel. As the air stream is passed through the cascade, the direction of air is turned. Pressure and velocity measurements are made at up and downstream of cascade as shown. The cascade is mounted on a turn-table so that its angular direction relative to the direction of inflow can be changed, which enables tests to be made for a range of incidence angle. As the flow passes through the cascade, it is deflected and there will be a circulation and thus the lift generated will be $\rho V_m \Gamma$ (Fig 4.9 & 4.10). V_m isthe mean velocity that makes an angle α_m with the axialdirection

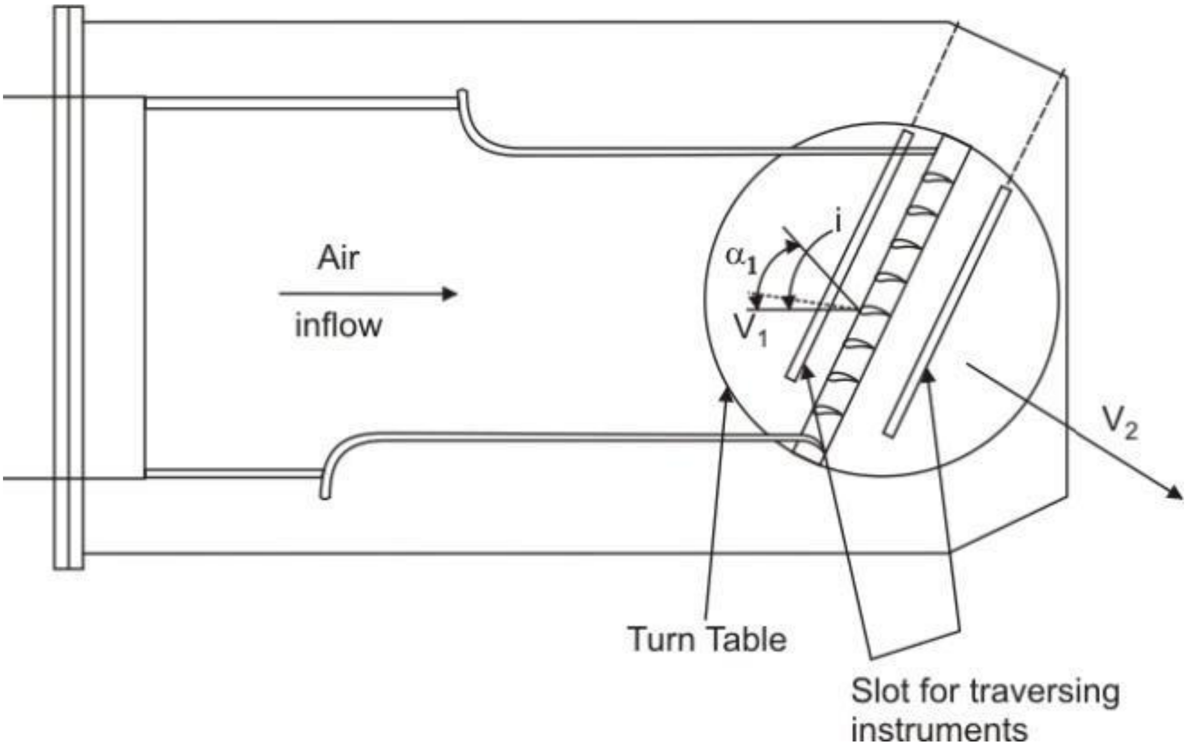


Figure 4.8 Cascade Tunnel

Compressor cascade:

For a compressor cascade, the static pressure will rise across the cascade, i.e. $p_2 > p_1$

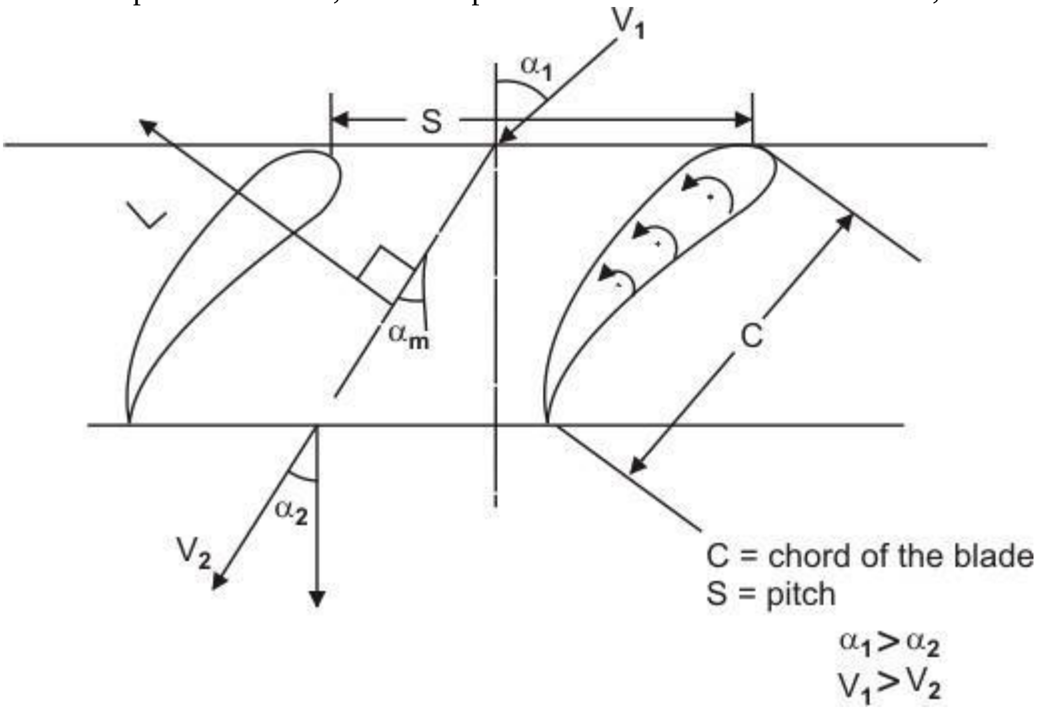


Figure 4.9 Compressor Cascade

C =chord of the blade

S = pitch

$\alpha_1 > \alpha_2$

$V_1 > V_2$

$$\tan \alpha_m = \frac{1}{2}(\tan \alpha_1 + \tan \alpha_2)$$

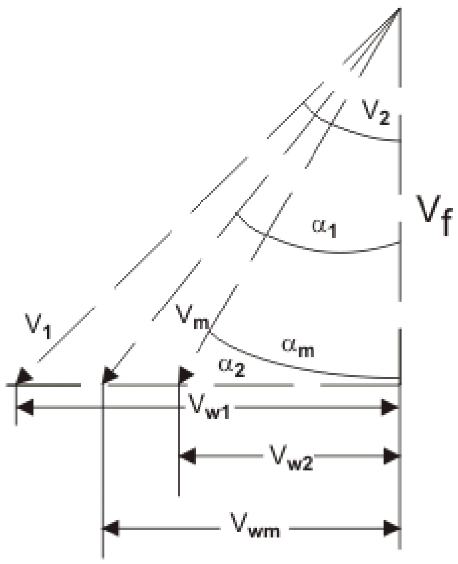


Figure 4.10 Velocity triangle

Circulation:

$$\Gamma = S(V_{w1} - V_{w2})$$

Lift = $\rho V_m \Gamma = \rho V_m S(V_{w1} - V_{w2})$

Lift coefficient, $$C_L = \frac{L}{\frac{1}{2}\rho V_m^2 C} = \frac{\rho V_m S(V_{w1} - V_{w2})}{\frac{1}{2}\rho V_m^2 C}$$

$$= \frac{2S}{C} * \frac{1}{V_m}(V_{w1} - V_{w2})$$

from velocity triangles,

$$V_{w1} = V_f \tan \alpha_1, \quad V_{w2} = V_f \tan \alpha_2$$

$$C_L = 2\frac{S}{C}\left(\frac{V_f}{V_m}\right)(\tan \alpha_1 - \tan \alpha_2)$$

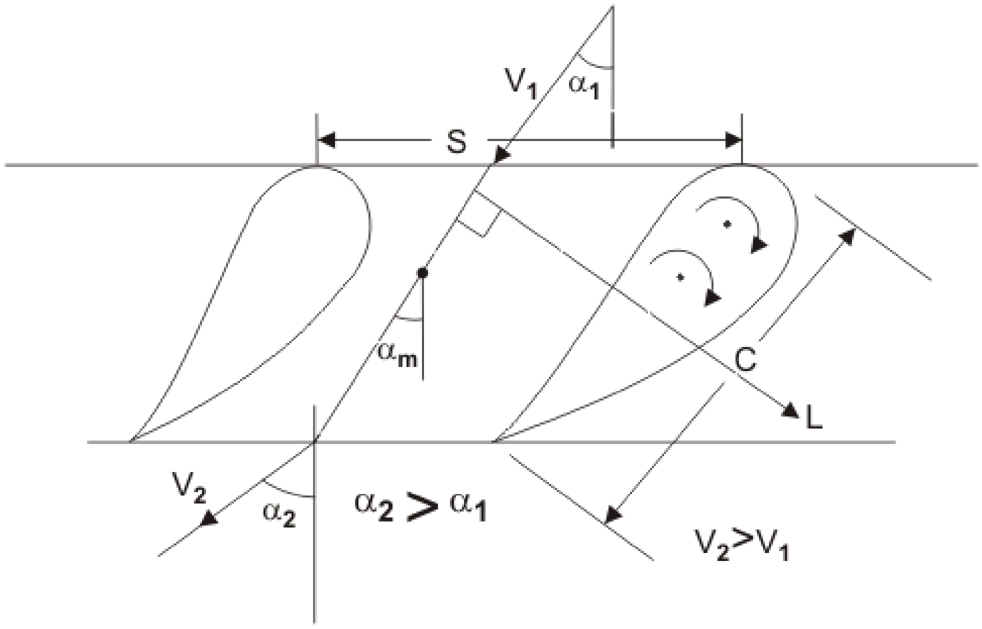
$$= 2\frac{S}{C}(\tan \alpha_1 - \tan \alpha_2)\cos \alpha_m, \ \tan \alpha_m = \frac{\tan \alpha_1 + \tan \alpha_2}{2}$$

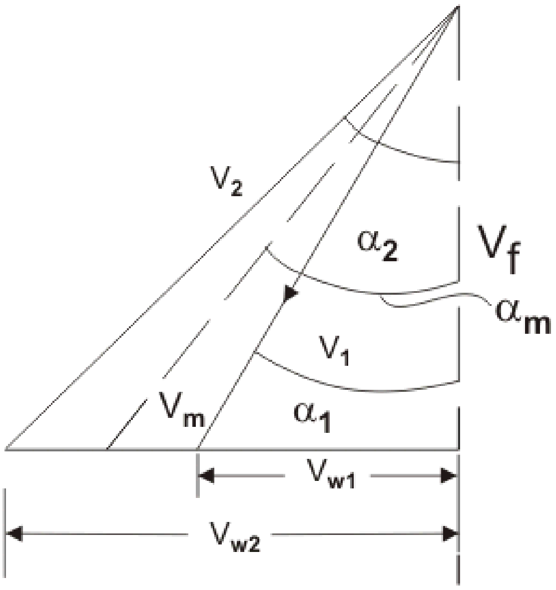
S, C -depend on the design of the cascade

α_1, α_2 - Flowangles at the inlet and outlet

Lift is perpendicular to α_m line

Turbine Cascade: Static pressure will drop across the turbine cascade,i.e. $p_2 < p_1$

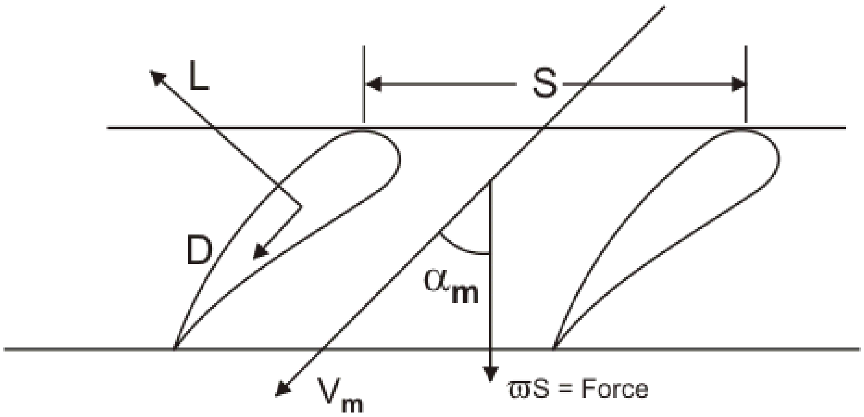




Compressor Cascade (Viscous Case)

In compressor cascade, due to losses intotal pressure $(\tilde{\varpi})$, there will be an axial force $S\tilde{\varpi}$ as shown in figure below. Thus the drag, which is perpendicular to the lift, is definedas

$$D = \tilde{w} S \cos \alpha_m$$



The lift will be reduced due to the effect of drag which can be expressed as:

Effective lift = $\bar{L} = L - \bar{\varpi} S \sin \alpha_m = \rho V_m \Gamma - \bar{\varpi} S \sin \alpha_m$

The lift has decreased due toviscosity, $\bar{L} = \rho V_m \Gamma - D \tan \alpha_m$

Actual lift coefficient

$$C_L = 2 \frac{S}{C} (\tan \alpha_1 - \tan \alpha_2) \cos \alpha_m - C_D \tan \alpha_m$$

$$C_D = \frac{D}{\frac{1}{2}\rho V_m^2 C}$$

Where, C_D =dragcoefficient,

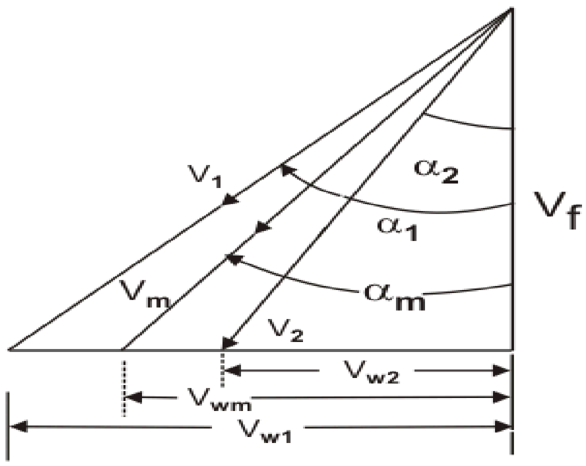
In the case of turbine, drag will contribute to work (and is considered as useful).

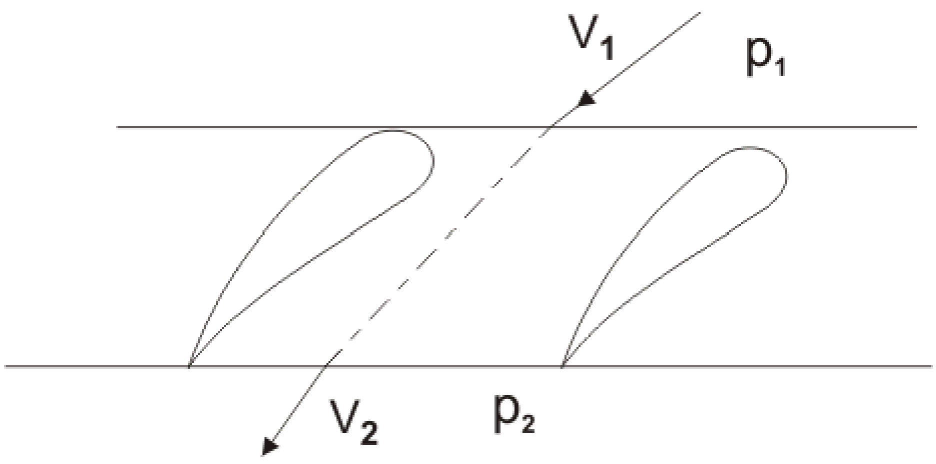
Blade efficiency (or diffusion efficiency)

For a compressor cascade, the blade efficiency is defined as:

$$\eta_b = \frac{\text{Actual rise in static pressure}}{\text{Ideal static pressure rise}}$$

Due to viscous effect, static pressure rise is reduced





$$\eta_b = \frac{(p_2 - p_1)_{\text{ideal}} - \text{loss}}{(p_2 - p_1)_{\text{ideal}}}$$

$$\eta_b = \frac{\frac{\rho}{2}(V_1^2 - V_2^2) - \bar{\varpi}}{\frac{\rho}{2}(V_1^2 - V_2^2)} = 1 - \frac{\bar{\varpi}}{\frac{\rho}{2}(V_1^2 - V_2^2)}$$

from velocity triangle:

$$V_1^2 = V_{w1}^2 + V_f^2 ,\ V_2^2 = V_{w2}^2 + V_f^2$$

$$V_1^2 - V_2^2 = (V_{w1} + V_{w2})(V_{w1} - V_{w2})$$

$$\eta_b = 1 - \frac{\bar{\varpi}}{\frac{\rho}{2}(V_{w1} + V_{w2})(V_{w1} - V_{w2})}$$

$$\frac{V_{w1} + V_{w2}}{2} = V_{wm}$$

$$= V_m \sin \alpha_m$$

Also we get $L \simeq \rho \Gamma V_m$

$$\eta_b = 1 - \frac{\bar{\varpi}}{\rho V_m \sin \alpha_m} \frac{(\cos \alpha_m) S}{(V_{w1} - V_{w2}) S} \cdot \frac{1}{\cos \alpha_m}$$

$$= 1 - \frac{D}{\rho V_m \Gamma \sin \alpha_m \cos \alpha_m} = 1 - \frac{D}{L \sin \alpha_m \cos \alpha_m}$$

$$= 1 - \frac{2D}{L \sin 2\alpha_m} \qquad [$$

$$(\eta_b)_{\text{comp cascade}} = 1 - \frac{2C_D}{C_L \sin 2\alpha_m}$$

η-maximum,

if $\frac{d\eta_b}{d\alpha_m} = 0 \Rightarrow \cos 2\alpha_m = 0$

$\alpha_m = 45°$

Thevalueof α_m for which efficiency ismaximum, $\alpha_m = 45°$

The blade efficiency for a turbine cascade is defined as:

$$\eta_b = \frac{\text{Ideal static pressure drop } (\Delta p)_s \text{ to obtain a certain change in kinetic energy}}{\text{Actual static pressure drop to produce the same change in kinetic energy}}$$

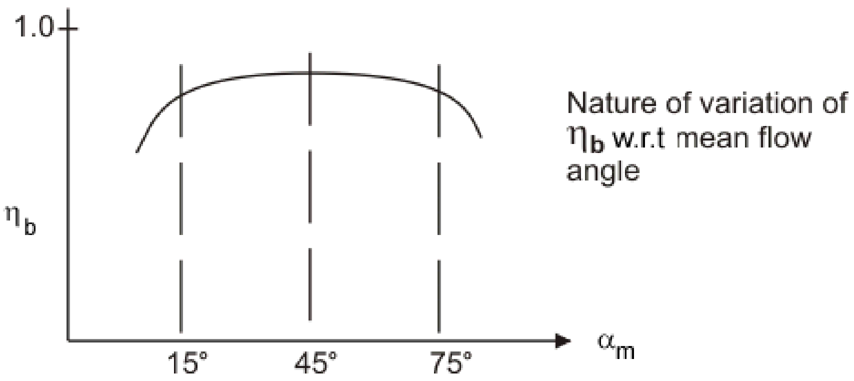
$$= \frac{\frac{\rho}{2}(V_2^2 - V_1^2)}{\frac{\rho}{2}(V_2^2 - V_1^2) + \bar{\varpi}} = \frac{1}{1 + \frac{\bar{\varpi}}{\frac{\rho}{2}(V_{w2} + V_{w1})(V_{w2} - V_{w1})}}$$

$$(\eta_b)_{\text{turbine}} = \frac{1}{1+\frac{2C_D}{C_L \sin 2\alpha_m}}$$

For very small ratio of C_D / C_L

$$(\eta_b)_{\text{turbine}} = \left(1+2\frac{C_D}{C_L} * \frac{1}{\sin 2\alpha_m}\right)^{-1}$$

$$(\eta_b)_{\text{turbine}} = 1-2\frac{C_D}{C_L \sin 2\alpha_m}$$ Which is same as the compressor cascade



Nature ofvariationof η_b wrt mean flowangle

In the above derivation for blade efficiency of both the compressor and turbine cascade, the lift is assumed as $\rho\Gamma V_m$, neglecting the effect of drag. With the corrected expression of lift, actual blade efficiencies are asfollows:

$$(\eta_b)_{\text{comp cascade}} = \frac{1-\frac{C_D}{C_L}\cot\alpha_m}{1+\frac{C_D}{C_L}\tan\alpha_m}$$

$$(\eta_b)_{\text{turb cascade}} = \frac{1-\frac{C_D}{C_L}\tan\alpha_m}{1+\frac{C_D}{C_L}\cot\alpha_m}$$

A gas turbine unit for power generation or a turbojet engine for production of thrust primarily consists of a compressor, combustion chamber and a turbine. The air as it passes through the compressor, experiences an increase in pressure. There after the air is fed to the combustion chamber leading to an increase in temperature. This high pressure and temperature gas is then passed through the turbine, where it is expanded and the required power is obtained.

Turbines, like compressors, can be classified into radial, axial and mixed flow machines. In the axial machine the fluid moves essentially in the axial direction through the rotor. In the radial type, the fluid motion is mostly radial. The mixed-flow machine is characterized by a combination of axial and radial motion of the fluid relative to the rotor. The choice of turbine type depends on the application, though it is not always clear that any one type is superior.

Comparing axial and radial turbines of the same overall diameter, we may say that the axial machine, just as in the case of compressors, is capable of handling considerably greater mass flow. On the other hand, for small mass flows the radial machine can be made more efficient than the axial one. The radial turbine is capable of a higher pressure ratio per stage than the axial one. However, multistage is very much easier to arrange with the axial turbine, so that large overall pressure ratios are not difficult to obtain with axial turbines. In this chapter, we will focus on the axial flow turbine.

Generally the efficiency of a well-designed turbine is higher than the efficiency of a compressor. Moreover, the design process is somewhat simpler. The principal reason for this fact is that the fluid undergoes a pressure drop in the turbine and a pressure rise in the compressor. The pressure drop in the turbine is sufficient to keep the boundary layer generally well behaved, and the boundary layer separation which often occurs in compressors because of an adverse pressure gradient, can be avoided in turbines. Offsetting this advantage is the much more critical stress problem, since turbine rotors must operate in very high temperature gas. Actual blade shape is often more dependent on stress and cooling considerations than on aerodynamic considerations, beyond the satisfaction of the velocity-triangle requirements.

Because of the generally falling pressure in turbine flow passages, much more turning in a giving blade row is possible without danger of flow separation than in an axial compressor blade row. This means much more work, and considerably higher pressure ratio, per stage.

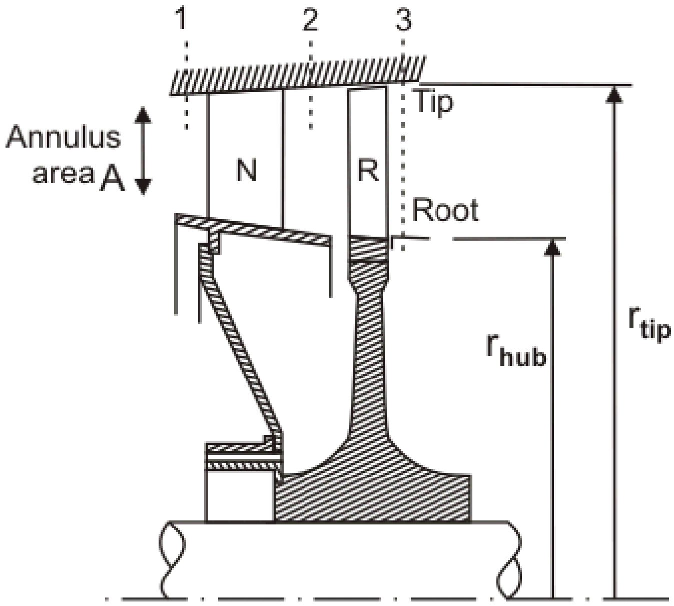
In recent years advances have been made in turbine blade cooling and in the metallurgy of turbine blade materials. This means that turbines are able to operate successfully at increasingly high inlet gas temperatures and that substantial improvements are being made in turbine engine thrust, weight, and fuel consumption.

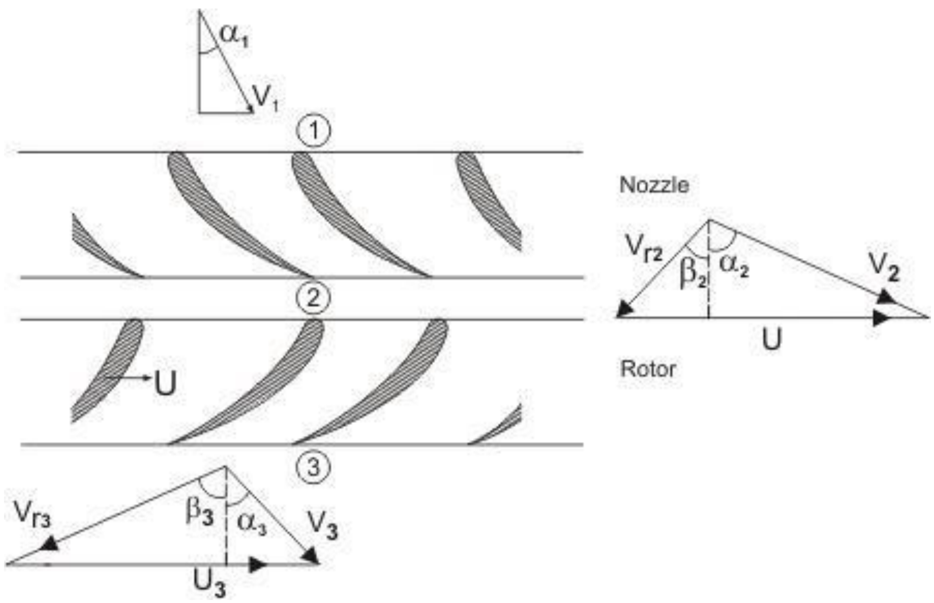
Two-dimensional theory of axial flow turbine

An axial turbine stage consists of a row of stationary blades, called nozzles or stators, followed by the rotor, as Figure 13.1 illustrates. Because of the large pressure drop per stage, the nozzle and rotor blades may be of increasing length, as shown, to accommodate therapidly expanding gases, while holding the axial velocity to something like a uniform value through the stage.

It should be noted that the hub-tip ratio for a high pressure gas turbine in quite high, that is, it is having blades of short lengths. Thus, the radial variation in velocity and pressure may be neglected and the performance of a turbine stage is calculated from the performance of the blading at the mean radial section, which is a two-dimensional "pitch-line design analysis ". A low-pressure turbine will typically have a much lower hub-tip ratio and a larger blade twist. A two dimensional design is not valid in thiscase.

In two dimensional approach the flow velocity will have two components, one axial and the other peripheral, denoted by subscripts 'f' and 'ω'respectively. The absolute velocity is denoted by V and the relative velocity with respect to the impeller by V_r. The flow conditions '1' indicates inlet to the nozzle or stator vane, '2' exit from the nozzle or inlet to the rotor and '3' exit from the rotor. Absolute angle is represented by α and relative angle by β as before.





Axial Turbine Stage

A section through the mean radius would appear as in Figure.13.1. One can see that the nozzles accelerate the flow imparting an increased tangential velocity component. The velocity diagram of the turbine differs from that of the compressor in that the change in tangential velocity in the rotor, ΔV_w ,is in the direction opposite to the blade speed U. The reaction to this change in the tangential momentum of the fluid is a torque on the rotor in the direction of motion. Hence the fluid does work on therotor.

Again applying the angular momentum relation-ship, we may show that the power output as,

$$P = \dot{m}(U_2 V_{w_2} - U_3 V_{w_3}) \qquad (13.1)$$

In an axial turbine,

$$U_2 \simeq U_3 = U \ \ (say)$$

The work output per unit mass flow rate is

$$W_T = U(V_{w_2} - V_{w_3})$$

Again,

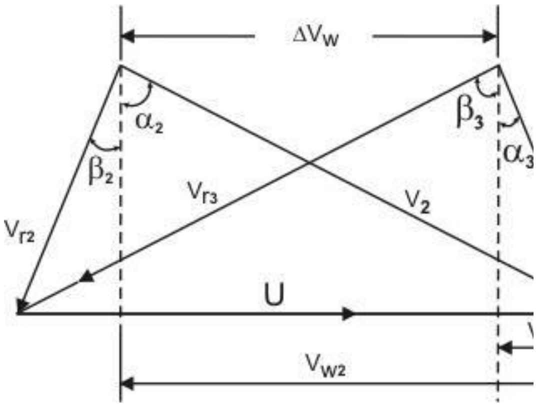
$$W_T = C_p (T_{o_1} - T_{o_3})$$

Defining

$$\Delta T_o = T_{o_1} - T_{o_3} = T_{o_2} - T_{o_3}$$

We find that the stage work ratio is

$$\frac{\Delta T_o}{T_{o_1}} = \frac{W_T}{C_p T_{o_1}} = \frac{U(V_{w_2} - V_{w_3})}{C_p T_{o_1}} \qquad (13.2)$$



Combined velocity diagram

The velocity diagram gives the following relation:

$$\frac{U}{V_f} = \tan\alpha_2 - \tan\beta_2$$

$$= \tan\alpha_3 - \tan\beta_3$$

Thus, $$W_T = U(V_{w_2} - V_{w_3})$$

$$= UV_f[\tan\alpha_2 - \tan\alpha_3]$$

i.e, $$W_T = UV_f[\tan\alpha_2 - \tan\alpha_3] \qquad (13.3)$$

The Eq (13.3) gives the expression for W_T in terms of gas angles associated with the rotor blade.

Note that the "work-done factor" required in the case of the axial compressor is unnecessary here. This is because in an accelerating flow the effect of the growth of boundary layer along the annulus passage in much less than when there is a decelerating flow with an adverse pressure gradient.

Instead of temperature drop ratio [defined in Eq (13.2)], turbine designers generally refer to the work capacity of a turbine stage as,

$$\Psi = \frac{c_p \Delta T_o}{U^2} = \frac{V_{w_2} - V_{w_3}}{V}$$

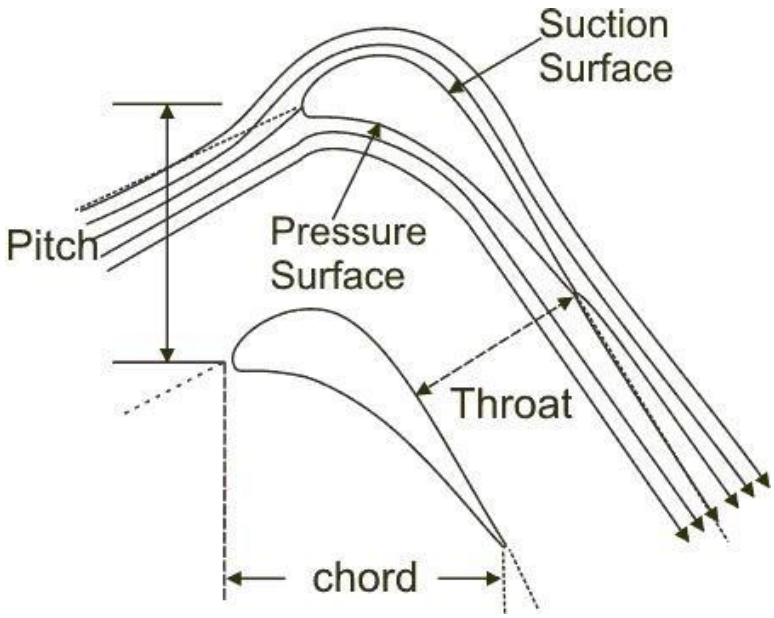
$$= \frac{V_f}{U}[\tan\beta_2 - \tan\beta_3] \qquad (13.4)$$

Ψ isa dimensionless parameter, which is called the "blade loading capacity" or "temperature drop coefficient". In gas turbine design, V_f is kept generally constant across a stage and the ratio V_f/U is called "the flow coefficient". ϕ

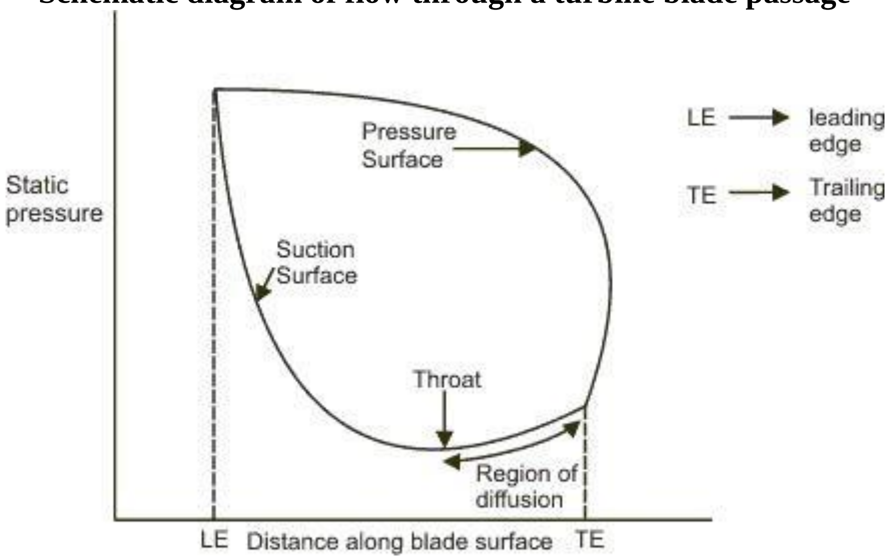
Thus, Eq (13.4) can be written as,

$$\psi = \phi[\tan\beta_2 - \tan\beta_3] \qquad (13.5)$$

As the boundary layer over the blade surface is not very sensitive in the case of a turbine, the turbine designer has considerably more freedom to distribute the total stage pressure drop between the rotor and the stator. However, locally on the suction surface of the blade there could be a zone of an adverse pressure gradient depending on the turning and on the pitch of the blades. Thus, the boundary layer could grow rapidly or even separate in such a region affecting adversely the turbine efficiency. Figure 13.3 illustrates the schematic of flow within the blade passage and the pressure distribution over the section surface depicting a zone of diffusion. Different design groups have their own rules, learned from experience of blade testing, for the amount of diffusion which is permissible particularly for highly loaded blades.



Schematic diagram of flow through a turbine blade passage



Pressure distribution around a turbine blade

Degree of reaction

Another useful dimensionless parameter is the "degree for reaction" or simply the "reaction" R. It may be defined for a turbine as the fraction of overall enthalpy drop (or pressure drop) occurring in the rotor

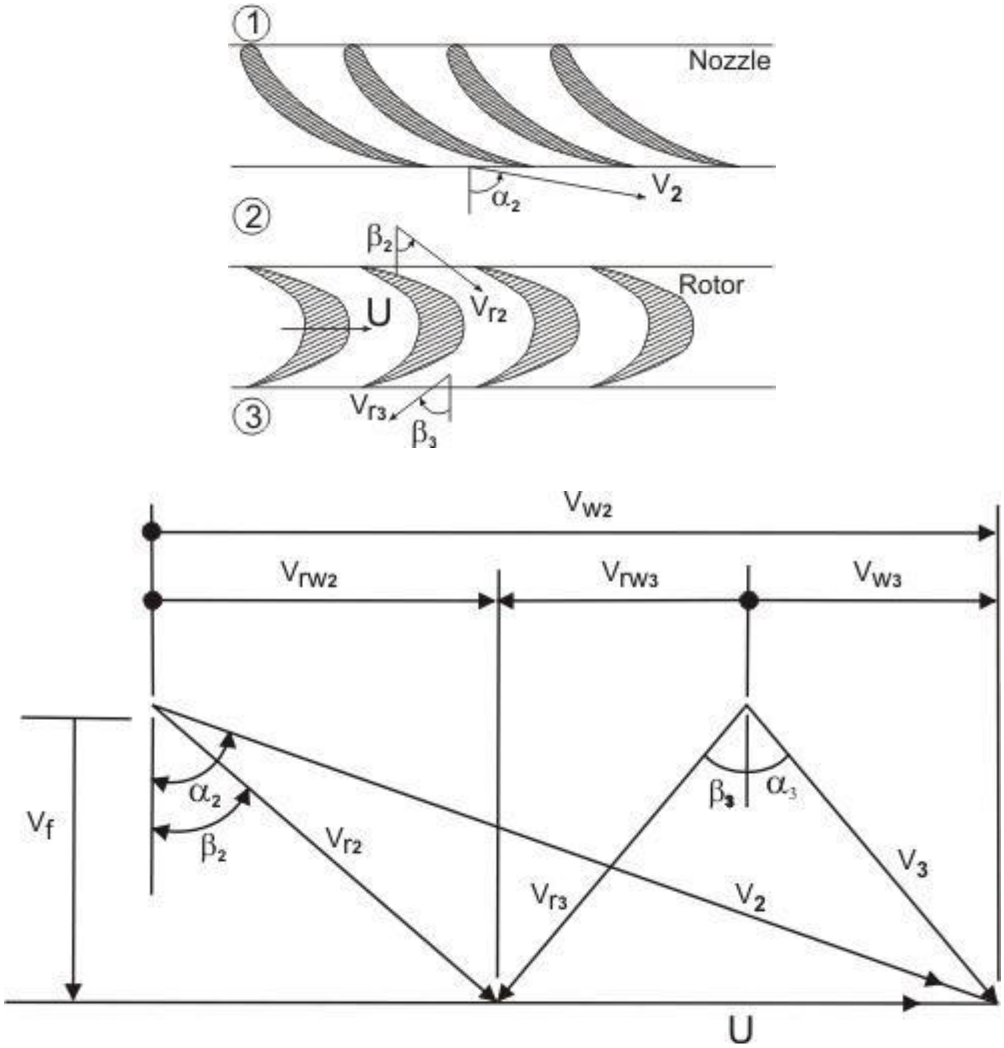
Thus, $$R = \frac{h_2 - h_3}{h_{o_1} - h_{o_3}} \qquad (14.1)$$

or, $$\frac{T_2 - T_3}{T_{o_1} - T_{o_3}}$$

Turbine stage in which the entire pressure drop occurs in the nozzle is called "impulse stages". Stages in which a portion of the pressure drop occurs in the nozzle and the rest in the rotor are called reaction stages. In a 50% reaction turbine, the enthalpy drop in the rotor would be half of the total for thestage.

An impulse turbine stage is shown in Fig14.1, along with the velocity diagram for the common case of constant axial velocity. Since no enthalpy change occurs within the rotor,the energy equation within the rotor requiresthat $|V_{r_2}| = |V_{r_3}|$. If the axial velocity is held constant, then this requirement is satisfiedby

$$\beta_3 = -\beta_2$$



Impulse turbine stage with constant axial velocity

From the velocity diagram, we can see that

$$V_{r_{w3}} = -V_{r_{w2}}$$

i.e.

$$V_{w_2} - V_{w_3} = 2V_{r_{w2}}$$

$$= 2(V_{w_2} - U)$$

$$= 2U\left(\frac{V_f}{U}\tan\alpha_2 - 1\right)$$

Then,

$$\psi = \frac{V_{w_2} - V_{w_3}}{U}$$

$$= 2(\phi\tan\alpha_2 - 1) \qquad (14.2)$$

The Eq (14.2) illustrates the effect of the nozzle outlet angle on the impulse turbine work output.

It is evident, then, that for large power output the nozzle angle should be as large as possible. Two difficulties are associated with very large α_2. For reasonable axial velocities (i.e., reasonable flow per unit frontal area), it is evident that large α_2 creates very large absolute and relative velocities throughout the stage. High losses are associated with such velocities, especially if the relative velocity V_{r_2} is supersonic. In practice, losses seem to beminimized forvaluesof α_2 around 70^o. In addition, one can see that forlarge α_2 $[\tan\alpha_2 > (2U/V_f)]$, The absolute exhaust velocity will have a swirl in the direction opposite to U. While we have not introduced the definition of turbine efficiency as yet, it is clear that, in a turbojet engine where large axial exhaust velocity is desired, the kinetic energy associated with the tangential motion of the exhaust gases is essentially a loss. Furthermore, application of the angular momentum equation over the entire engine indicates that exhaust swirl is associate with (undesirable) net torque acting on the aircraft. Thus the desire is for axial or near-axial absolute exhaust velocity (at least for the last stage if a multistage turbine is used). For the special case of constant V_f and axial exhaustvelocity $V_{w_3} = 0$ and $V_{w_2} = 2U$, the Eq.14.2becomes,

$$\psi = 2 \qquad \left[\because \tan\alpha_2 = \frac{V_w}{V_f} = \frac{2U}{V_f} = 2/\phi\right]$$

For a given power and rotor speed, and for a given peak temperature, Eq. (14.2) is sufficient to determine approximately the mean blade speed (and hence radius) of a single-stage impulse turbine having axial outlet velocity. If , as is usually the case, the blade speed is too high (for stress limitations), or if the mean diameter is too large relative to the other engine components, it is necessary to employ a multistage turbine in which each stage does part of thework.

STAGE EFFICENCY

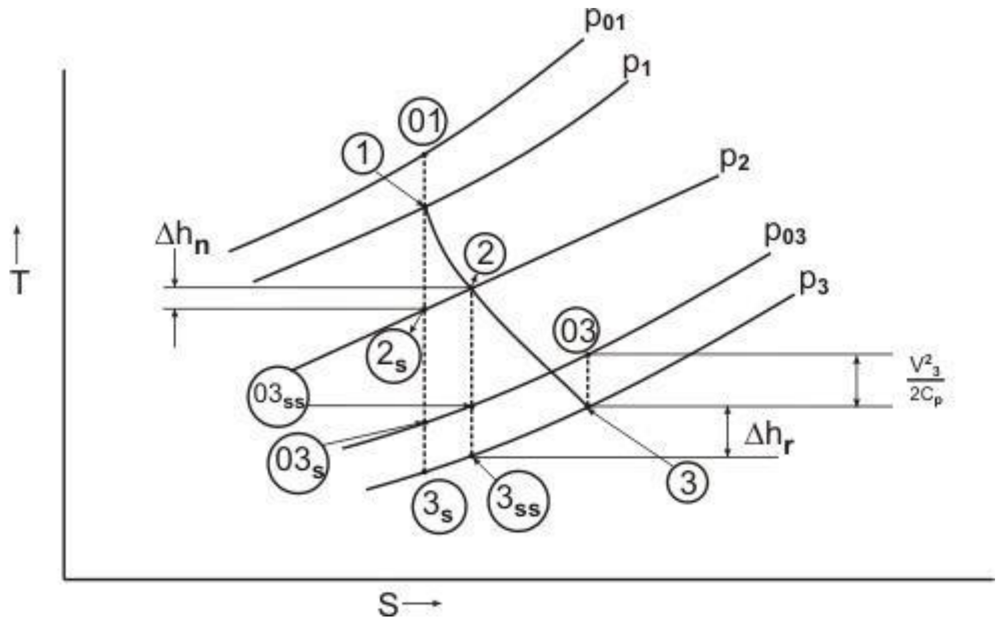
The aerodynamic losses in the turbine differ with the stage configuration, that is, the degree of reaction. Improved efficiency is associated with higher reaction, which tends to mean less work per stage and thus a large number of stages for a given overall pressureratio.

The understanding of aerodynamic losses is important to design, not only in the choice of blading type (impulse or reaction) but also in devising ways to control these losses, for example, methods to control the clearance between the tip of the turbine blade and the outer casing wall. The choices of blade shape, aspect ratio, spacing, Reynolds number, Mach number and flow incidence angle can all affect the losses and hence the efficiency of turbine stages.

Two definitions of efficiency are in common usage: the choice between them depends on the application for which the turbine is used. For many conventional applications, useful turbine output is in the form of shift power and the kinetic energy of the exhaust, $V_3^2/2$, is considered as a loss. In this case, ideal work would be $C_P(T_{01} - T_{3s})$ and a total to static turbine efficiency, η_{ts} based on the inlet and exit static conditions, isused.

Thus, $$\eta_{ts} = \frac{T_{o1} - T_{o3}}{T_{o1} - T_{3s}} \qquad (15.1)$$

The ideal (isentropic) to actual expansion process in turbines is illustrated in Fig 15.1.



T-S diagram: expansion in a turbine

Further,

$$\eta_{ts} = \frac{T_{o1} - T_{o3}}{T_{o1}\,[1-(P_3/P_{o1})^{(\gamma-1)/\gamma}]}$$

$$= \frac{1-(To_3 / T_{o1})}{1-(p_3 / p_{o1})^{(\gamma-1)/\gamma}} \qquad (15.2)$$

In some applications, particularly in turbojets, the exhaust kinetic energy is not considered a loss since the exhaust gases are intended to emerge at high velocity. The ideal work in this case is then $C_P(T_{01}-T_{03s})$ ratherthan $C_P(T_{01}-T_{3s})$. This requires a different definitionof efficiency, the total-to-totalturbineefficiency η_{tt}, definedby

$$\eta_{tt} = \frac{T_{01}-T_{03}}{T_{01}-T_{03s}} = \frac{1-(T_{03}/T_{01})}{1-(p_{03}/p_{01})^{(\gamma-1)/\gamma}} \qquad (15.3)$$

One can compare η_{tt} & η_{ts} by making theapproximation,

$$T_{03s}-T_{3s} \cong T_{03} - T_3 = V_3^2 / 2C_p,$$

and using Eqs. (15.2) and (15.3) it can be shown that

$$\eta_{tt} = \frac{\eta_{ts}}{1-V_3^2[2C_p(T_{01}-T_{3s})]}$$

Thus

$$\eta_{tt} > \eta_{ts}$$

The actual turbine work can be expressed as,

$$W_t = \eta_{tt} C_p T_{01}[1-(\frac{P_{03}}{P_{01}})^{(\gamma-1)/r}]. \qquad (15.4)$$

or,

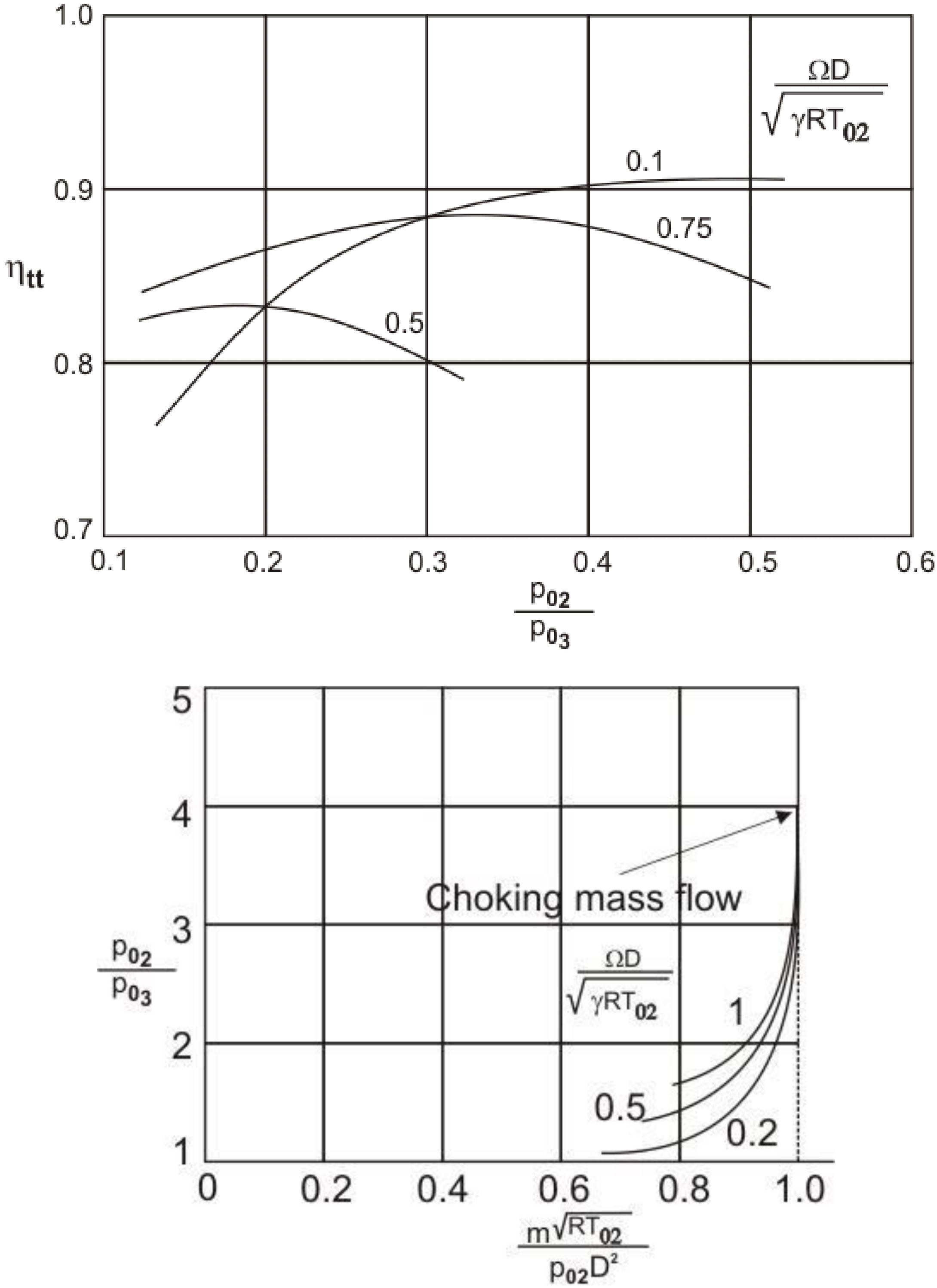
$$W_t = \eta_{ts} C_p T_{01}[1-(\frac{P_3}{P_{01}})^{(\gamma-1)/r}].$$

Turbine Performance

For a given design of turbine operating with a given fluid at sufficiently high Reynolds member, it can be shown from the dimensional analysis as,

$$\frac{P_{0_2}}{P_{0_3}} = f(\frac{\dot{m}\sqrt{RT_{0_2}}}{P_{0_2} D^2}, \frac{\Omega D}{\sqrt{\gamma R T_{0_2}}}),$$

Where, stagnation states 02 and 03 are at the turbine inlet and outlet, respectively. Figure (15.2) shows the overall performance of a particular single-stage turbine.



Typical characteristics of a turbine stage

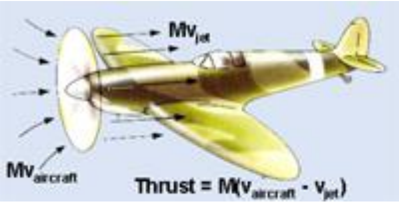
One can see that pressure ratios greater than those for compressor stages can be obtained with satisfactory efficiency.

The performance of turbines is limited principally by two factors: compressibility and stress. Compressibility limits the mass flow that can pass through a given turbine and, as we will see, stress limits the wheel speed U. The work per stage, for example, depends on the square of the wheel speed. However, the performance of the engine depends very strongly onthe maximum temperature. Of course, as the maximum temperature increases, the allowable stress level diminishes; hence in the design of the engine there must be a compromise between maximum temperature and maximum rotor tip speedU.For given pressure ratio and adiabatic efficiency, the turbine work per unit mass is proportional to the inlet stagnation temperature. Since, in addition, the turbine work in a jet or turbo shaft engine is commonly two or three times the useful energy output of the engine, a 1% increase in turbine inlet temperature can produce a 2% or 3% increase in engine output. This considerable advantage has supplied the incentive for the adoption of fairly elaborate methods for cooling the turbine nozzle and rotorblades.

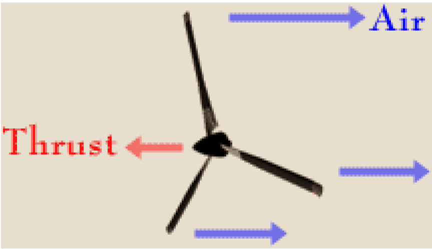
UNIT - III
INLETS, NOZZLES AND PROPERLLER THEROY

PROPERLLER

Thrust is the force that moves the aircraft through the air. Thrust is generated by the propulsion system of the aircraft. There are different types of propulsion systems develop thrust in different ways, although it usually generated through some application of Newton's Third Law. Propeller is one of the propulsion systems. The purpose of the propeller is to move the aircraft through the air. The propeller consists of two or more blades connected together by a hub. The hub serves to attach the blades to the engine shaft. .

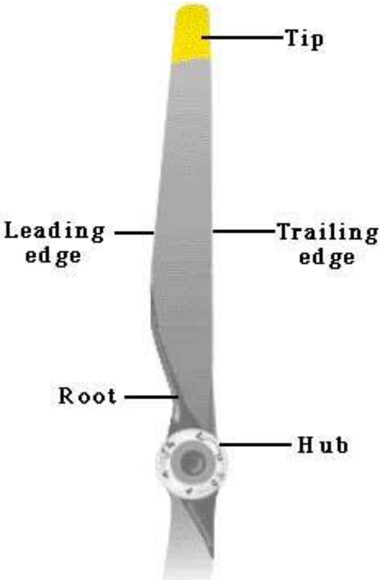


The blades are made in the shape of an airfoil like wing of an aircraft. When the engine rotates the propeller blades, the blades produce lift. This lift is called **thrust** and moves the aircraft forward. Most aircraft have propellers that pull the aircraft through the air. These are called **tractor** propellers. Some aircraft have propellers that **push** the aircraft. These are called **pusher** propellers.

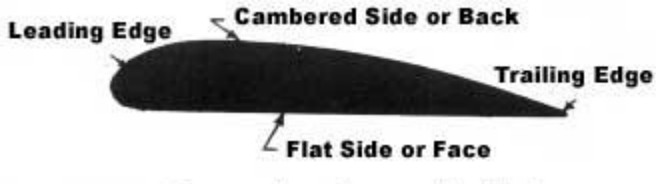


Description

Leading Edge of the airfoil is the cutting edge that slices into the air. As the leading edge cuts the air, air flows over the blade face and the camber side.

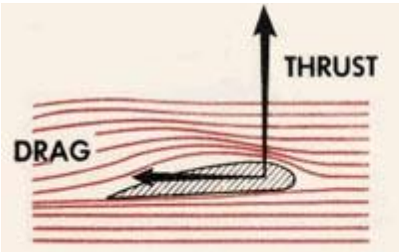


Blade Face is the surface of the propeller blade that corresponds to the lower surface of an airfoil or flat side, we called Blade Face.

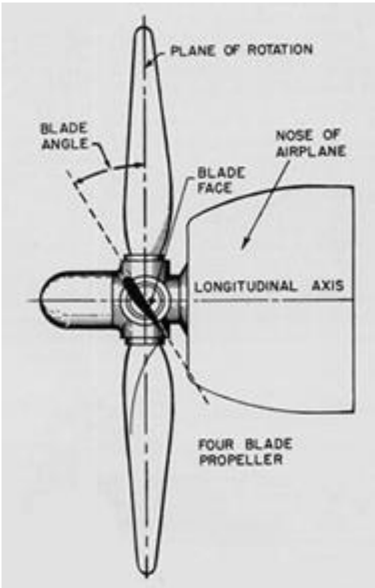


Cross section of a propeller blade.

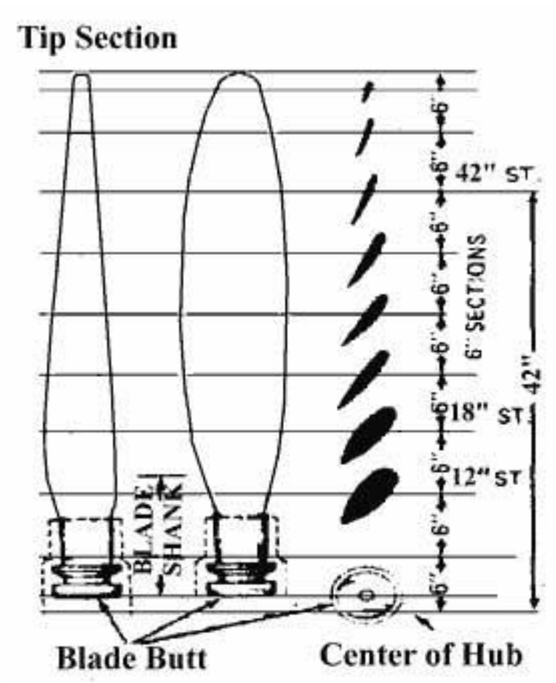
Blade Back / Thrust Face is the curved surface of the airfoil.



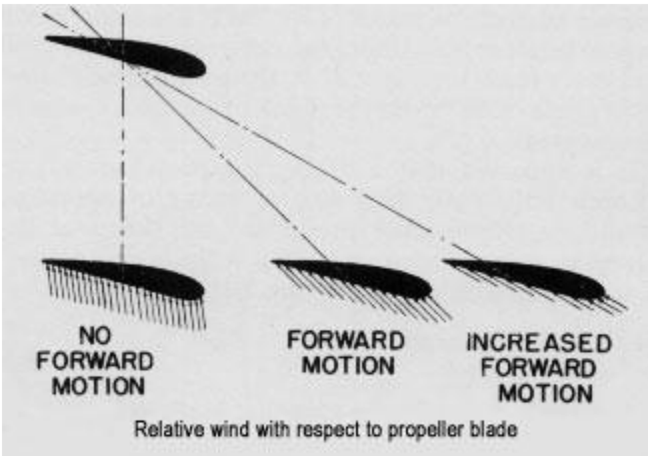
Blade Shank (Root) is the section of the blade nearest the hub.
Blade Tip is the outer end of the blade farthest from the hub.
Plane of Rotation is an imaginary plane perpendicular to the shaft. It is the plane that contains the circle in which the blades rotate.



Blade Angle is formed between the face of an element and the plane of rotation. The blade angle throughout the length of the blade is not the same. The reason for placing the blade element sections at different angles is because the various sections of the blade travel at different speed. Each element must be designed as part of the blade to operate at its own best angle of attack to create thrust when revolving at its best design speed

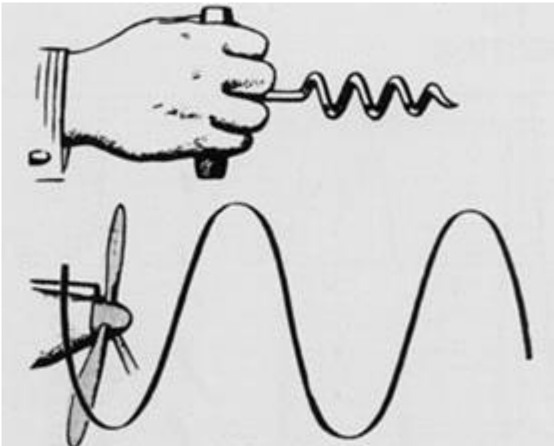


Blade Element is the airfoil sections joined side by side to form the blade airfoil. These elements are placed at different angles in rotation of the plane of rotation.
The reason for placing the blade element sections at different angles is because the various sections of the blade travel at different speeds. The inner part of the blade section travels slower than the outer part near the tip of the blade. If all the elements along a blade are at the same blade angle, the relative wind will not strike the elements at the same angle of attack.
Thrust is produced by the propeller attached to the engine driveshaft. While the propeller is rotating in flight, each section of the blade has a motion that combines the forward motion of the aircraft with circular movement of the propeller. The slower the speed, the steeper the angle of attack must be to generate lift. Therefore, the shape of the propeller's airfoil (cross section) must change from the center to the tips. The changing shape of the airfoil (cross section) across the blade results in the twisting shape of the propeller.



Relative wind with respect to propeller blade

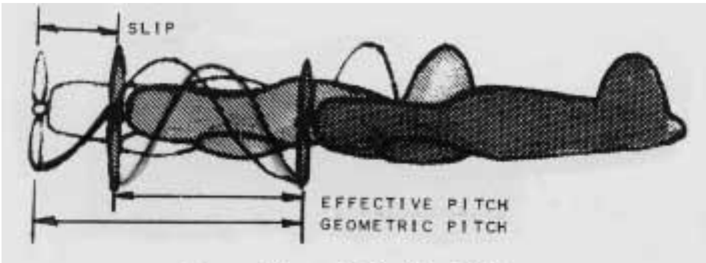
Relative Wind is the air that strikes and passes over the airfoil as the airfoil is driven through the air.
Angle of Attack is the angle between the chord of the element and the relative wind. The best efficiency of the propeller is obtained at an angle of attack around 2 to 4 degrees.
Blade Path is the path of the direction of the blade element moves.



Propeller Path

Pitch refers to the distance a spiral threaded object moves forward in one revolution. As a wood screw moves forward when turned in wood, same with the propeller move forward when turn in the air.

Geometric Pitch is the theoretical distance a propeller would advance in one revolution.



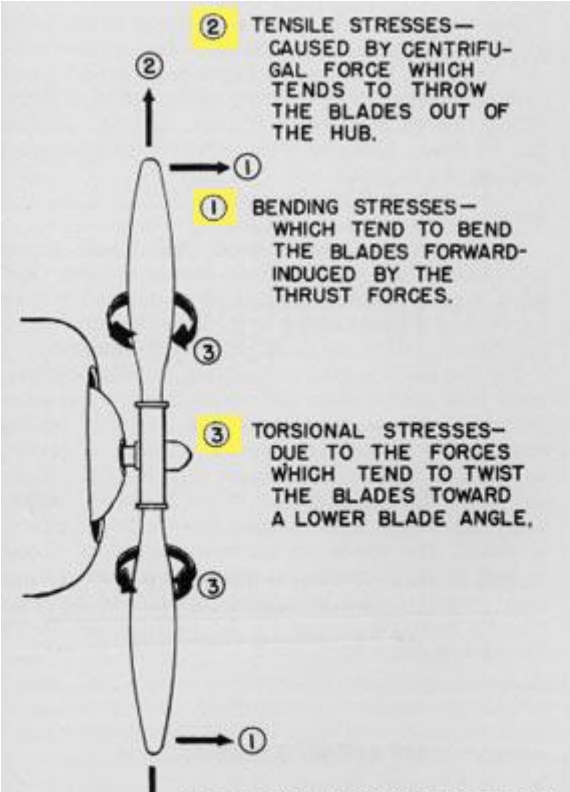
Geometric and Effective Pitch

Effective Pitch is the actual distance a propeller advances in one revolution in the air. The effective pitch is always shorter than geometric pitch due to the air is a fluid and always **slips.**

Forces and stresses acting on a propeller in flight

The **forces** acting on a propeller in flight are:

1. **Thrust** is the air force on the propeller which is parallel to the direction of advance and induce bending stress in the propeller.
2. **Centrifugal force** is caused by rotation of the propeller and tends to throw the blade out from the center.
3. **Torsion or Twisting forces** in the blade itself, caused by the resultant of air forces which tend to twist the blades toward a lower blade angle.



Forces acting on propeller during operation

The stress acting on a propeller in flight is:

1. **Bending stresses** are induced by the trust forces. These stresses tend to bend the blade forward as the airplane is moved through the air by the propeller.
2. **Tensile stresses** are caused by centrifugal force.
3. **Torsion stresses** are produced in rotating propeller blades by two twisting moments. One of these stresses is caused by the air reaction on the blades and is called the aerodynamic twisting moment. The stress is caused by centrifugal force and is called the centrifugal twisting moment.

Type of propellers

In designing propellers, the maximum performance of the airplane for all condition of operation from takeoff, climb, cruising, and high speed. The propellers may be classified under eight general types as follows:

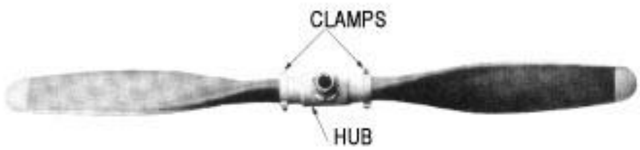
1. **Fixed pitch:** The propeller is made in one piece. Only one pitch setting is possible and is usually two blades propeller and is often made of wood or metal.

Wooden Propellers: Wooden propellers were used almost exclusively on personal and business aircraft prior to World War II .A wood propeller is not cut from a solid block but is built up of a number of separate layers of carefully selected .any types of wood have been used in making propellers, but the most satisfactory are yellow birch, sugar maple, black cherry, and black walnut. The use of lamination of wood will reduce the tendency for propeller to warp. For standard one-piece wood propellers, from five to nine separate wood laminations about 3/4 in. thick is used.

Fixed-pitch one-piece wood propeller.

Metal Propellers: During 1940, solid steel propellers were made for military use. Modern propellers are fabricated from high-strength, heat-treated, aluminum alloy by forging a single bar of aluminum alloy to the required shape. Metal propellers are now extensively used in the construction of propellers for all type of aircraft. The general appearance of the metal propeller is similar to the wood propeller, except that the sections are generally thinner.

2. **Ground adjustable pitch:** The pitch setting can be adjusted only with tools on the ground before the engine is running. This type of propellers usually has a split hub. The blade angle is specified by the aircraft specifications. The adjustable - pitch feature permits compensation for the location of the flying field at various altitudes and also for variations in the characteristics of airplanes using the same engine. Setting the blade angles by loosened the clamps and the blade is rotated to the desired angle and then tightens the clamps.



A ground-adjustable propeller.

3. **Two-position:** A propeller which can have its pitch changed from one position to one other angle by the pilot while in flight.
4. **Controllable pitch:** The pilot can change the pitch of the propeller in flight or while operating the engine by mean of a pitch changing mechanism that may be operated by hydraulically.
5. **Constant speed:** The constant speed propeller utilizes a hydraulically or electrically operated pitch changing mechanism which is controlled by governor. The setting of the governor is adjusted by the pilot with the rpm lever in the cockpit. During operation, the constant speed propellers will automatically changes its blade angle to maintain a constant engine speed. If engine power is increase, the blade angle is increased to make the propeller absorb the additional power while the rpm remain constant. At the other position, if the engine power is decreased, the blade angle will decrease to make the blades take less bite of air to keep engine rpm remain constant. The pilot selects the engine speed required for any

particular type of operation.

6. **Full Feathering:** A constant speed propeller which has the ability to turn edge to the wind and thereby eliminate drag and wind milling in the event of engine failure. The term Feathering refers to the operation of rotating the blades of the propeller to the wind position for the purpose of stopping the rotation of the propeller to reduce drag. Therefore, a Feathered blade is in an approximate in-line-of-flight position, streamlined with the line of flight (turned the blades to a very high pitch). Feathering is necessary when the engine fails or when it is desirable to shutoff an engine in flight.

7. **Reversing:** A constant speed propeller which has the ability to assume a negative blade angle and produce a reversing thrust. When propellers are reversed, their blades are rotated below their positive angle, that is, through flat pitch, until a negative blade angle is obtained in order to produce thrust acting in the opposite direction to the forward thrust. Reverse propeller thrust is used where a large aircraft is landed, in reducing the length of landing run.

8. **Beta Control:** A propeller which allows the manual repositioning of the propeller blade angle beyond the normal low pitch stop. Used most often in taxiing, where thrust is manually controlled by adjusting blade angle with the power lever.

Control and Operation

Propeller Control

Basic requirement: For flight operation, an engine is demanded to deliver power within a relatively narrow band of operating rotation speeds. During flight, the speed-sensitive governor of the propeller automatically controls the blade angle as required to maintain a constant r.p.m. of the engine. Three factors tend to vary the r.p.m. of the engine during operation. These factors are **power, airspeed, and air density.** If the r.p.m. is to maintain constant, the blade angle must vary directly with power, directly with airspeed, and inversely with air density. The speed-sensitive governor provides the means by which the propeller can adjust itself automatically to varying power and flight conditions while converting the power to thrust.

Fundamental Forces: Three fundamental forces are used to control blade angle. These forces are

1. Centrifugal twisting moment, centrifugal force acting on a rotating blade which tends at all times to move the blade into low pitch.
2. Oil at engine pressure on the outboard piston side, which supplements the centrifugal twisting moment toward low pitch.
3. Propeller Governor oil on the inboard piston side, which balances the first two forces and move the blades toward high pitch Counterweight assembly (this is only for counterweight

propeller) which attached to the blades , the centrifugal forces of the counterweight will move the blades to highpitch setting.

Constant Speed, Counterweight Propellers

The Counterweight type propeller may be used to operate either as a controllable or constant speed propeller. The hydraulic counterweight propeller consists of a hub assembly, blade assembly, cylinder assembly, and counterweight assembly.

The counterweight assembly on the propeller is attached to the blades and moves with them. The centrifugal forces obtained from rotating counterweights move the blades to high angle setting. The centrifugal force of the counterweight assembly is depended on the rotational speed of the propellers r.p.m. The propeller blades have a definite range of angular motion by an adjusting for high and low angle on the counterweight brackets.

Controllable : the operator will select either low blade angle or high blade angle by two-way valve which permits engine oil to flow into or drain from the propeller.

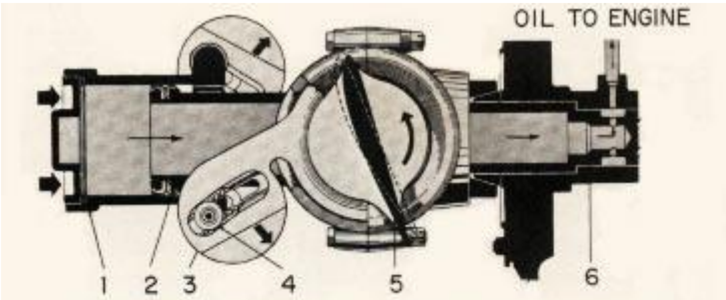
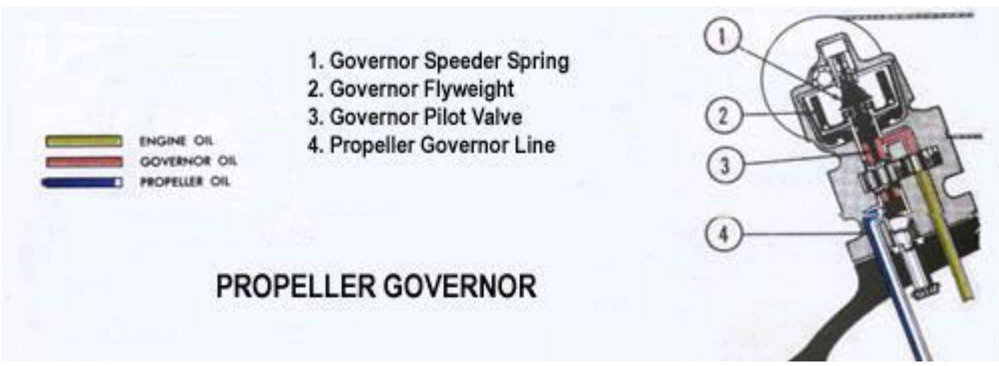


Diagram of Controllable Two Position Pitch Mechanism

(1) Propeller cylinder, (2) propeller piston, (3) propeller counterweight and bracket, (4) propeller counterweight shaft and bearing, (5) propeller blade, (6) engine propeller shaft.

Constant Speed: If an engine driven **governor** is used, the propeller will operate as a constant speed. The propeller and engine speed will be maintained constant at any r.p.m. setting within the operating range of the propeller.

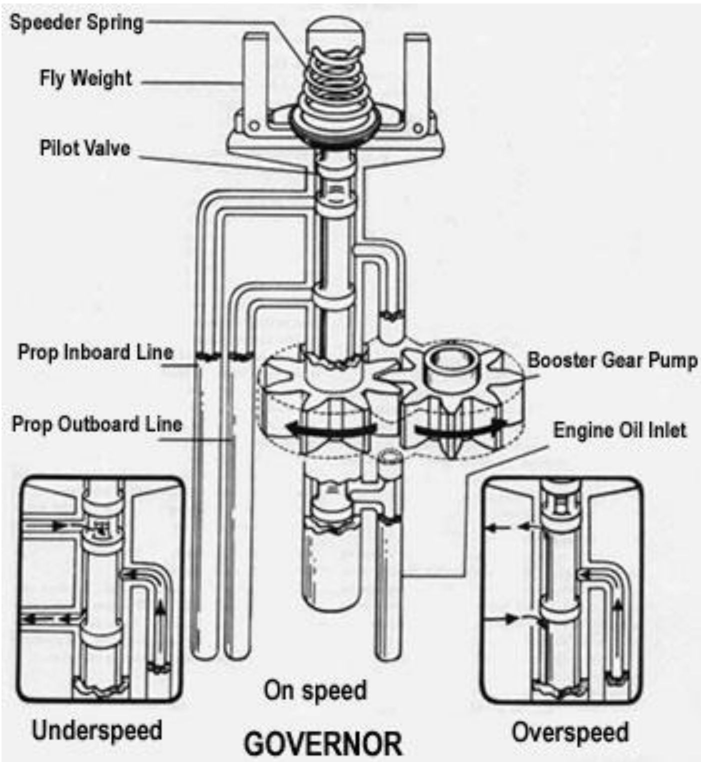


Governor Operation (Constant speed with counterweight) the Governor supplies and controls the flow of oil to and from the propeller. The engine driven governor receives oil from the engine lubricating system and boost its pressure to that required to operate the pitch-changing mechanism. It consists essentially of:

1. A gear pump to increase the pressure of the engine oil to the pressure required for propeller operation.
2. A relief valve system which regulates the operating pressure in the governor.

3. A pilot valve actuated by flyweights which control the flow of oil through the governor.
4. The speeder spring provides a mean by which the initial load on the pilot valve can be changed through the rack and pulley arrangement which controlled by pilot.
5. The governor maintains the required balance between all three control forces by metering to, or drain from, the inboard side of the propeller piston to maintain the propeller blade angle for constant speed operation.

The governor operates by means of flyweights which control the position of a pilot valve. When the propeller r.p.m. is below that for which the governor is set through the speeder spring by pilot, the governor flyweight move inward due to less centrifugal force act on flyweight than compression of speeder spring. If the propeller r.p.m. is higher than setting, the flyweight will move outward due to flyweight has more centrifugal force than compression of speeder spring. During the flyweight moving inward or outward, the pilot valve will move and directs engine oil pressure to the propeller cylinder through the engine propeller shaft.



Principles of Operation (Constant Speed with Counterweight Propellers)

The changes in the blades angle of a typical constant speed with counterweight propellers are accomplished by the action of two forces, one is hydraulic and the other is mechanical.

1. The cylinder is moved by oil flowing into it and opposed by centrifugal force of counterweight. This action moves the counterweight and the blades to rotate toward the low angle position.

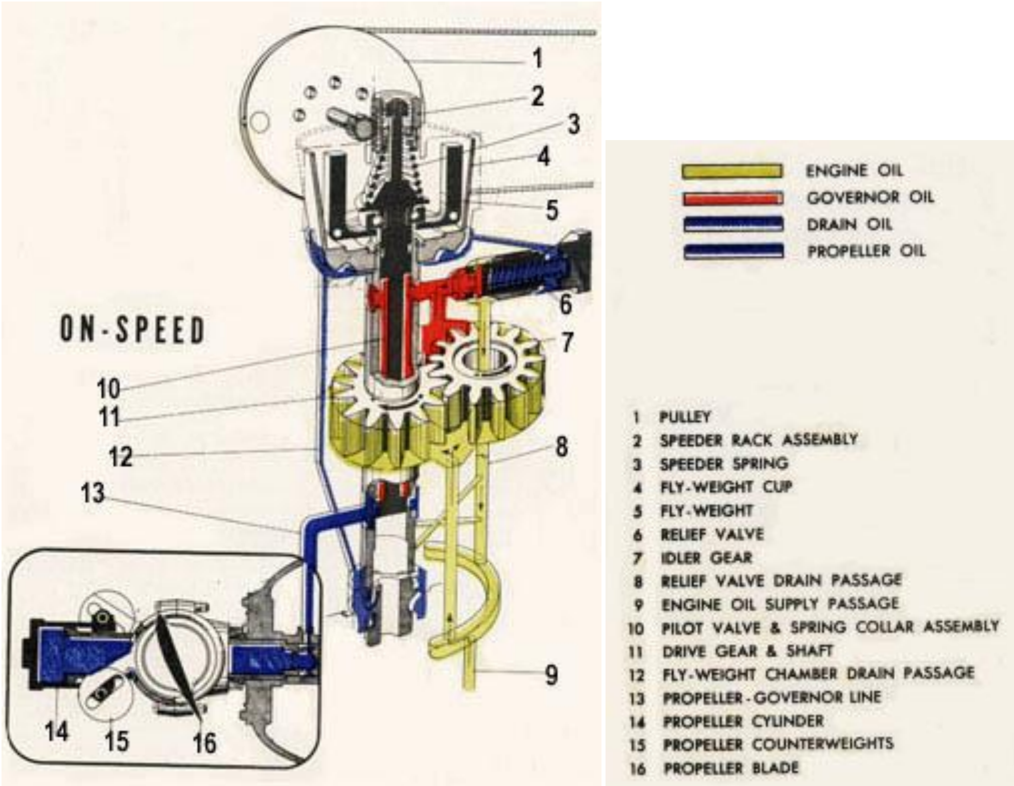
2. When the oil allowed draining from the cylinder, the centrifugal force of counterweights take effect and the blades are turned toward the high angle position.

3. The constant speed control of the propeller is an engine driven governor of the flyweight type.

Governor Operation Condition

On-Speed Condition

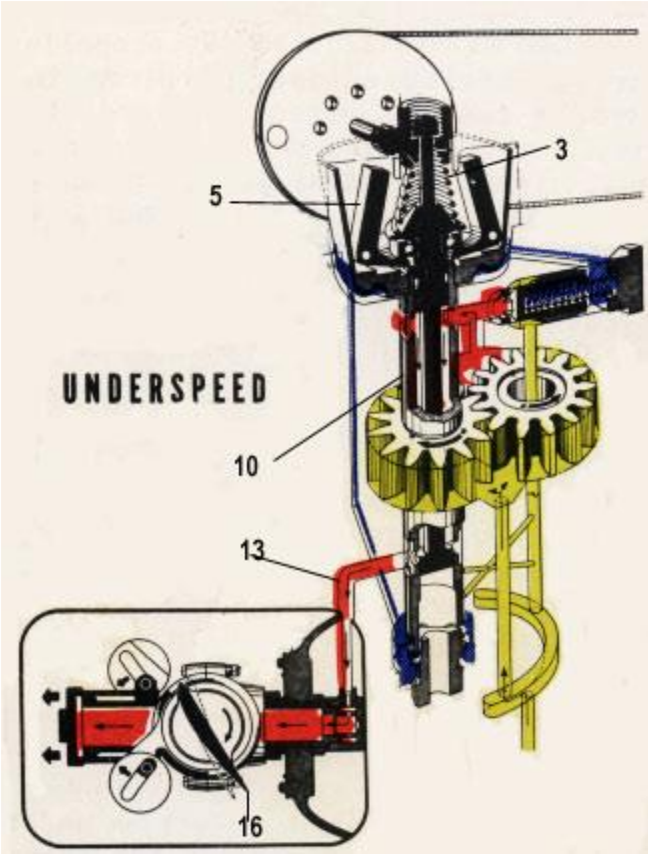
The on-speed condition exists when the propeller operation speed are constant. In this condition, the force of the flyweight (5) at the governor just balances the speeder spring (3) force on the pilot valve (10) and shutoff completely the line (13) connecting to the propeller , thus preventing the flow of oil to or from the propeller.



The pressure oil from the pump is relieved through the relief valve (6). Because the propeller counterweight (15) force toward high pitch is balanced by the oil force from cylinder (14) is prevented from moving, and the propeller does not change pitch

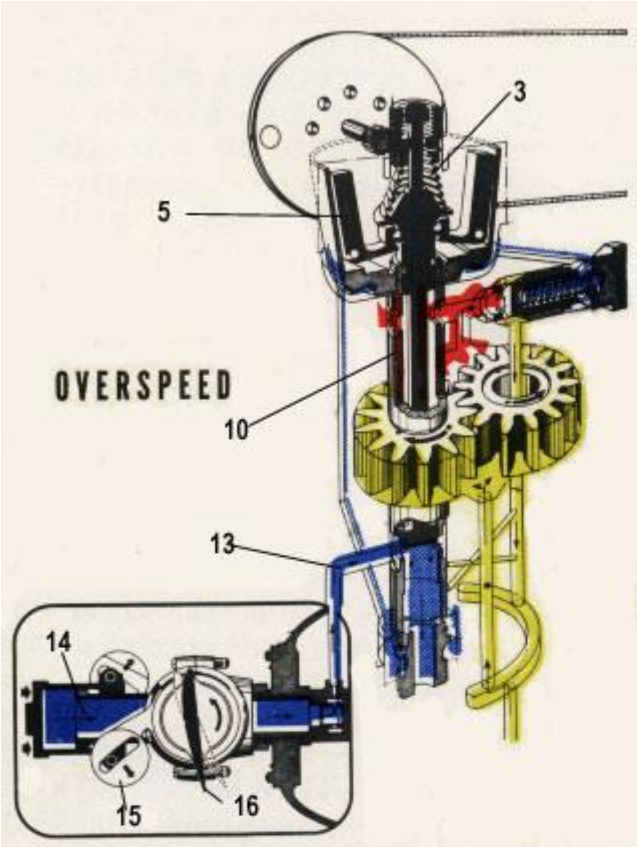
Under-Speed Condition

The under-speed condition is the result of change in engine r.p.m. or propeller r.p.m. Which the r.p.m. is tend to lower than setting or governor control movement toward a high r.p.m. Since the force of the flyweight (5) is less than the speeder spring (3) force, the pilot valve (10) is forced down. Oil from the booster pumps flows through the line (13) to the propeller. This forces the cylinder (14) move outward, and the blades (16) turn to lower pitch, less power is required to turn the propeller which in turn increases the engine r.p.m. As the speed is increased, the flyweight force is increased also and becomes equal to the speeder spring force. The pilot valve is move up, and the governor resumes its on-speed condition which keeps the engine r.p.m. constant.



Over-Speed Condition

The over-speed condition which occurs when the aircraft altitude change or engine power is increased or engine r.p.m. is tends to increase and the governor control is moved towards a lower r.p.m. In this condition, the force of the flyweight (5) overcomes the speeder spring (3) force and raise the pilot valve (10) open the propeller line (13) to drain the oil from the cylinder (14). The counterweight (15) force in the propeller to turn the blades towards a higher pitch. With a higher pitch, more power is required to turn the propeller which in turn slows down the engine r.p.m. As the speed is reduced, the flyweight force is reduced also and becomes equal to the speeder spring force. The pilot valve is lowered, and the governor resumes its on-speed condition which keeps the engine r.p.m. constant.



Flight Operation

This is just only guide line for understanding. The engine or aircraft manufacturers' operating manual should be consulted for each particular aircraft.

Takeoff: Placing the governor control in the full forward position. This position is setting the propeller blades to low pitch angle Engine r.p.m. will increase until it reaches the takeoff r.p.m. for which the governor has been set. From this setting, the r.p.m. will be held constant by the governor, which means that full power is available during takeoff and climb.

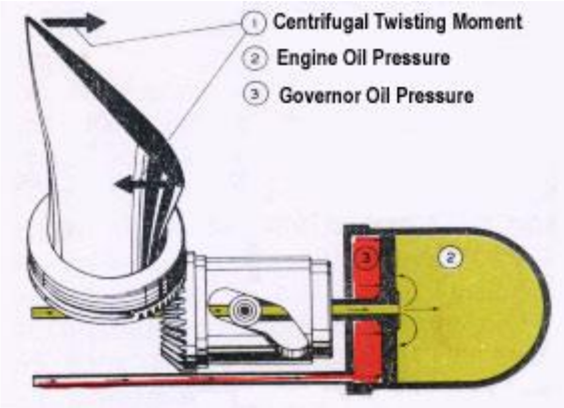
Cruising: Once the cruising r.p.m. has been set, it will be held constant by the governor. All changes in attitude of the aircraft, altitude, and the engine power can be made without affecting the r.p.m. as long as the blades do not contact the pitch limit stop.

Power Descent: As the airspeed increase during descent, the governor will move the propeller blades to a higher pitch in order to hold the r.p.m. at the desired value.

Approach and Landing: Set the governor to its maximum cruising r.p.m. position during approach. During landing, the governor control should be set in the high r.p.m. position and this move the blades to full low pitch angle.

Hydromantic Propellers

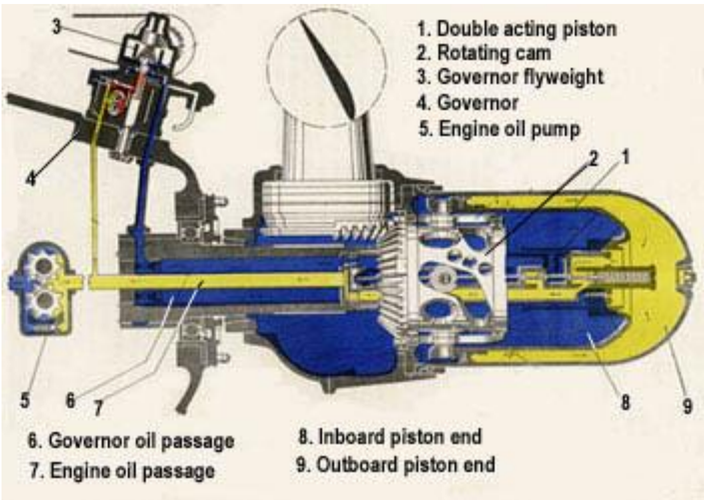
Basic Operation Principles:
The pitch changing mechanism of hydromantic propeller is a mechanical-hydraulic system in which hydraulic forces acting upon a piston are transformed into mechanical forces acting upon the blades.



Piston movement causes rotation of cam which incorporates a bevel gear (Hamilton Standard Propeller). The oil forces which act upon the piston are controlled by the governor

Single Acting Propeller: The governor directs its pump output against the inboard side of piston only; A single acting propeller uses a single acting governor. This type of propeller makes use of three forces during constant speed operation, the blades centrifugal twisting moment and this force tends at all times to move the blades toward low pitch, oil at engine pressure applied against the outboard side of the propeller piston and this force to supplement the centrifugal twisting moment toward the low pitch during constant speed operation., and oil from governor pressure applied against the inboard side of the piston. The oil pressure from governor was boosted from the engine oil supply by governor pump and the force is controlled by metering the high pressure oil to or draining it from the inboard side of the propeller piston which balances centrifugal twisting moment and oil at the engine pressure.

Double Acting Propeller: The governor directs its output either side of the piston as the operating condition required. Double acting propeller uses double acting governor. This type of propeller, the governor pump output oil is directed by the governor to either side of the propeller piston.



Principle Operation of Double Acting:

Over speedsCondition: When the engine speed increases above the r.p.m. for which the governor is set. Oil supply is boosted in pressure by thru engine driven propeller governor, is directed against the inboard side of the propeller piston. The piston and the attached rollers move outboard. As the piston moves outboard, cam and rollers move the propeller blades toward a higher angle, which in turn, decreases the engine r.p.m.

Under speedCondition: When the engine speed drops below the r.p.m. for which the governor is set. Force at flyweight is decrease and permit speeder spring to lower pilot valve, thereby open the oil passage allow the oil from inboard side of piston to drain through the governor. As the oil from inboard side is drained, engine oil from engine flows through the propeller shaft into the outboard piston end. With the aid of blade centrifugal twisting moment, The engine oil from outboard moves the piston inboard. The piston motion is transmitted through the cam and rollers. Thus, the blades move to lower angle

The Feathering System

Feathering: For some basic model consists of a feathering pump, reservoir, a feathering time-delay switch, and a propeller feathering light. The propeller is feathered by moving the control in the cockpit against the low speed stop. This causes the pilot valve lift rod in the governor to hold the pilot valve in the decrease r.p.m. position regardless of the action of the governor flyweights. This causes the propeller blades to rotate through high pitch to the feathering position.

Some model is initiated by depressing the feathering button. This action, auxiliary pump, feather solenoid, which positions the feathering valve to transfer oil to feathering the propeller. When the propeller has been fully feathered, oil pressure will buildup and operate a pressure cutout switch which will cause the auxiliary pump stop. Feathering may be also be accomplished by pulling the engine emergency shutdown handle or switch to the shutdown position.

Un feathering: Some model is accomplished by holding the feathering button switch in the out position for about 2 second. This creates an artificial under speed condition at the governor and causes high-pressure oil from the feathering pump to be directed to the rear of the propeller piston. As soon as the piston has moved inward a short distance, the blades will have sufficient angle to start rotation of the engine. When this occurs, the un-feathering switch can be released and the governor will resume control of the propeller.

INLETS

Role of the air intakes

Any vehicle with air-breathing propulsion needs at least one air intake to feed its engine so that it can move. The role of the air intake is to capture the airflow which the propulsion (engines) and conditioning (radiators) systems need. They must do this in such a way as to yield the best possible propulsive balance, which is expressed in two objectives:

- provide maximumthrust,

- Induce minimumdrag.

Maximum thrust is obtained by designing the air intake to transform kinetic energy (i.e., the velocity of the flow as it arrives in front of the air intake) into potential energy (the pressure after the diffuser, at the engine input) with the best possible efficiency. Efficiency is a parameter that is calculated by taking the ratio of the total pressure in front of the engine to that of the upstream flow. This thrust is maximum if the air intake captures just what the engine needs for each flight configuration and provides the engine input with a flow of good homogeneity (low distortion) to ensure correct engine operation, which is essential for turbojets. Minimum drag is obtained with air intakes that are dimensioned to just what the engine needs (critical regime) and the design Mach number (Shock-on-lip Mach number). Careful attention is paid to the side walls and cowls in light of the small variations in angle of attack and yaw angle about the flight configurations.

Engine Inlet Ducts

The dynamic intake is the first component that meets the flow in its evolution through the engine. It is positioned to provide the minimum external resistance. The task of the air intake is to channel the flow at low velocity through the compressor (or to the combustor in the case of the ramjet) without causing the detachment of the boundary layer (because by the slowdown of the flow, the static pressure increases, and the flow is then submitted an adverse pressure gradient). The air intake must be designed to provide the engine the required flow rate and also so that the output of the dynamic intake flow entering the compressor is uniform,stable,andwith–goodquality‖.Sothegoalsofthe Inletare:

– slow down the flow (up to M = 0.4 to0.5);

– increase thepressure;

– Uniform flow at the upstream of thecompressor;

– Minimal loss of totalpressure;

– Minimum aerodynamicdisorder;

– Minimum weight (i.e.minimumlength).

The performance should not be prejudice in the presence of an incidence angle or yaw. It is useful to observe that the requirement that the flow is uniform before the compressor could bemore important than to have small total pressure loss. The inlet is essentially a duct where the air flows in stationary conditions. itis designed according to rules of gas dynamics; since such laws have different implications depending on how the flow enters in the duct, if in supersonic or subsonic conditions, the main classification of distinguishesbetween:

1. Subsonicinlet;
2. Supersonicinlet

SUBSONICINTAKES

Subsonic intakes are found in the turbojet or turbofan engines powering most of the present civil transports. The surface of the inlet is a continuous smooth curve where the very front (most upstream portion) is called the inlet lip. A subsonic aircraft has an inlet with a relatively thick lip. Concerning turboprop engines, the intakes are much complicated by the propeller and gearbox at the inlet to the engine. Subsonic inlets have fixed geometry, although inlets for some high bypass ratio turbofan engines are designed with blow-in-doors. These doors are spring-loaded parts installed in the perimeter of the inlet duct designed to deliver additional air to the aero engine during takeoff and climb conditions when the highest thrust is needed and the aircraft speed is low. The most common type of subsonic intake is thepivot intake. This type of intakes makes the fullest use of ram due to forward speed, and suffers the minimum loss of ram pressure with changes of aircraft altitude. However, as sonic speed is approached, the efficiency of this type of air intake begins to fall because of the formation of a shock wave at the intake lip. It consists of a simple forward entry hole with a cowl lip. The three major types of pitot intakes as shown in Fig 2.1 are asfollows

1. Poddedintakes

2. Integratedintake

3. Flushintakes.

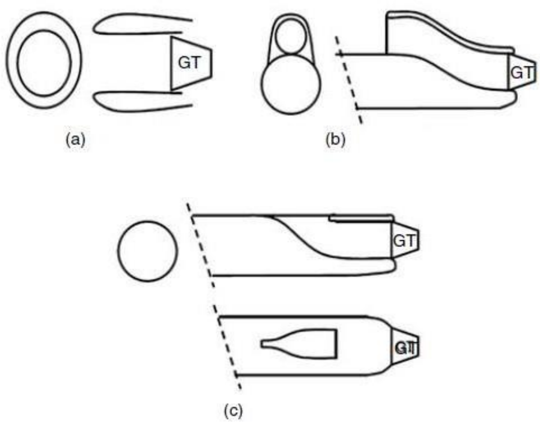


Fig. 2.2: Types of pitot intakes: (a) podded pitot, (b) integrated pitot, and (c) flush pitot.

Podded intake is common in transport aircraft. The integrated intake is used in combat (military) aircraft. The flush intake is usually used in missiles since they can be more readily accommodated into missile airframes.

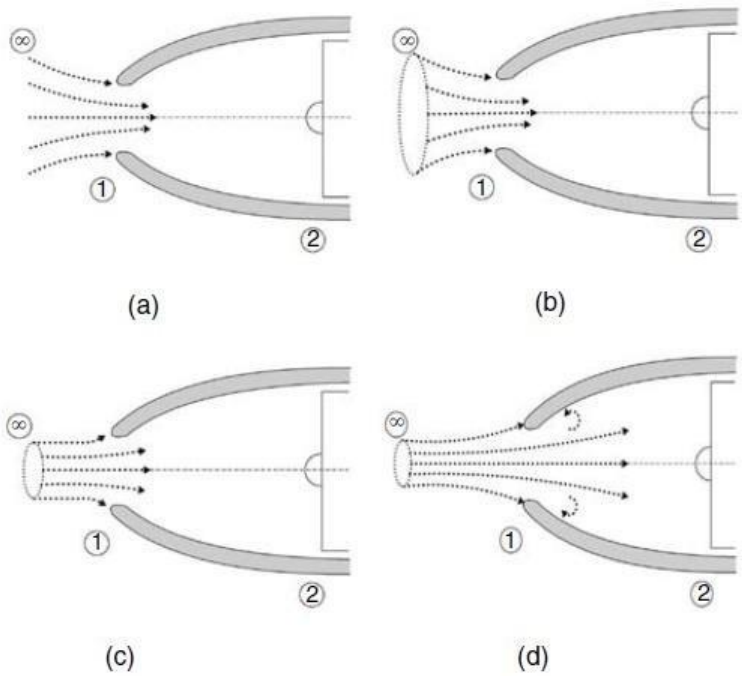


Fig. 2.2: Flow characteristics of podded intakes: (a) ground run, (b) climb, (c) high-speed cruise, and (d) top speed.

Subsonic inlet performance

Depending on the flight speed and the mass flow demanded by the engine, the inlet might have to operate with a wide range of incident stream conditions. Fig 2.3 a and b show the performances of subsonic intake during two typical subsonic conditions, takeoff and cruise, respectively. The first illustrates the stream tube, while the second depicts the pressure and speed variation and the third is a temperature–entropy diagram. The flow in intake is identified by three states, namely far upstream that is denoted as (∞), at the duct entry denoted by (1) and at the engine face denoted by (2). The flow outside the engine (fromstate ∞ to 1) is an isentropic one, where no losses are associated with the total temperature and pressure. For high speed, or cruise condition (Figure2.3b), the stream tube will have a divergent shape and following conditions can be stated:

$$u1 < u\infty,\ P1 > P\infty,\ P01 = P0\infty,\ T01 = T0\infty$$

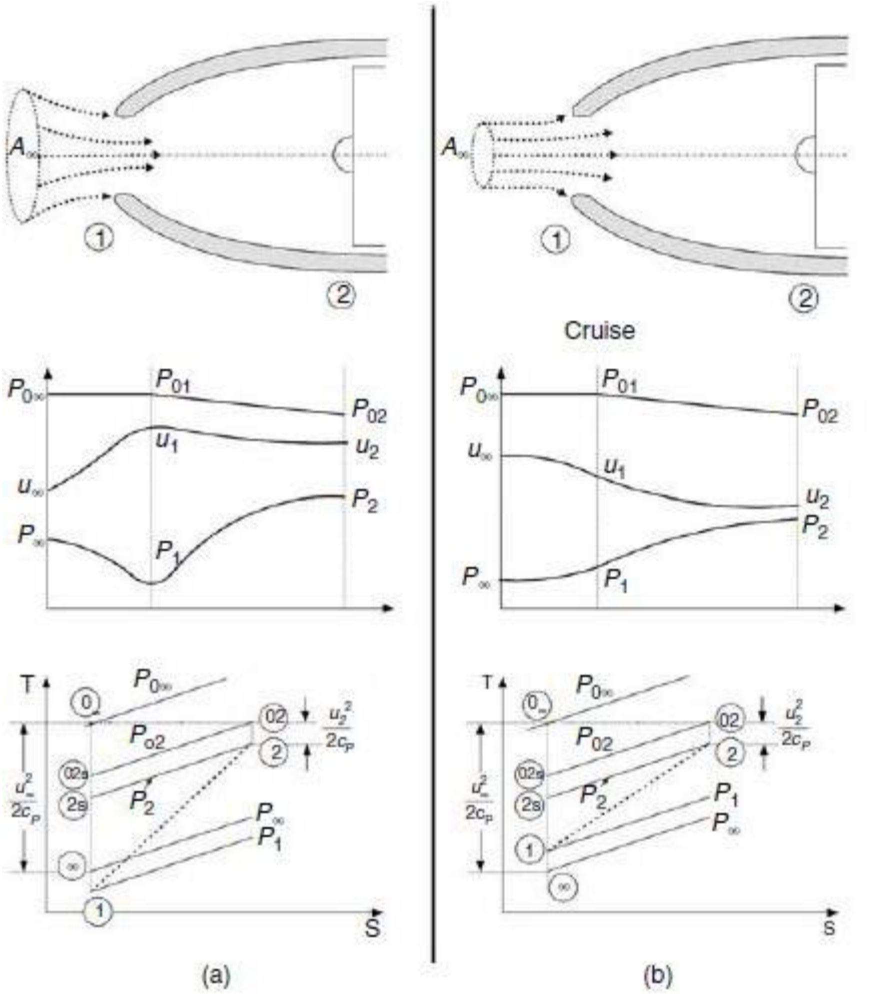


Fig. 2.2: Subsonic inlet during (a) takeoff and (b) cruise.

During low speed high-thrust operation (e.g., during takeoff and climb), as shown in Figure 2.3a,the same engine will demand more mass flow and the air stream upstream the intake will beaccelerated. The stream tube will have a converging shape and the following conditions are satisfied:

$$u1 > u\infty,\ P1 < P\infty,\ P01 = P0\infty,\ T01 = T0\infty$$

For both cases of takeoff and cruise, there will be internal diffusion within the intake up to the engine face. The static pressure will rise and the air speed will be reduced. The total pressure will also decrease owing to skin friction while the total temperature remains unchanged, as the flow through diffuser is adiabatic. Thus, for both takeoff and cruise conditions

$$P2 > P1,\ P02 < P0\infty,\ u2 < u1$$

Since the inlet speed to the engine (compressor/fan) should be nearly constant for different operating conditions, then

$$\left(\frac{P_2}{P_1}\right)_{\text{takeoff}} > \left(\frac{P_2}{P_1}\right)_{\text{cruise}}$$

If this pressure increase is too large, the diffuser may stall due to boundary layer separation.

Stalling usually reduces the stagnation pressure of the stream as a whole. Conversely, for cruise conditions (Fig.2.3b) to avoid separation or to have a less severe loading on the boundary layer, it is recommended to have a low velocity ratio ($u1/u\infty$) and consequently less internal pressure rise. Therefore, the inlet area is often chosen so as to minimize external acceleration during takeoff with the result that external deceleration occurs during level-cruise operation. Under these conditions the ―upstream capture area‖ $A\infty$ is less than the inlet area A1, and some flow is spilled over the inlet.

Supersonic intakes

The design of inlet systems for supersonic aircraft is a highly complex matter involving engineering trade-offs between efficiency, complexity, weight, and cost. A typical supersonic intake is made up of a supersonic diffuser, in which the flow is decelerated by a combination of shocks and diffuse compression, and a subsonic diffuser, which reduces the Mach number from high subsonic value after the last shock to the value acceptable to the engine. Subsonic intakes that have thick lip are quite unsuitable for supersonic speeds. The reason is that a normal shock wave ahead of the intake is generated, which will yield a very sharp static pressure rise without change of flow direction and correspondingly big velocity reduction. The adiabatic efficiency of compression through a normal shock wave is very low as compared with oblique shocks. At Mach 2.0 in the stratosphere adiabatic efficiency would be about 80% or less for normal shock waves, whereas its value will be about 95% or even more for an intake designed for oblique shocks.

Flight at supersonic speeds complicates the diffuser design for the following reasons

1. The existence of shock waves that lead to large decrease in stagnation pressure even in the absence of viscouseffects.

2. The large variation in capture stream tube area between subsonic and supersonic flight for a given engine, as much as a factor of four between $M\infty = 1$ and $M\infty = 3$.

3. As $M\infty$ increases, the inlet compression ratio becomes a larger fraction of the overall cycle compression ratios and as a result, the specific thrust becomes more sensitive to diffuser pressure ratio.

4. It must operate efficiently both during the subsonic flight phases (takeoff, climb, and subsonic cruise) and at supersonic designspeed.

Supersonic intake may be classified as follows

1. Ax symmetric or two-dimensionalintakes

The axisymmetric intakes use axisymmetric central cone to decelerate the flow down to subsonic speeds. The two-dimensional inlet has rectangular cross sections as found in the F-14 and F-15 fighter aircraft.

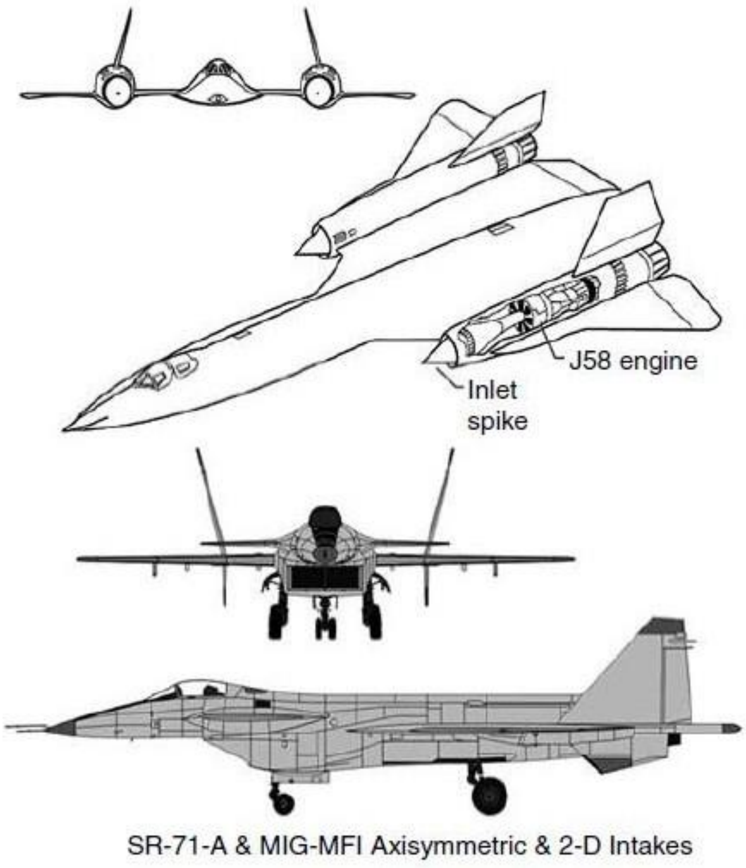


Fig. 2.2: Axisymmetric and two-dimensional supersonic intakes.

2. Variable or fixedgeometry

For variable geometry ax symmetric intakes, the central cone may move fore-and-aft to adjust the intake area. Alternatively, the inlet area is adjusted in the case of rectangular section through hinged flaps (or ramps) that may change its angles. For flight at Mach numbers much beyond 1.6, variable geometry features must be incorporated in the inlet to achieve high inlet pressure recoveries together with low externaldrag.

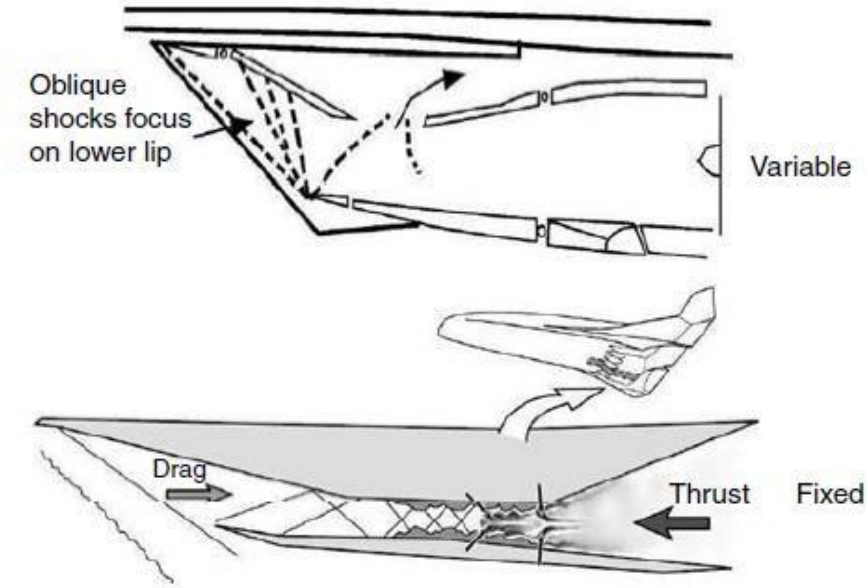


Fig. 2.3: Variable and fixed geometry supersonic intakes.

3. Internal, external or mixedcompression

As shown in Fig 2.4. The set of shocks situated between the fore body and intake lip are identified as external shocks, while the shocks found between the nose lip and the intakes throat are called internal shocks. Some intakes have one type of shocks either external or internal and given the same name as the shocks, while others have both types and denoted as mixed compression intakes.

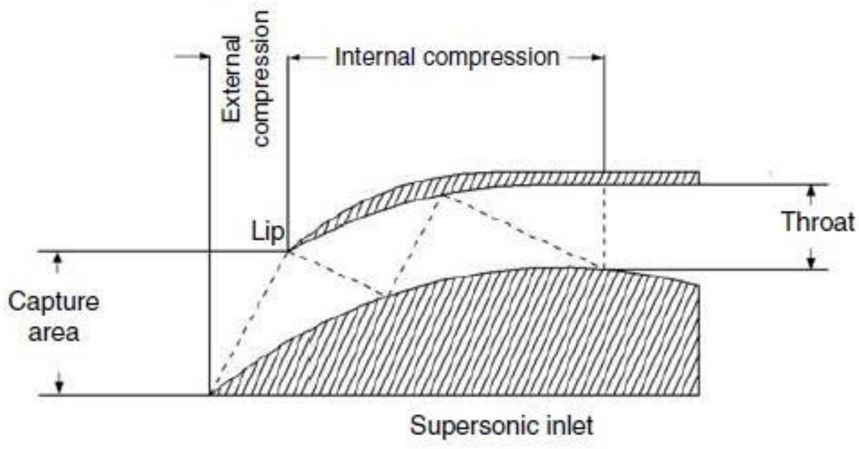


Fig. 2.4: External and internal compression supersonic intake**.**

External compression intake

External compression intakes complete the supersonic diffusion process outside the covered portion of the inlet where the flow is decelerated through a combination of oblique shocks (may be a single, double, triple, or multiple). These oblique shocks are followed by a normal shock wave that changes the flow from supersonic to subsonic flow. Both the normal shock wave and the throat are ideally located at the cowl lip. The supersonic diffuser is followed by a subsonic diffuser, which reduces the Mach number from high subsonic value after the last shock to the value acceptable to the engine. The simplest form of staged compression is the single oblique shock, produced by a single-angled wedge or cone that projects forward of the duct, followed by a normal shock as illustrated in Fig 2.5.The intake in this case is referred to as a two-shock intake. With a wedge, the flow after the oblique shock wave is at constant Mach number and parallel to the wedge surface. With a cone the flow behind the conical shock is itself conical, and the Mach number is constant along rays from the apex and varies along streamline. Fore body intake isfrequently used for external compression intake of wedge or coneform‖.

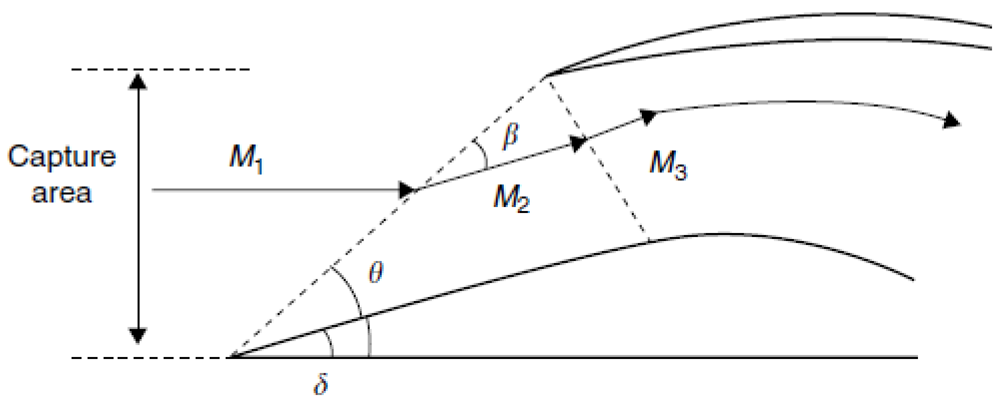


Fig. 2.5: Single oblique shock external compression intake.

Mode of operation
Critical condition

The capture area (Ac) for supersonic intakes is defined as the area enclosed by the leading edge, or ―highlight,‖ of the intake cowl, including the cross-sectional area of the fore body in that plane. The maximum flow ratio is achieved when the boundary of the free stream tube (A∞) arrives undisturbed at the lip. This means

$$\frac{A_\infty}{A_c} = 1.0$$

This condition is identified as the full flow [6] or the critical flow [5]. This condition depends on the Mach number, angle of the fore body and the position of the tip. In this case, the shock angle θ is equal to the angle subtended by lip at the apex of the body and corresponds to the maximum possible flow through the intake.

Subcritical operation

At Mach numbers (or speeds) below the value of the critical (design) value described above, the mass flow is less than that at the critical condition and the normal shock wave occurs in front of the cowl lip and this case is identified as subcritical. It is to be noted here that

$$\frac{A_\infty}{A_c} < 1.0$$

Supercritical operation

If the at air speeds is greater than the design value, then the oblique shock will impinge inside the cowl lip and the normal shock will move to the diverging section. This type of operation is referred to as the supercriticaloperation.

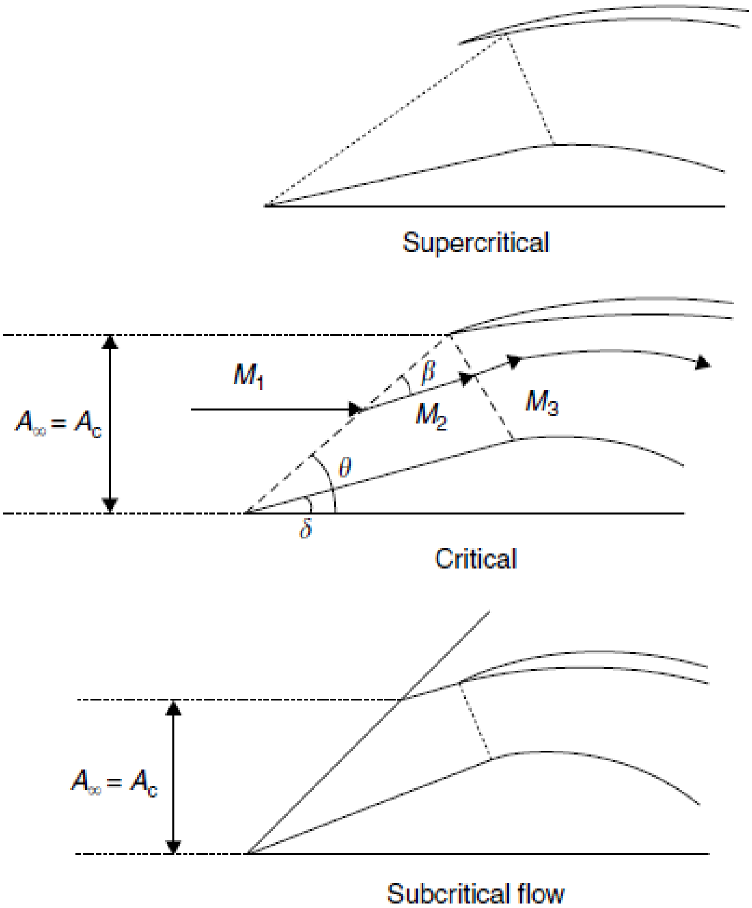


Fig. 2.5: Types of flow in an external compression intake.

Internal compression inlet

The internal compression inlet locates all the shocks within the covered passageway (Fig 2.6). The terminal shock wave is also a normal one, which is located near or at the throat. A principal difference between internal and external compression intakes is that with internal compression, since the system is enclosed, oblique shocks are reflected from an opposite wall, which have to be considered. The simplest form is a three-shock system. The single-wedge turns the flow toward the opposite wall. The oblique shock is reflected from the opposite wall and the flow passing the reflected shock is restored to an axial direction. A normal shock terminates the supersonic asusual.

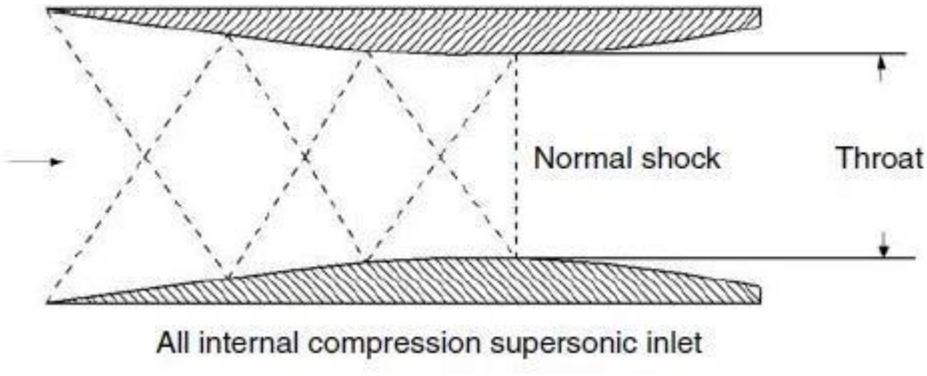


Fig. 2.5: Internal compression supersonic intake.

Over-speeding

Consider an intake designed for a Mach number of 1.7. The design point for this intake, represented by the point B in this plot, has been shown in Fig. 2.6.

One of the methods of starting the intake is by increasing the free-stream Mach number. Referring Fig. 2.6, conditions are modified so as to move the operating point from B to C. Once at C the intake starts and the expected shock system is developed. Then again the free-stream flow is decelerated to reach the design point B, while retaining the appropriate shock structure. This method of starting the intake is called ‗Over-speeding'. It is the simplest method for starting the intake from the point of view of design complexity as it does not require any modifications to the intake aircraft.

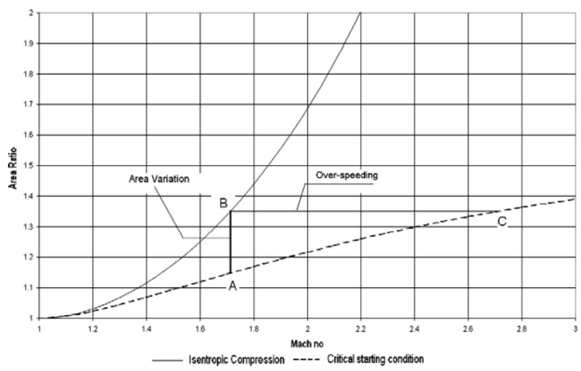


Fig 2.6: Method of Over-speeding and Area Variation for Starting an Intake

However, from Fig. 2.6, it can be seen that the amount of over-speeding required even for modest area ratios is very high. Hence, this method can best be employed for intakes aircrafts designed for a very low supersonic Mach number. Also, as has been discussed in Section 3.2, there exists a limiting area ratio (about 1.66) beyond which the intake cannot be started even if accelerated it to an infinitely high Mach number. Hence the method of over-speeding cannot be used for intakes which are designed for Mach numbers beyond 2. Another problem with this method is that the intake remains in the _non-started ‘condition for quite some time in the supersonic flight regime, which is not acceptable. Use of the method of _over- speeding ‘has not been reported on any aircraft and its study is primarily of academic interest. As a result of its deficiencies, other methods need to be explored for overcoming the starting problem.

Variable area

Referring again to Fig. 2.6, we can see that another way of modifying the conditions at point B so as to enable flow starting is by varying the area ratio, without over-speeding. The area ratio at B can be reduced so as to reach point A. Once the intake starts at point A, the area ratio can again be modified to operate at the design point. This method is referredto as variation of Intake Geometry ‘and can be used for starting intakes designed even for high Mach numbers. However, it does require complex mechanisms for varying the area ratio. This adds to cost, weight and complexity. This method can further be categorized depending on the mechanism of modifying the intakegeometry.

a) Rotation of a pivotedcowl

In this method, a cowl is pivoted to the main body and the position of the other end of the cowl is controlled, so as to vary the intake geometry. By this method, the entry area can be changed, thereby modifying the overall area ratio. This approach is relatively less complicated and has been employed in hypersonicintakes

b) Lateral movement of the ramp / cowl /centre-body

By moving the ramp or cowl or centre-body, a variation in the throat area as well as the inlet area can be achieved. Such a system has considerable flexibility and has been used in most military aircrafts. The centre-body system is for axisymmetric intakes. Although slightly more complex in design due to the requirement of moving a larger portion of the intake, this method gives better off-design performance in terms of matching the intake and hence enjoys wide-spreadapplication.

Porous bleed

Another method employed for starting intakes is the use of porosity in the cowl. By doing this, the back-pressure at the throat is reduced so that, the normal shock can be positioned. Also, the use of bleed allows additional flow spillage without creating adverse conditions at the intake entry. The effective reduction in back pressure is dictated by the amount of mass removed using porous bleed and thus by the size and density of the holes used. Use of porosity in a 2D intake at the wedge has been experimentally studied. The total pressure recovery as well as drag increase as a function of Mach number. These have been however found to be within limits, when compared with a solid wedge intake. Other factors that dictate the performance characteristics include the amount of bleed and its location. Cub bison, et al has experimentally studied the effects of variation in the location and bleed amount on the performance in ax symmetric intakes. It has been reported that the best starting and stability characteristics were obtained when porosity was located just downstream of the terminal shock. Also, the intake performance in terms of pressure recovery has been reported to have improved with an increase in bleed percentage, though at the cost of increased inlet flow distortion.

Thus, it is seen that the use of porosity does enhance the overall performance of the intake, particularly under off-design conditions. Also, this method does not require any of the complicated and weight intensive equipment as needed in case of variable geometry. However, one of the major disadvantages associated with this method is the flow distortion, and this has been the major deterrent for most applications. Also, there are always losses related to mass flow even after the intake has started. This leads to a lower critical performance. An improvement suggested is the use of variable bleed by controlling the bleed plenum exit area. This however comes at the cost of added complexity.

Other methods for Starting

Innovative methods for starting an intake designed for high supersonic and hypersonic speeds have been discussed in this section. The capture cross-section in hypersonic aircraft intakes employing the airframe-integrated-scramjet concept has been recommended to be rectangular. The initial compression is performed by the vehicle bow-shock. The intake is then expected to start at ramjet speeds (around Mach 4) and operate over a large Mach number range. One of the proposed designs employs a rectangular capture area and elliptic throat (also known as REST; acronym for Rectangular to Elliptic Shape Transition). The combustor too is elliptic allowing better combustion characteristics. Three views of such a design are depicted in Fig. 3.5. Key features of this type of intakeinclude

- Self-starting characteristics at Mach4.
- Area ratio greater than the critical area ratio required forstarting.

Starting is assisted by the use of spillage holes on the side walls and the peculiar shape of the cowl. Due to its simple construction which excludes any moving parts, this configuration seems to have a good scope for development in the future.

Combustion chamber

The combustion process in aircraft engines and gas turbines is a heat addition process to the compressed air in the combustor or burner. Thus, the combustion is a direct-fired air heater in which fuel is burned. The combustor is situated between the compressor and turbine, where it accepts air from the compressor and delivers it at elevated temperature to the turbine.

Main requirements from gas turbine combustors

1. Its length and frontal area to remain within the limits set by other engine components, that is, size and shape compatible with engineenvelop.

2. Its diffuser minimizes the pressureloss.

3. The presence of a liner to provide stable operation [i.e., the flame should stay alight over a wide range of air–fuel ratios(AFRs)].

4. Fulfils the pollutant emissions regulations (low emissions of smoke, unburned fuel, and gaseous pollutantspecies).

5. Ability to utilize much broader range offuels.

6. Durability and relightingcapability.

7. High combustion efficiency at different operating conditions: (1) altitude ranging from sea level to 11 km for civil transport and higher for some military aircraft and (2) Mach numbers ranging from zero during ground run to supersonic for militaryaircraft.

8. Design for minimum cost and ease ofmaintenance.

9. An outlet temperature distribution (pattern form) that is tailored to maximize the life of the turbine blades and nozzle guidevanes.

10. Freedom from pressure pulsations and other manifestations of combustion-induced instabilities.

11. Reliable and smooth ignition both on the ground (especially at very low ambient temperature) and in the case of aircraft engine flameout at highaltitude.

12. The formation of carbon deposits (coking) must be avoided, particularly the hard brittle variety. Small particles carried into the turbine in the high velocity gas stream can erode the blades. Furthermore, aerodynamically excited vibration in the combustion chamber might cause sizeable pieces of carbon to break free, resulting in even worse damage to theturbine.

NOZZLES

A nozzle (from nose, meaning 'small spout') is a tube of varying cross-sectional area (usually axisymmetric) aiming at increasing the speed of an outflow, and controlling its direction and shape. Nozzle flow always generates forces associated to the change in flow momentum, as we can feel by hand-holding a hose and opening the tap. In the simplest case of a rocket nozzle, relative motion is created by ejecting mass from a chamber backwards through the nozzle, with the reaction forces acting mainly on the opposite chamber wall, with a small contribution from nozzle walls. As important as the propeller is to shaft-engine propulsions, so it is the nozzle to jet propulsion, since it is in the nozzle that thermal energy (or any other kind of high-pressure energy source) transforms into kinetic energy of the exhaust, and its associated linear momentum producing thrust.

The flow in a nozzle is very rapid (and thus adiabatic to a first approximation), and with very little frictional loses (because the flow is nearly one-dimensional, with a favorable pressure gradient except if shock waves form, and nozzles are relatively short), so that the isentropic model all along the nozzle is good enough for preliminary design. The nozzle is said to begin where the chamber diameter begins to decrease (by the way, we assume the nozzle is axisymmetric, i.e. with circular cross-sections, in spite that rectangular cross-sections, said two-dimensional nozzles, are sometimes used, particularly for their ease of direction ability). The meridian nozzle shape is irrelevant with the 1D isentropic models; the flow is only dependent on cross-section area ratios.

Real nozzle flow departs from ideal (isentropic) flow on two aspects:

- Non-adiabatic effects. There is a kind of heat addition by non-equilibrium radical-species recombination and a heat removal by cooling the walls to keep the strength of materials in long-duration rockets (e.g. operating temperature of cryogenic SR-25 rockets used in Space Shuttle is 3250 K, above steel vaporization temperature of 3100 K, not just melting, at 1700 K). Short-duration rockets (e.g. solid rockets) are not actively cooled but rely on ablation; however, the nozzle-throat diameter cannot let widen too much, and reinforced materials (e.g. carbon, silica) are used in the throat region.

- There is viscous dissipation within the boundary layer, and erosion of the walls, what can be critical if the erosion widens the throat cross-section, greatly reducing exit-area ratio and consequentlythrust.
- Axial exit speed is lower than calculated with the one-dimensional exit speed, when radial outflow is accountedfor.

We do not consider too small nozzles, say with chamber size <10 mm and neck size <1 mm, where the effect of boundary layers become predominant.

Restricting the analysis to isentropic flows, the minimum set of input parameters to define the propulsive properties of a nozzle is:

- Nozzle size, given by the exit area, A_e; the actual area law, provided the entry area is large enough that the entry speed can be neglected, only modifies the flow inside the nozzle, but not the exitconditions.
- Type of gas, defined with two independent properties for a perfect-gas model, that we take as the thermal capacity ratio □$\equiv c_p/c_v$, and the gas constant, $R \equiv R_u/M$, and with R_u=8.314 J/(mol·K) and M being the molar mass, which we avoid using, to reserve the symbol M for the Mach number. If $c_p$is given instead of □, then we compute it from □$\equiv c_p/c_v = c_p/(c_p$□$R)$, having used Mayer's relation,c_p□c_v□R.
- Chamber (or entry) conditions: p_c and T_c (a relatively large chamber cross-section, and negligible speed, is assumed at the nozzle entry: A_c□□, M_c□0). Instead of subscript 'c' for chamber conditions, we will use 't' for total values because the energy conservation implies that total temperature is invariant along the nozzle flow, and the non-dissipative assumption implies that total pressure is also invariant, i.e. $T_t=T_c$ and $p_t=p_c$.
- Discharge conditions: p_0, i.e. the environmental pressure (or back pressure), is the only variable of importance (because pressure waves propagate at the local speed of sound and quickly tend to force mechanical equilibrium, whereas the environmental temperature T_0 propagates by much slower heat-transfer physical mechanisms). Do not confuse discharge pressure, p_0, with exit pressure, p_e, explainedbelow.

The objective is to find the flow conditions at the exit [p_e, T_e, v_e] for a given set of the above parameters, [A_e,□ ,R,p_c,T_c,,p_0], so that:

$$\dot{m} = \rho_e v_e A_e = \frac{p_e}{R T_e} v_e A_e, \quad F = \dot{m} v_e, \quad M_e = \frac{v_e}{\sqrt{\gamma R T_e}} \qquad (1)$$

If the nozzle flow is subsonic, then the exit pressure coincides with the discharge pressure, $p_e=p_0$, at the steady state (if at an initial state they were not equal, the time it would take to equalise is of the order of the nozzle length divided by the sound speed), and the other variables would be obtained from the isentropic relations, i.e.:

A converging nozzle can only become supersonic at the exit stage; the speed increases monotonically along the nozzle. If a converging nozzle is fed from a constant pressure constant temperature chamber, the flow rate grows as the discharge pressure is being reduced, until the flow becomes sonic (choked) and the flow rate no longer changes with further decreasing in discharge-pressure (a set of expansion waves adjust the exit pressure to this lower discharge pressure). Except for old-time turbojets and military fighter aircraft, all commercial jet engines (after Concorde was retired) use converging nozzles discharging at subsonic speed (both, the hot core stream and the colder fan stream).

A converging-diverging nozzle (CD-nozzle), is the only one to get supersonic flows with M>1 (when chocked). It was developed by Swedish inventor Gustaf de Laval in 1888 for use on a steam turbine. Supersonic flow in CD-nozzles presents a rich behavior, with shock waves and expansion waves usually taking place inside and/or outside. Several nozzle geometries have been used in propulsion systems:

1. The classical quasi-one-dimensional Laval nozzle, which has a slender geometry, with a rapidly converging short entrance, a rounded throat, and a long conical exhaust of some 15º half-cone angle (the loss of thrust due to jet divergence is about 1.7%). Rarely used in modernrockets.
2. Bell-shape nozzles (or parabolic nozzles), which are as efficiency as the simplest conical nozzle, but shorter and lighter, though more expensive to manufacture. They are the present standard in rockets; e.g. the Shuttle main engine (SME) nozzles yield 99% of the ideal nozzle thrust (and the remainder is because of wall friction, not because of wall shapeeffect).
3. Annular and linear nozzles, designed to compensate ambient pressure variation, like the Aero spike nozzle. They are underdevelopment.

We present below the 1D model of gas flow in nozzles. For more realistic design, beyond this simple model, a 2D (or axisymmetric) analysis by the method of characteristics and boundary layer effects should follow, to be completed with a full 3D nozzle-flow analysis by CFD.

Choked flow

Choking is a compressible flow effect that obstructs the flow, setting a limit to fluid velocity because the flow becomes supersonic and perturbations cannot move upstream; in gas flow, choking takes place when a subsonic flow reaches M=1, whereas in liquid flow, choking takes place when an almost incompressible flow reaches the vapour pressure (of the main liquid or of a solute), and bubbles appear, with the flow suddenly jumping toM>1.

Going on with gas flow and leaving liquid flow aside, we may notice that M=1 can only occur in a nozzle neck, either in a smooth throat where dA=0, or in a singular throat with discontinuous area slope (a kink in nozzle profile, or the end of a nozzle). Naming with a '*' variables the stage where M=1 (i.e. the sonic section, which may be a real throat within the nozzle or at some extrapolated imaginary throat downstream of a subsonic nozzle)

Area ratio

Nozzle area ratio □ (or nozzle expansion ratio) is defined as nozzle exit area divided by throat area, □≡A_e/A^*, in converging-diverging nozzles, or divided by entry area in converging nozzles. Notice that □ so defined is □>1, but sometimes the inverse is also named 'area ratio' (this contraction area ratio is bounded between 0 and 1); however, although no confusion is possible when quoting a value (if it is >1 refers to A_e/A^*, and if it is <1 refers to A^*/A_e), one must be explicit when saying 'increasing area ratio' (we keep to □≡A_e/A^*>1).

To see the effect of area ratio on Mach number, is plotted in Fig. 1 for ideal monatomic (□=5/3), diatomic (□=7/5=1.40), and low-gamma gases as those of hot rocket exhaust (□=1.20); gases like CO_2 and H_2O have intermediate values (□=1.3). Notice that, to get the same high Mach number, e.g. M=3, the area ratio needed is A^*/A=0.33 for □=1.67 and A^*/A=0.15 for □=1.20, i.e. more than double exit area for the same throat area (that is why supersonic wind tunnels often use a monatomic working gas.

Losses & Real Effects in Nozzles

• Flow divergence

• No uniformity

• poloss due to heataddition

• Viscouseffects

– Boundary layers-drag

--boundary layer-shock interactions

• Heatlosses

• Nozzle erosion(throat)

• Transients

• Multiphaseflow

• Real gasproperties

• Nonequilibriumflow.

Variable geometrynozzles

Variable area nozzle, which is sometimes identified as adjustable nozzle, is necessary for engines fitted with afterburners. Generally, as the nozzle is reduced in area, the turbine inlet temperature increases and the exhaust velocity and thrust increase. Three methods are available, namely:

1. Central plug at nozzleoutlet
2. Ejector typenozzle
3. IRISnozzle

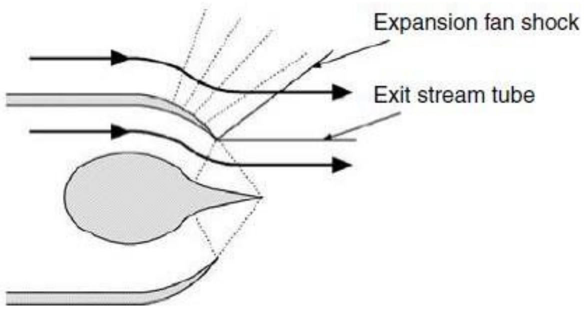


Fig 3.1: Plug nozzle at design point.

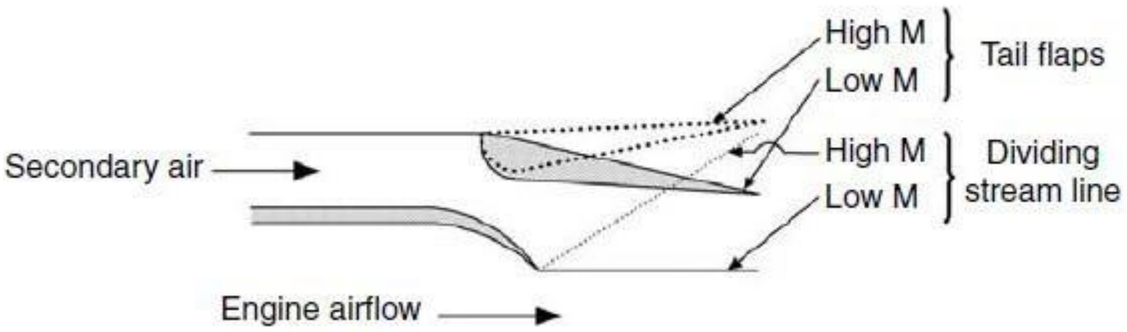


Fig 3.2: Variable geometry ejector nozzle.

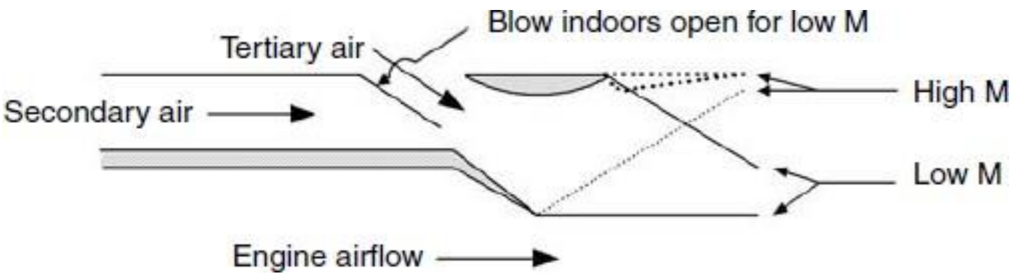


Fig 3.3: Ejector nozzle with blow-in doors for tertiary air.

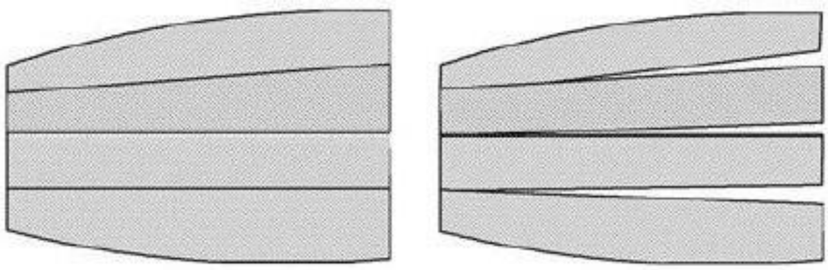


Fig 3.3: Iris variable nozzle.

Thrust reversal

Stopping an aircraft after landing is not an easy problem due to the increases in its gross weight, wing loadings and landing speeds. The amount of force required for stopping an aircraft at a given distance after touchdown increases with the gross weight of the aircraft and the square of the landing speed. The size of modern transport aircraft, which results in higher wing loadings and increased landing speeds, makes the use of wheel brakes alone unsatisfactory for routine operations. Moreover, in the cases of wet, icy, or snow-covered runways, the efficiency of aircraft brakes may be reduced by the loss of adhesion between aircraft tire and the runway.

On turbojet engines, low-bypass turbofan engines, whether fitted with afterburner or not, and mixed turbofan engines, the thrust reverser is achieved by reversing the exhaust gas flow (hot stream). On high-bypass ratio turbofan engines, reverse thrust is achieved by reversing the fan (cold stream) airflow. Mostly, in this case it is not necessary to reverse the hot stream as the majority of the engine thrust is derived from the fan, although some engines use both systems.

A good thrust reverser must fulfill the following conditions

2. Must not affect the engine operation whether the thrust reverser is applied orstowed

3. Withstand high temperature if it is used in the turbineexhaust

4. Mechanicallystrong

5. Relatively light inweight

6. When stowed should be streamlined into the engine nacelle and should not add appreciably to the frontal area of theengine

7. Reliable and failsafe

8. Cause few increased maintenanceproblems

9. Provide at least 50% of the full forwardthrust.

Classification of thrust reverser systems

The most commonly used reversers are clamshell-type, external-bucket type doors and blocker doors, as shown in Fig 3.4.

Clamshell door system—sometimes identified as preexist thrust reverser [9]—is a pneumatically operated system. When reverse thrust is applied, the doors rotate to uncover the ducts and close the normal gas stream. Sometimes clamshell doors are employed together with cascade vanes; type (A) in Figure 3.4. Clamshell type is normally used for non-afterburningengines.

The bucket target system; type (B) in Fig3.4, is hydraulically actuated and uses bucket-type doors to reverse the hot gas stream. Sometimes it is identified as post exit or target thrust reverser. In the forward (stowed) thrust mode, the thrust reverser doors form the convergent–divergent final nozzle for the engine. When the thrust reverser is applied, the reverser automatically opens to form a ―clamshell‖ approximately three-fourth to one nozzle diameter to the rear of the engine exhaust nozzle. When the thrust reverser is applied, the reverser automatically opens to form a ―clamshell‖ approximately three-fourth to one nozzle diameter to the rear of the engine exhaust nozzle. The thrust reverser in Boeing 737-200 aircraft is an example for this type of thrust reverser.

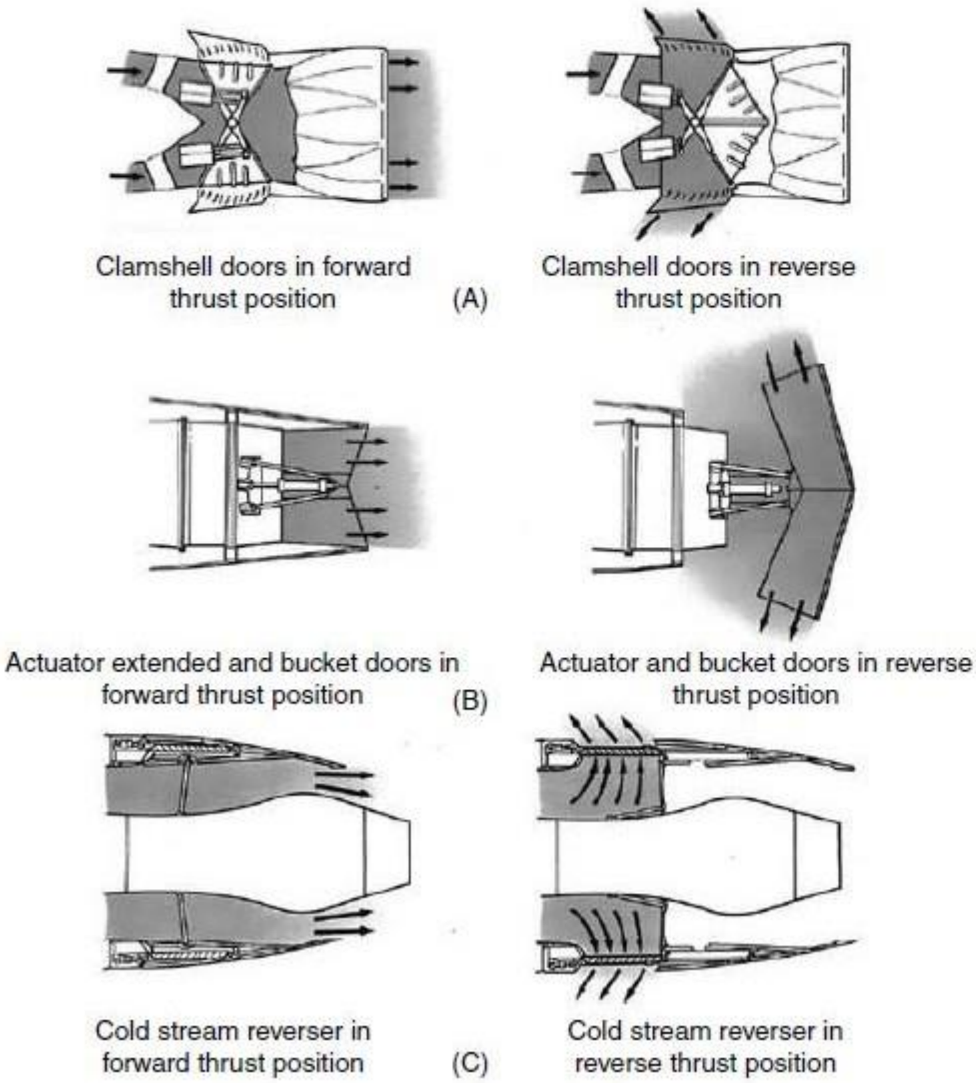


Fig 3.4: Methods of thrust reversal

High by-pass turbofan engines normally use blocker doors to reverse the cold stream airflow; type (C) in Figure 3.4. Cascade-type reverser uses numerous turning vanes in gas path to direct the gas flow outward and forward during operation. Some types utilize a sleeve to cover the fan cascade during forward thrust. Aft movement of the reverse sleeves causes blocker doors to blank off the cold stream final nozzle and deflect fan discharge air forward through fixed cascade vanes, producing reverse thrust. In some installations, cascade turning vanes are used in conjunction with clamshell to reverse the turbine exhaust gases. Both the cascade and the clamshell are located forward of the turbine exhaust nozzle. For reverse thrust, the clamshell blocks the flow of exhaust gases and exposes the cascade vanes, which act as an exhaust nozzle. Some installations in low-bypass turbofan unmixed engines use two sets of cascade: forward and rearward. For the forward cascade, the impinging exhaust is turned by the blades in the cascade into the forward direction. Concerning the rearward cascade, the exhaust from the hot gas generator strikes the closed clamshell doors and is diverted forward and outward through a cascade installed in these circumferential openings in the enginenacelle.

UNIT - IV
THERMODYNAMICS OF REACTING SYSTEMS

A combustor is a component or area of a gas turbine, ramjet, or scramjet engine where combustion takes place. It is also known as a burner, combustion chamber or flame holder. In a gas turbine engine, the combustor or combustion chamber is fed high pressure air by the compression system. The combustor then heats this air at constant pressure. After heating, air passes from the combustor through the nozzle guide vanes to the turbine. In the case of a ramjet or scramjet engines, the air is directly fed to the nozzle.

A combustor must contain and maintain stable combustion despite very high air flow rates. To do so combustors are carefully designed to first mix and ignite the air and fuel, and then mix in more air to complete the combustion process. Early gas turbine engines used a single chamber known as a can type combustor. Today three main configurations exist: can, annular and cannular (also referred to as can-annular tubo-annular). Afterburners are often considered another type of combustor.

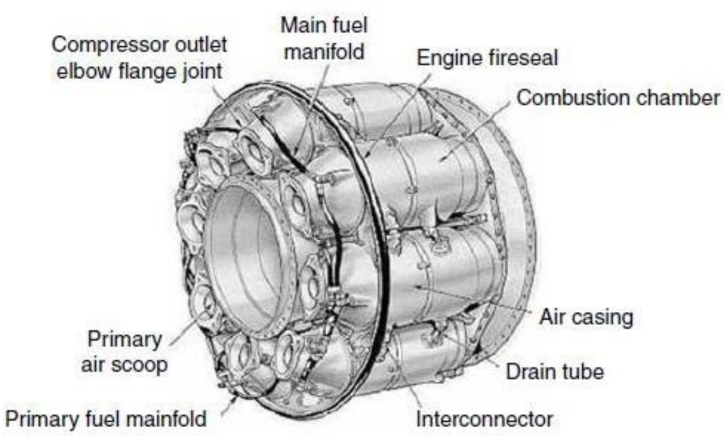
Combustors play a crucial role in determining many of an engine's operating characteristics, such as fuel efficiency, levels of emissions and transient response (the response to changing conditions such as fuel flow and air speed

Classification of combustion chambers

There are three main types of subsonic combustion chambers in use in gas turbine engines, namely, multiple chamber (tubular or can type), tubo-annular chamber, and the annular chamber.

Tubular (or can) combustion chambers

Tubular type is sometimes identified as multiple- or can-type combustion chamber. As shown in Fig 2.7 this type of combustor is composed of cylindrical chambers disposed around the shaft connecting the compressor and turbine. Compressor delivery air is split into a number of separate streams, each supplying a separate chamber. These chambers are interconnected to allow stabilization of any pressure fluctuations. Ignition starts sequentially with the use of twoigniters.



Mutiple can

Fig 2.7: Multiple combustion chambers.

The number of combustion chambers varies from 7 to 16 per engine. The can-type combustion chamber is typical of the type used on both centrifugal and axial-flow engines. It is particularly well suited for the centrifugal compressor engine since the air leavingthe

Compressor is already divided into equal portions as it leaves the diffuser vanes. It is then a simple matter to duct the air from the diffuser into the respective combustion chambers arranged radially around the axis of the engine.

The advantages of tubular type are as follows:

- Mechanicallyrobust
- Fuel flow and airflow patterns are easilymatched
- Rig testing necessitates only a small fraction of total engine air massflow
- Easy replacement formaintenance.

The disadvantages are as follows:

• Bulky andheavy

• High pressureloss

• Requiresinterconnectors

• Incur problem oflight-round

• Large frontal area and highdrag

Turbo-annular combustion chambers

This type may also be identified as can-annular or cannular. It consists of a series of cylindrical burners arranged within common single annulus as is shown in Fig 2.8. Thus, it bridges the evolutionary gap between the tubular (multiple) and annular types. It combines the compactness of the annular chamber with the best features of the tubular type. The combustion chambers are enclosed in a removable shroud that covers the entire burner section. This feature makes the burners readily available for any required maintenance. Cannular combustion chambers must have fuel drain valves in two or more of the bottom chambers. This ensures drainage of residual fuel to prevent its being burned at the next start. The flow of air through the holes and louvers of the can-annular system is almost identical to the flow through other types of burners. Reverse-flow combustors are mostly of the can-annular type. Reverse-flow combustors make the engine morecompact.

Advantages of can-annular types are as follows:

• Mechanicallyrobust.

• Fuel flow and airflow patterns are easilymatched.

• Rig testing necessitates only a small fraction of total engine air massflow.

• Shorter and lighter than tubularchambers.

• Low pressureloss.

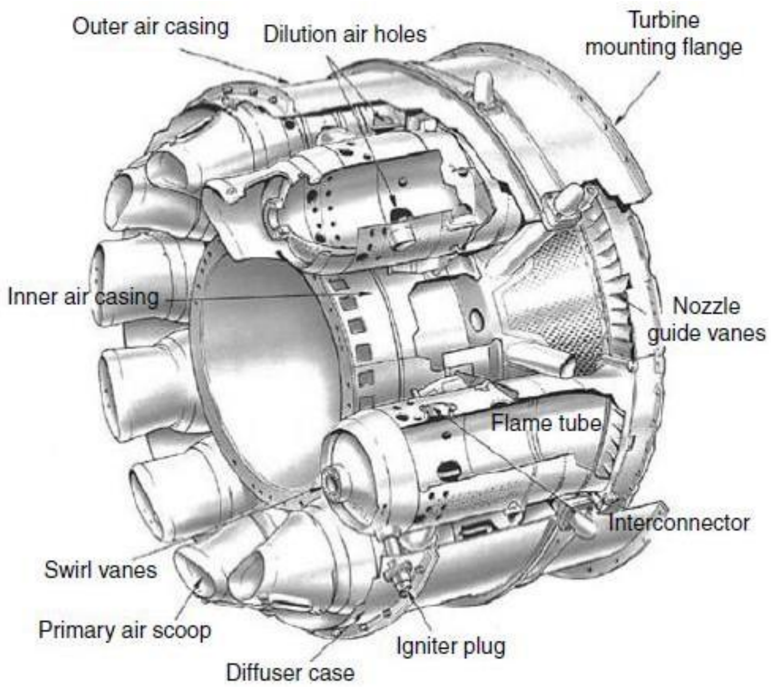


Fig2.8: Turbo-annular combustion chamber

Their disadvantages are as follows:

• Less compact thanannular

• Requiresconnectors

• Incur a problem of lightaround.

Annular combustion chambers

In this type an annular liner is mounted concentrically inside an annular casing. This combustor representsthe idealconfiguration forcombustorssince its―cleanǁ aerodynamic layout results in compact dimensions (and consequently an engine of small diameter) (Fig 2.9) and lower pressure loss than other designs. Usually, enough space is left between the outer liner wall and the combustion chamber housing to permit the flow of cooling air from the compressor. Normally, this type is used in many engines using axial-flow compressor and also others incorporating dual type compressors (combinations of axial flow and centrifugal flow). Currently, most aero engines use annular typecombustors.

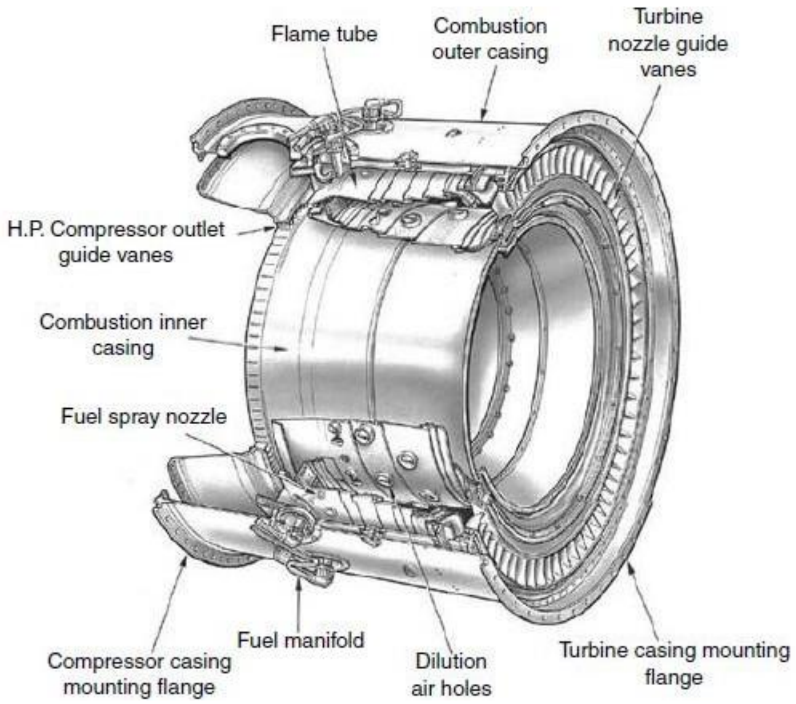


Fig 2.8: Annular type combustor.

The advantages of annular type may be summarized as follows:

• Minimum length and weight (its length is nearly 0.75 of cannular combustorlength).

• Minimum pressureloss.

• Minimum engine frontalarea.

• Less wall area than cannular and thus cooling air required is less; thus the combustion efficiency rises as the unburnt fuel is reduced.

• Easylight-round.

• Designsimplicity.

• Combustion zoneuniformity.

• Permits better mixing of the fuel andair.

• Simple structure compared to canburners.

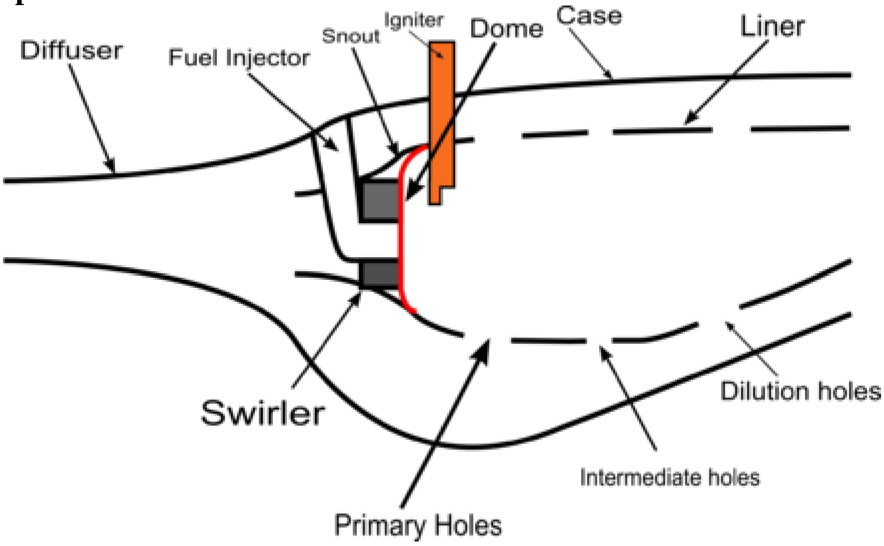
• Increaseddurability.

Disadvantages

• Serious buckling problem on outerliner.

• Rig testing necessitates full engine air massflow.

Components



Case

The case is the outer shell of the combustor, and is a fairly simple structure. The casing generally requires little maintenance.[4] The case is protected from thermal loads by the air flowing in it, so thermal performance is of limited concern. However, the casing serves as a pressure vessel that must withstand the difference between the high pressures inside the combustor and the lower pressure outside. That mechanical (rather than thermal) load is a driving design factor in the case.[5]

Diffuser

The purpose of the diffuser is to slow the high speed, highly compressed, air from the compressor to a velocity optimal for the combustor. Reducing the velocity results in an unavoidable loss in total pressure, so one of the design challenges is to limit the loss of pressure as much as possible. Furthermore, the diffuser must be designed to limit the flow distortion as much as possible by avoiding flow effects like boundary layer separation. Like most other gas turbine engine components, the diffuser is designed to be as short and light as possible.

Liner

The liner contains the combustion process and introduces the various airflows (intermediate, dilution, and cooling, see Air flow paths below) into the combustion zone. The liner must be designed and built to withstand extended high temperature cycles. For that reason liners tend to be made from super alloys like Hastelloy X. Furthermore, even though high performance alloys are used, the liners must be cooled with air flow.Some combustors also make use of thermal barrier coatings. However, air cooling is still required. In general, there are two main types of liner cooling; film cooling and transpiration cooling. Film cooling works by injecting (by one of several methods) cool air from outside of the liner to just inside of the liner. This creates a thin film of cool air that protects the liner, reducing the temperature at the liner from around 1800 Kelvin's (K) to around 830 K, for example. The other type of liner cooling, transpiration cooling, is a more modern approach that uses a porous material for the liner. The porous liner allows a small amount of cooling air to pass through it, providing cooling benefits similar to film cooling. The two primary differences are in the resulting temperature profile of the liner and the amount of cooling air required. Transpiration cooling results in a much more even temperature profile, as the cooling air is uniformly introduced through pores. Film cooling air is generally introduced through slats or louvers, resulting in an uneven profile where it is cooler at the slat and warmer between the slats. More importantly, transpiration cooling uses much less cooling air (on the order of 10% of total airflow, rather than 20-50% for film cooling). Using less air for cooling allows more to be used for combustion, which is more and more important for high performance, high thrust engines.

Snout

The snout is an extension of the dome (see below) that acts as an air splitter, separating the primary air from the secondary air flows (intermediate, dilution, and cooling air; see Air flow paths section below).

Dome / swirler

The dome and swirler are the part of the combustor that the primary air (see Air flow paths below) flows through as it enters the combustion zone. Their role is to generate turbulence in the flow to rapidly mix the air with fuel. Early combustors tended to use bluff body domes (rather than swirlers), which used a simple plate to create wake turbulence to mix the fuel and air. Most modern designs, however, are swirl stabilized (use swirlers). The swirler establishes a local low pressure zone that forces some of the combustion products to recirculate, creating the high turbulence. However, the higher the turbulence, the higher the pressure loss will be for the combustor, so the dome and swirler must be carefully designed so as not to generate more turbulence than is needed to sufficiently mix the fuel and air.

Fuel injector

Fuel injectors of a cannular combustor on a Pratt & Whitney JT9D turbofan

The fuel injector is responsible for introducing fuel to the combustion zone and, along with the swirler (above), is responsible for mixing the fuel and air. There are four primary types of fuel injectors; pressure-atomizing, air blast, vaporizing, and premix/prevaporizing injectors. Pressure atomizing fuel injectors rely on high fuel pressures (as much as 3,400 kilopascals (500 psi)) to atomize the fuel. This type of fuel injector has the advantage of being very simple, but it has several disadvantages. The fuel system must be robust enough to withstand such high pressures, and the fuel tends to be heterogeneously atomized, resulting in incomplete or uneven combustion which has more pollutants and smoke.

The second type of fuel injector is the air blast injector. This injector "blasts" a sheet of fuel with a stream of air, atomizing the fuel into homogeneous droplets. This type of fuel injector led to the first smokeless combustors. The air used is just same amount of the primary air (see Air flow paths below) that is diverted through the injector, rather than the swirler. This type of injector also requires lower fuel pressures than the pressure atomizing type.

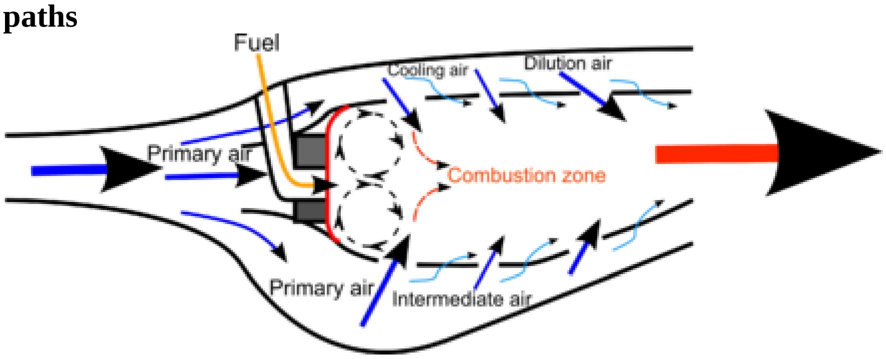
The vaporizing fuel injector, the third type, is similar to the air blast injector in that primary air is mixed with the fuel as it is injected into the combustion zone. However, the fuel-air mixture travels through a tube within the combustion zone. Heat from the combustion zone is transferred to the fuel-air mixture, vaporizing some of the fuel (mixing it better) before it is combusted. This method allows the fuel to be combusted with less thermal radiation, which helps protect the liner. However, the vaporizer tube may have serious durability problems with low fuel flow within it (the fuel inside of the tube protects the tube from the combustion heat).

The premixing/prevaporizing injectors work by mixing or vaporizing the fuel before it reaches the combustion zone. This method allows the fuel to be very uniformly mixed with the air, reducing emissions from the engine. One disadvantage of this method is that fuel may auto-ignite or otherwise combust before the fuel-air mixture reaches the combustion zone. If this happens the combustor can be seriously damaged.

Igniter

Most igniters in gas turbine applications are electrical spark igniters, similar to automotive spark plugs. The igniter needs to be in the combustion zone where the fuel and air are already mixed, but it needs to be far enough upstream so that it is not damaged by the combustion itself. Once the combustion is initially started by the igniter, it is self-sustaining and the igniter is no longer used. In can-annular and annular combustors (see Types of combustors below), the flame can propagate from one combustion zone to another, so igniters are not needed at each one. In some systems ignition-assist techniques are used. One such method is oxygen injection, where oxygen is fed to the ignition area, helping the fuel easily combust. This is particularly useful in some aircraft applications where the engine may have to restart at high altitude.

Air flow paths



Primary air

This is the main combustion air. It is highly compressed air from the high-pressure compressor (often decelerated via the diffuser) that is fed through the main channels in the dome of the combustor and the first set of liner holes. This air is mixed with fuel, and then combusted.

Intermediate air

Intermediate air is the air injected into the combustion zone through the second set of liner holes (primary air goes through the first set). This air completes the reaction processes, cooling the air down and diluting the high concentrations of carbon monoxide (CO) and hydrogen (H_2).

Dilution air

Dilution air is airflow injected through holes in the liner at the end of the combustion chamber to help cool the air to before it reaches the turbine stages. The air is carefully used to produce the uniform temperature profile desired in the combustor. However, as turbine blade technology improves, allowing them to withstand higher temperatures, dilution air is used less, allowing the use of more combustion air.

Cooling air

Cooling air is airflow that is injected through small holes in the liner to generate a layer (film) of cool air to protect the liner from the combustion temperatures. The implementation of cooling air has to be carefully designed so it does not directly interact with the combustion air and process. In some cases, as much as 50% of the inlet air is used as cooling air. There are several different methods of injecting this cooling air, and the method can influence the temperature profile that the liner is exposed to (see Liner, above).

INTRODUCTION

Study of combustion process in all combustion systems is one of the most important and complex problems. Generally, the main objective is to achieve a stable combustion proves that appears in industrial furnaces, gas turbine combustors and boiler furnaces. The conditions of reaching stable flame differ according to the type offlame.

This part introduces the basic definitions of flame types and their classifications. The stability concepts and stability methods are indicated. Among these methods are two opposed flame stability means, bluff body and sudden expansion. All these methods are discussed in detail in the following section.

1.1 FlameDefinition

The flame is self-sustaining propagation of a localized combustion zone at subsonic velocity. Flame can be classified based upon how the fuel and oxidizer reach the reaction zone; these are non-premixed flames and premixed flames. Also, the flame can be classified according to the flow characteristics of incoming reactants into turbulent and laminar flames. A detailed description of these two categories will bediscussed.

a) Non-premixedflames

In this type, the fuel and oxidizer are coming in either side of the reaction zone and the products go out of the reaction zone. In such flames the reaction zone is established at a location where the total enthalpy of thereactants present balances the total enthalpy of the products generated plus any energy losses. Thus, for non-premixed flames the reaction ideally takes place at stoichiometric conditions thereby producing the maximum possible flame temperature for a given combination of the reactantspecies.

b) Premixedflames

The flames in this type can be classified into; fully premixed flames and partially premixed flames as described below:

i) Fully premixed flames

In this type the fuel and the oxidizer are thoroughly mixed prior to reaching the reaction zone, also known as flame front. In these flames the position of the reaction zone is not defined by the diffusion of reactants, but occurs due to balancing of the local convective velocity of the reactants with the rate of consumption of the reactants popularly known as the flame speed. Based on the stabilizing method, fully premixed flames can be burned at equivalence ratios other than 1. Thus, lower flame temperatures can be achieved in this type offlames.

ii) Partially premixedflames

Here the fuel is injected into the oxidizer flow just upstream of the flame or part of combustion air is added to the fuel. Under such conditions, there is not enough time for the fuel and oxidizer to mix thoroughly and thus, concentration gradients across the flow are generated in the reactants stream that enter the flamefront.

1.2 Flame Stabilization

Flame stabilization is of fundamental importance in the design, the efficient performance and the reliable operation of high-speedpropulsionsystems. In gas turbines and other combustion equipment, the velocities at which the gases flow are much higher than the maximum flame speeds and the burner should be insensitive to large excess air as the power output is regulated by the fuel mass flowrate.
It is found that, the flow velocity and burning velocity are the most important factors that the flame stabilization depends on. The burning velocity should be equal to the flow velocity for a stationary flame front. In house and industrial applications flame stability is achieved by attaching the flame to a simple device known as a burner.

Flame stabilization is usually accomplished by causing some of the combustion products to recirculate and hence to continually ignite the fuel mixture. The hot recirculating gases transfer heat to the colder ones ignites those and initiates flame spread. The burnt gases transfer heat to the recirculation zone to balance the heat lost in igniting the combustible gas. Sufficient energy must be fed to the stabilization region to continuously ignite the coming gasflow.

1.3 Parameters Influencing FlameStability

There are many factors that affecting on the stabilization of flames are described here, these include:

a) Blockage effect

If flame holder is located in a duct, which is the normal case, then an additional parameter controlling its stability characteristics is known as the blockage ratio, (BR). This is defined as the ratio of the area of projected flame holder to the cross sectional area of the duct. All stability theorems show that stability limit is widened as the characteristic dimension of the flame holder increases.

b) Flame holder size

An increase in flame holder size improves stability by extending the residence time of reaction in the recirculation zone.

c) Flame holder shape

The shape of flame holder affects its stability characteristics, which influences the size and shape of the wake region.

d) Fueltype

The fuel type has an effect on stability limit as for kerosene type fuel, it's found that combustion can be sustained at leaner mixture strengths with fuels of lower specific gravity. The paraffin fuels will operate at lower fuel air ratio than aromatic fuels.

The stability is further improved with:

1. The increase in fuelvolatility.
2. Finer atomization and reduction of the mean drop size offuel.

e) Streamvelocity

Any increase in stream velocity invariably has an adverse effect on flame stability. Any increase in velocity reduces the range of mixtures strengths over which combustion is possible and increases the weak extinction limit.

f) Pressure

The increase in the reactants pressure always improves flame stability. The studies performed, by several investigators on bluff body flame holders in can-type, burners, and stirred reactors have fully confirmed the beneficial effect of increased pressure in extending the range of stable operating conditions. For the gases mixture, the increase in the pressure expands the stability loop by enhancing the blowout velocity, especially for rich and near stoichiometric mixture.

1.4 Flame Stabilization Methods

Free jets flames are hardly stable, so some mechanisms are needed to enhance the flammability limits of the flame such as using a bluff body, a swirler, a pilot flame, a counter flow technique, by fixing putting flame holder in the combustible mixture flow.

1.4.1 Flame stabilization by using a bluffbody

Bluff bodies are used to stabilize flames in high velocity flow in variety of propulsion and industrial combustion systems. These can be employed for supplementary firing in industrial boilers and heat recovery steam generators, and also used in ramjet and turbojet after burner system. In addition, these flame holders are used to study the fundamentals of turbulent flame characteristics or as computational test cases, and have been targeted as one of the three stationary laboratories premixed flame configurations forstudy.

The usual shapes of the bluff bodies used are cylindrical rods, rectangular discs, baffles, cones or “vee” gutters, as given in Fig. (1.1) which produces in their wake a low velocity recirculated flow in which combustion can be initiated and maintained. The propagation of flame to other regions is rendered possible by the transport of heat and radicals from the boundaries of the re-circulation zone to the adjacent freshmixture.

Most of the understanding concepts of the flame stabilization process isreferred to the pioneering studies carried out; these studies found that thewake behind the bluff body can be divided into the recirculation zone and the mixing zone that keeps the recirculation zone away from the unburned reactants as shown in Fig. (1.2). The mixing zone, characterized by turbulent shear layers with large temperature gradients and rigorous chemical reaction, is fed by turbulent mixing processes with cool combustible gas coming from the approaching stream (based on the mixing model indicated by Williams [26]).

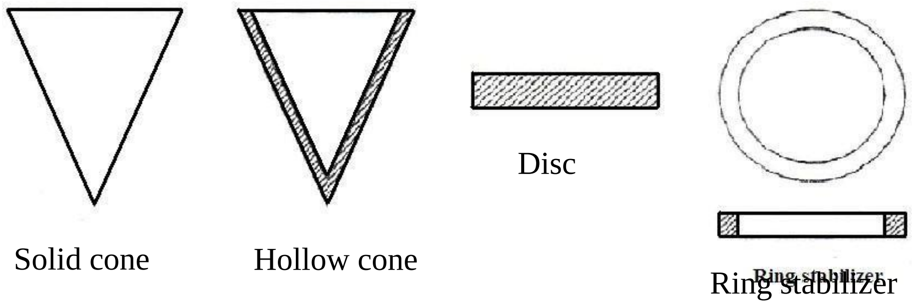


Fig. (1.1) Different bluff body shapes.

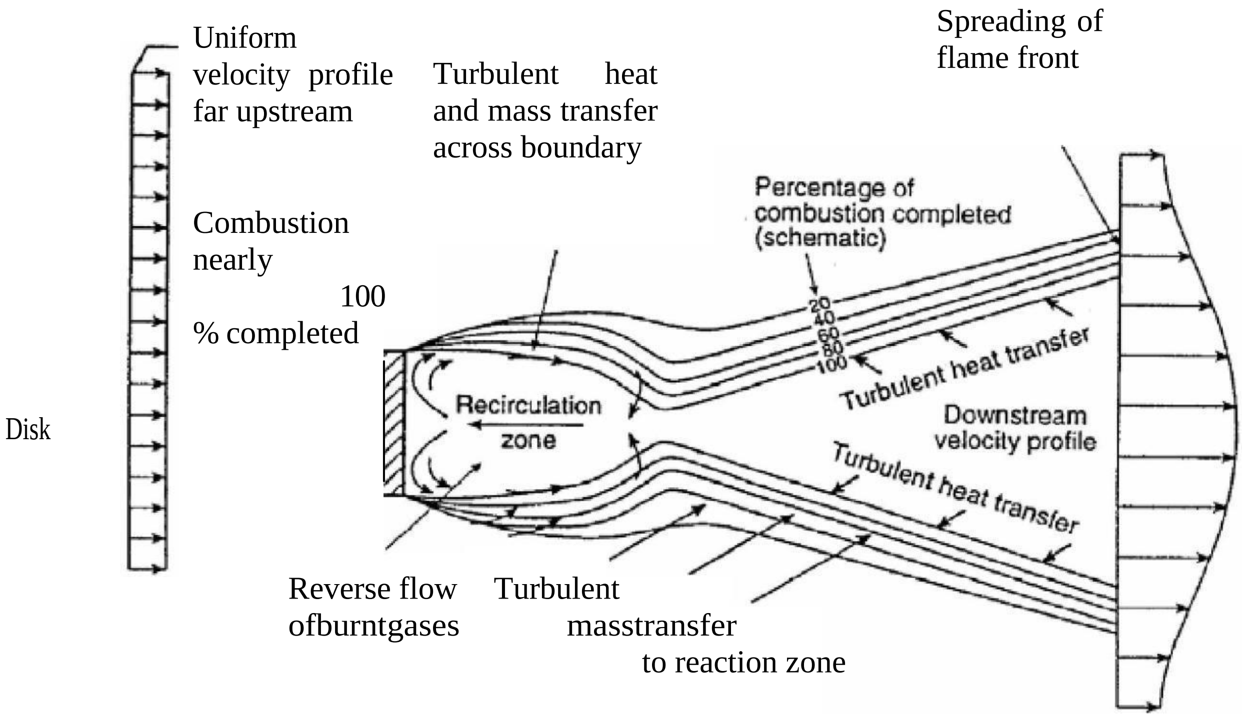


Fig. (1.2) Flame stabilization zone behind the bluff body (Williams 1966)

1.4.2 Flame stabilization using aswirler

A swirler is a number of curved swirl vanes have different angles normally 30, 45 and 60 degrees, which promotes the formation of recirculation zone and this is the essential mechanism for flame stabilization. Swirlnig flow can be produced either by tangential jet injections or by vane swirlers. The swirl angle determines the size and the strength of the recirculation zone and most of flame properties. Figure (1.3) indicates a schematic diagram of a simple swirler that used in burner.

1.4.3 Flame stabilization using a pilotflame

A pilot flame is an annular premixed flame located around the main jet flame used as a heat source to stabilize the main jet to the burner rim; the pilot flame is usually required to stabilize a jet flame due to its high exit velocity. A relative large pilot flame is used to produce a very lean jet flame. Figure (1.4) shows a photograph of pilot flame burner.

1.4.4 Flame stabilization using a counter flow stabilizingtechnique

A counter flow technique is to make the direction of the fuel jet in an opposite direction to the air flow, this technique reduces the flowing velocity of the combustible mixture against the burning velocity, also insures good mixing between the air and the fuel. Figure (1.5) shows a schematic diagram of a counter flow stabilized burner.

1.4.5 Flame stabilization using a transverseflow

From the recent and new techniques to form flame stability is that takes place by transversing the flow stabilizing technique, flame stability by pulsed high voltage discharge, flame stability by magnetic source at discharge.

UNIT-V
PREMIXED FLAMES

Formation of Flame

If the combustible substances produce vapour during burning process, a flame is produced. Flame is a luminous zone of the rapid exothermic reaction in combustion of vapour withthe formation of light and heat energy. A non-luminous region is appeared just after the flame where the temperature is slightly reduced. A flame is bounded between the ignition zone and a non-luminous gaseous zone. The combustion of gaseous fuels in a flame need the intimate contact of fuels with an oxidant, either oxygen or air prior to the reaction. The ranges of flammability and the point at which the mixture spontaneously ignites must be known. They must be heated to the combustion temperature and the flame produced will be at a high temperature. Then the reaction takes place in within a narrow zone or region in the flame. This combustion zone is called the flame front with this mixture is often several thousanddegrees.

Types of Flames

Flamesmaybeofdifferenttypesdependingontheextentofmixingoffuelandoxidizeror how the mixture reaches the reaction zone. The flow patterns in the reaction vessel, such as wellmixedandplugflowsarethemajortoolstoclassifytheflamesindifferenttypes.The flame may be turbulent and laminar types depending on the flow behavior of the combustiongases.

Inapremixedflamethefuelandtheoxidantaremolecularlymixedbeforethecombustion process takesplace.

The flames are mainly classified as:

i)Non-PremixedorDiffusionFlamesandii)Premixedflames.Theseflamesmaybeboth laminar and turbulenttypes.

Non-Premixed Flames

In many combustion processes, the fuel and air are often initially not mixed. The fuel and oxidizer are kept on either side of the reaction zone and moved to the reaction zone. The resultantflameistermedasthediffusionornon-premixedflame.Insomecases,thegasand air are injected in coaxial parallel tubes and ignited. The flow behavior of the non-premixedflameislaminartype.Molecularorturbulentdiffusionisresponsibleforthemixing of the gases in non-premixedflames.

In a laminar flow region, the reach the reaction front by diffusion, before the reaction takes place.Theproductsofcombustionalsodiffusetocomeoutfromthereactionzone.Theflame isalsocalledLaminarNon-PremixedFlamesorDiffusionFlame.Thediffusionflameoccurs at the interface of the gaseous fuel and air. With the progress of time as the flame propagates the thickness of the reaction zone increases. In diffusion flame the combustion rate will solely controlled by the diffusion rate, not by the kinetic rate of reaction. The mechanism of this flame is very complicated. The fuel approaches gradually towards the flame zone, there is a deficiency of oxygen and it is pyrolysed to smaller molecules or radicals. Then, there will be the formation of carbon soot and fuel burns with a bright luminous yellowflame.

As the flame propagates, the oxygen concentration in the zone increases and the product of pyrolyzedproductsarefurtherreacted.Graduallythereactionoccursunderthestoichiometric proportion of oxygen and combustible. Under these circumstances, the total enthalpy of the reactants will be equal to the total enthalpy of the products and the energy losses. Thenon-premixed flames at stoichiometric conditions optimize the flame temperature with a definite fuel-air ratio.

Premixed Flames

Inthistype,thefuelandoxidantgasaremixedtogetheratambientconditionbeforebeing delivered to the flame zone. As the mixture approaches the flame front, it is heated by conductionandradiation.Areactiontakesplacebeforereachingtheflamefront.Gradually themixtureissufficientlyheatedatthereactionfrontandthechemicalreactiontakesplace.

If the fuel and the oxidant gas are thoroughly mixed prior to reaching the flame front, the locationoftheflamefrontdoesnotdependonthediffusionofreactants.Theflamespeedor the velocity of the reactants in the reaction zone is important. In another case, the fuel is injected into the oxidant gas flow in the upstream side of the flame front but the mixing is improper.

Theturbulentpre-mixedflameplaysanimportantroleinthevariouspracticalapplications because;itincreasestheignitionoffuelwiththereductionintheemissionofgases.Boththe laminarpremixedorlaminarnon-premixedflames(diffusionflame)areslowprocessesand noteconomic.Turbulenceactuallyresultsinareductioninflamelength.Theturbulencecan be increased by recirculation of the fuel-airmixture.

Flame Structure

The example of the laminar premixed flame is the flame in a Bunsen burner as shown in Fig.1. In the premixed type, the laminar flame is the simplest type. The structure of the flame may be analyzed by a flame in the burner. The flame consists of four distinct regions as shown in Fig 1.Zonecontainingunburntgases,2.Reactionzone,3.Incompletecombustionzoneand4.Completecombustionzone.Theidealizedshapeofthereactionzoneofalaminarpremixed flame is a cone. The height of the cone represents the flame length, and depends on the velocity at the burneroutlet.

Flame Propagation

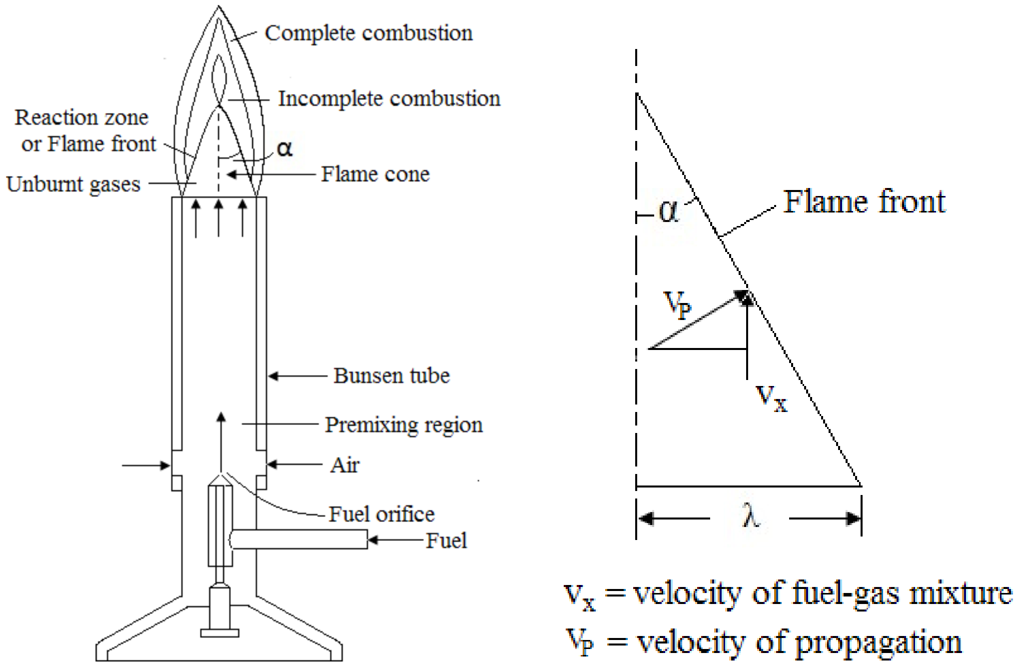


Fig.1 Laminar premixed flame in a Bunsen burner.

Since the flame front is stationary, the velocity of flame propagation v_p, with respect to the unburntmixturewillbeequaltotheflowvelocityoftheunburntmixturenormaltotheflame front. This is also called burningvelocity.

Velocity of flame propagation,$v_p = v_x$ Sin α
Where, α = flame cone angle and v_x is the velocity air fuel mixture.

Velocityofflamepropagationdependsonthepropertiesofthefuel-airmixture,pressureand temperature of the process. A premixed flame of a particular fuel-air combination is characterized by three main parameters, the burning velocity, flame temperature and flammabilitylimit,whicharealsodeterminedbythepressure,temperatureand,air-flueratio. Dilution of the burning mixture with an inert gas, such as helium or nitrogen, lowers the temperature and, consequently, the reaction rate. Higher amounts of inert gas extinguish the flame.

The Propagation Velocity in a Tube

Thepropagatingflamealongavortexaxisiscompletelydifferentfromthe‘normal’flames, whichpropagateinatubeorexpandsphericallyfromanignitionpoint.Itspropagatingspeed is nearly equal to the maximum tangential velocity in thevortex.

Forafueloxidizermixtureenclosedincylindricaltubewhoseoneendisclosedwhileother endisopened,letusconsidertheflamefrontisperpendiculartothedirectionofflow.

Forthepremixedgasesinalaminarflow,theflamespeed$V_f$isobtainedfromtheburning velocityV_b.Burningvelocityisequaltonormalvelocityofflamepropagation and gas velocity.

Flammability limit

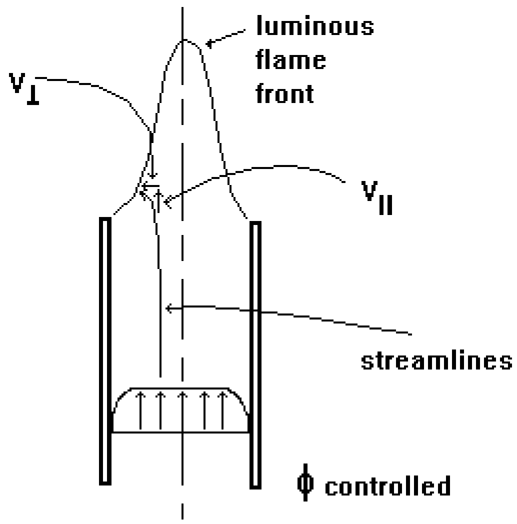
A spontaneous or explosive reaction of fuel may be possible when ignited in presence of either air or oxygen for a particular range of gas composition at fixed temperature and pressure. Flammability limits describe the range of fuel concentration in terms of volume % and this reaction may occur even when the mixture is cold. For example, the lower flammabilitylimitforaviationkeroseneisabout0.7%byvolumeandthehigherflammability limit is about 5% by volume. For hydrogen gas, lower limit 4%, and higher is 75%. Flammability limits also are much wider in oxygen than in air. So it highly depends on the burningatmosphere,pressure,temperature,andontheignitionsource.

Flame Propagation - Laminar Premixed Flames

Flames can be classified to organize their study:

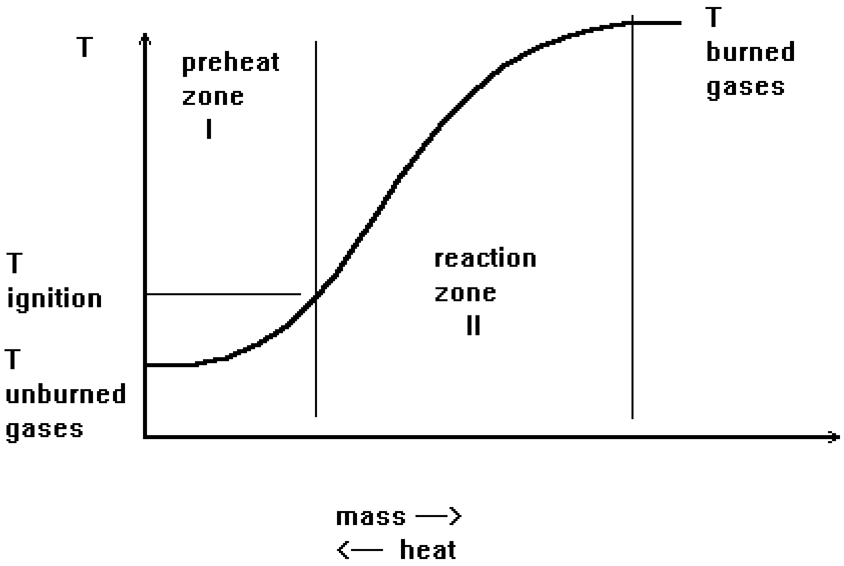
1. premixed or diffusion
2. laminar or turbulent
3. homogeneous or heterogeneous
4. stationary or traveling
5. deflagration or detonation
6. luminous or non-luminous

6.1 Laminar Pre-mixed Flames



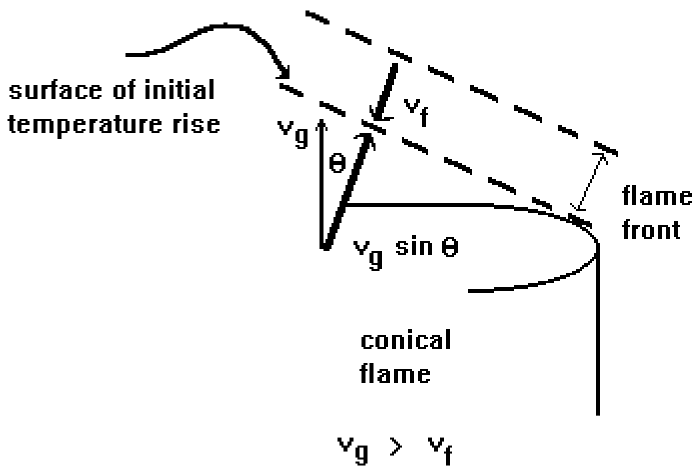
pre-mixed flame

Flame propagation refers to the propagation of the reaction zone or “combustion wave” through a combustible mixture. When the transport of heat and active species (free radicals) have initiated chemical reaction in the adjacent layer of the combustible mixture, the layer itself becomes the source of heat and radicals and is then capable of initiating reaction in the next layer. A quantitative theory of flame propagation must be based on the transfer of heat and mass from the reaction zone to the unburned mixture.



The enthalpy rise across the flame due to combustion is balanced by conduction from the reaction zone.

The flame propagation speed in a pre-mixed fuel/air mixture can be visualized:



To avoid flame blow-off, $v_f = v_g \sin\theta$. The flame velocity, v_f, may be defined as the velocity component of the cold unburned gas normal to the one dimensional flame front. In flames stabilized on a burner tube (Bunsen burner) the flame front is inclined at an angle against the gas flow and it is possible therefore to stabilize a flame (obtain a stationary flame) at gas flow rates higher than the rate of flame propagation. This is the reason why Bunsen burners can maintain flames over a range of flow rates and fuel oxidant mixture ratios - a flat flame burner is stable only for the flow rate of the gas that exactly matches the flame velocity.

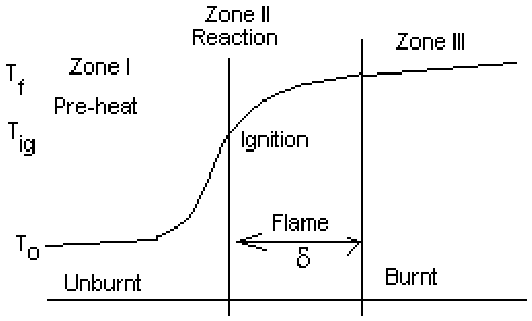
Flame velocity depends on:

- fuel type
- fuel-oxidant mass ratio (equivalence ratio)
- initial temperature of combustible mixture

- pressure
- flow pattern
- geometry of system

There are several theories that attempt to predict pre-mixed laminar flame propagation speed:

1. Thermal theories: energy transfer to an unburned mixture is the controlling mechanism in flame propagation



2. Diffusion theories: Mass transfer (particularly of chain carriers) is the controlling mechanism in flame propagation
3. Comprehensive theories: combination of the two above models

Thermal theories

Mallard and Le Chatlelier (1885)

Assumption: the energy conducted from zone II equals that necessary to bring unburned gases to ignition temperature

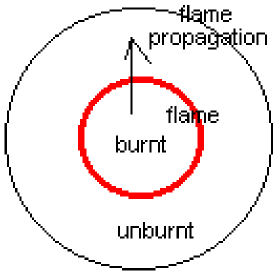
$$\dot{m}C_p\left(T_{ig}-T_o\right)=Ak_{cond}\left(T_f-T_{ig}\right)/\delta$$

$\dot{m}$	Mass flow rate into the combustion zone
k_{cond}	Thermal conductivity of burning gases
δ	Flame thickness

For a one-dimensional flame:

$$\dot{m}=\rho uA=\rho S_L A$$

S_L laminar burning velocity – flame front velocity (relative to a stationary reference point) through an unburned mixture



$$\rho S_L C_p (T_{ig} - T_o) = k_{cond} (T_f - T_{ig}) / \delta$$

$$S_L = \frac{k_{cond}}{\rho C_p} \frac{(T_f - T_{ig})}{(T_{ig} - T_o)} \frac{1}{\delta}$$

$$\delta = S_L \tau = S_L \frac{1}{dx/dt}$$

τ Characteristic reaction time

d[fuel]/dt reaction rate in terms of fractional conversion (x = [fuel]/$[\text{fuel}]_o$)

$$S_L = \frac{k_{cond}}{\rho C_p} \frac{(T_f - T_{ig})}{(T_{ig} - T_o)} \frac{1}{S_L} \frac{dx}{dt}$$

$$S_L{}^2 = \frac{k_{cond}}{\rho C_p} \frac{(T_f - T_{ig})}{(T_{ig} - T_o)} \frac{dx}{dt}$$

$$S_L = \sqrt{\frac{k_{cond}}{\rho C_p} \frac{(T_f - T_{ig})}{(T_{ig} - T_o)} \frac{dx}{dt}}$$

Effect of pressure:

$$S_L \propto \sqrt{\alpha \cdot rate} \propto \sqrt{\frac{\rho^{n-1}}{\rho}} \propto p^{n/2-1}$$

Effect of temperature:

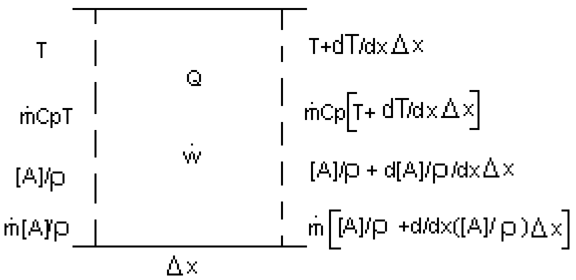
$$S_L \propto \bar{T}^{0.375} T_o T_b^{n/2} \exp(-E_A / 2 T_o T_b \, P^{(n-2)/2})$$

$$\bar{T} = \frac{T_f + T_o}{2}$$

n = order of the reaction

Zeldovich, Frank-Kamentskii& Semenov thermal theory (1938):

Diffusion of molecules (but not free radicals) as well as heat is included.



[A]	Molar concentration of reactant
-Dρ[d[A]/ρ]/dx	diffusive molar flux
$\dot{m}$	Mass flow rate per unit area (mass flux)
$\dot{m}$ [A]/ρ	convected mass
$\dot{m} C_p T$	convected heat
Q	heat of reaction
T	temperature
$\dot{w}$	Species production or reaction rate

The molar flux (mol/cm^2 sec) convected into the control volume through a flow cross section of unity:

$$\dot{m}\left\{\frac{[A]}{\rho}+\frac{d\,[A]/\rho}{dx}\Delta x\right\}-\dot{m}\frac{[A]}{\rho}=\dot{m}\frac{d\,[A]/\rho}{dx}\Delta x$$

The molar flux (mol/cm^2 sec) diffused into the control volume through a flow cross section of unity:

$$\frac{d}{dx}\left\{-D\rho\left[\frac{[A]}{\rho}+\frac{d\,[A]/\rho}{dx}\Delta x\right]\right\}-\left\{-D\rho\frac{d\,[A]/\rho}{dx}\right\}=-D\rho\frac{d^2\,[A]/\rho}{dx^2}\Delta x$$

The molar flux (mol/cm^2 sec) reacting in the control volume:

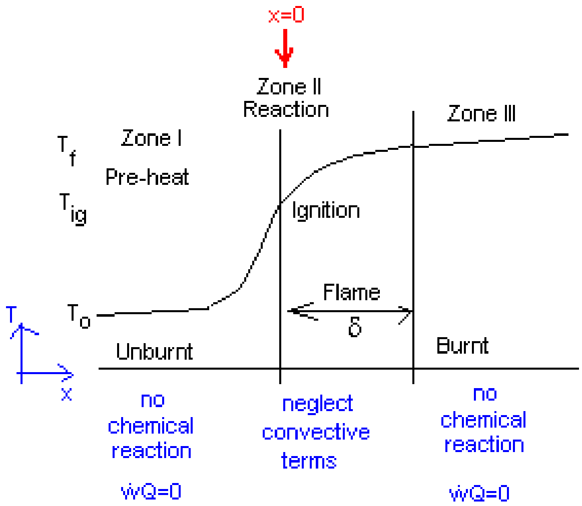
$$-\dot{w}\Delta x$$

Conservation of mass ==> diffusion + convection + sink = 0

$$D\rho\frac{d^2\,[A]/\rho}{dx^2}-\dot{m}\frac{d\,[A]/\rho}{dx}+\dot{w}=0$$

Conservation of energy:

$$k\frac{d^2T}{dx^2}-\dot{m}C_p\frac{dT}{dx}+\dot{w}Q=0$$



I. pre-heat zone

$$\frac{d^2T}{dx^2}-\frac{\dot{m}C_p}{k}\frac{dT}{dx}=0\qquad\begin{cases}T(x\to-\infty)=T_o\\ T(x=0)=T_{ig}\end{cases}$$

II. reaction zone

$$\frac{d^2T}{dx^2}+\frac{\dot{w}Q}{k}=0\qquad\begin{cases}T(x\to\infty)=T_f\\ T(x=0)=T_{ig}\end{cases}$$

Continuity of heat conduction: $k\left[\frac{dT}{dx}\right]_{\substack{x=0\\I}} = k\left[\frac{dT}{dx}\right]_{\substack{x=0\\II}}$

Multiple both sides of the reaction zone II energy equation by 2 dT/dx:

$$2\frac{dT}{dx}\frac{d^2T}{dx^2} = -2\frac{\dot{w}Q}{k}\frac{dT}{dx}$$

Use the product rule of differentiation to write:

$$\frac{d}{dx}\left(\frac{dT}{dx}\right)^2 = -2\frac{\dot{w}Q}{k}\frac{dT}{dx}$$

Integrate from T_{ig} to T_f:

$$\left(\frac{dT}{dx}\right)^2_{x\to\infty} - \left(\frac{dT}{dx}\right)^2_{x=o} = -2\frac{Q}{k}\int_{T_{ig}}^{T_f}\dot{w}dT$$

$$\left(\frac{dT}{dx}\right)^2_{x=o} = 2\frac{Q}{k}\int_{T_{ig}}^{T_f}\dot{w}dT$$

Now, work on the energy equation for the preheat zone I:

$$\frac{d}{dx}\left(\frac{dT}{dx}\right) - \frac{\dot{m}C_p}{k}\frac{dT}{dx} = 0 \qquad \begin{cases} T(x\to-\infty) = T_o \\ T(x=0) = T_{ig} \end{cases}$$

Integrate once:

$$\frac{dT}{dx} = \frac{\dot{m}C_p}{k}T + C_1$$

$$x\to-\infty, \quad T\to T_o \quad \Rightarrow \quad \frac{dT}{dx} = 0 \quad \Rightarrow \quad C_1 = -\frac{\dot{m}C_p}{k}T_o$$

$$\frac{dT}{dx} = \frac{\dot{m}C_p}{k}\{T - T_o\}$$

At x=0, the boundary between zones I and II, we know that the heat flux must be the same. Hence,

$$\left.\frac{dT}{dx}\right|_{x=0,I} = \left.\frac{dT}{dx}\right|_{x=0,II} = \frac{\dot{m}C_p}{k}\{T_{ig} - T_o\} = \sqrt{\frac{2Q}{k}\int_{T_{ig}}^{T_f}\dot{w}dT}$$

Substitute in the expression for the flame velocity $\dot{m} = \rho_o S_L$ where the area A=1 cm^2, and solve for S_L.

$$S_L = \frac{\sqrt{2kQ\int_{T_{ig}}^{T_f}\dot{w}dT}}{\rho_o C_p\{T_{ig} - T_o\}}$$

In this model, you also observe the square root dependence of the laminar flame velocity on the rate of fuel consumption. In either model, the global reaction rate models for fuel consumption (from Topic 5 Kinetics) can be used here.

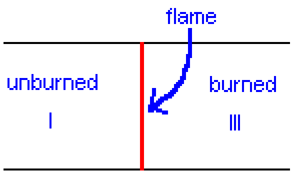
Diffusion Theories

Lewis and Von Elbe, 1934, developed for ozone reactionTanford and Pease, 1947, diffusion of radicals is more important than temperature gradients

Comprehensive Theories

These flames spread models combine thermal and diffusion principles and must be solved numerically.

In addition to heat and mass transfer across the flame, momentum must also be conserved. We can use these laws to **estimate the pressure drop** across a flame.



Conservation of mass: $\rho_1 S_L = \rho_2 u_2$

Conservation of momentum:

$$p_1 + \rho_1 S_L{}^2 = p_2 + \rho_2 u_2{}^2$$

$$p_u - p_b = p_1 - p_2 = \rho_2 u_2{}^2 - \rho_1 S_L{}^2 = \rho_2 u_2 u_2 - \rho_1 S_L{}^2$$

$$p_1 - p_2 = \rho_1 S_L u_2 - \rho_1 S_L{}^2 = \rho_1 S_L (u_2 - S_L) = \rho_1 S_L{}^2 \left(\frac{u_2}{S_L} - 1 \right)$$

$$p_1 - p_2 = \rho_1 S_L{}^2 \left(\frac{\rho_1}{\rho_2} - 1 \right)$$

For a gaseous fuel, $\rho_1 = \frac{p}{RT} = \frac{101 kPa\, kg\, K}{0.287 kJ\, 300 K} \frac{kN}{kPa\, m^2} \frac{kJ}{kN \cdot m} = 0.1 \frac{kg}{m^3}$

A typical laminar flame speed: $S_L = 40 cm/\sec$

A typical density ratio between burned and unburned gases: $\frac{\rho_1}{\rho_2} = 7$

$$p_1 - p_2 = \rho_1 S_L{}^2 \left(\frac{\rho_1}{\rho_2} - 1 \right) \cong 0.1 \frac{kg}{m^3} 0.16 \frac{m^2}{s^2} 6 \frac{kN \cdot s^2}{kg \cdot m} \frac{kPa \cdot m^2}{kN} = 0.10 kPa$$

One-tenth of a kPa pressure drop is very small.

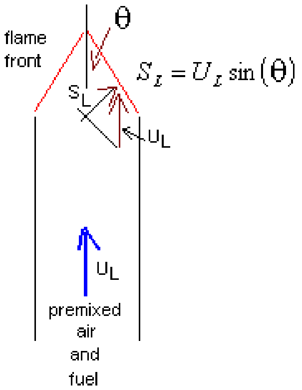
The laminar burning velocity S_L is a property of a fuel/air mixture. It varies with the fuel and its temperature, pressure and equivalence ratio: $S_L = S_L(fuel, T, p, \phi)$

Successfully modeling flame speed is compounded because of the uncertainty of available data. For example, the thermal properties of radicals needed in some models are not known. Also, there is still substantial uncertainty surrounding reaction rate constants. Consequently, if you really need to know flame speed, experiments are required.

Experimental Methods for Measuring Flame Speed

1. Bunsen burner - pre-mixed

Laminar burning velocity is proportional to the laminar gas velocity: $S_L = U_L \sin(\theta)$



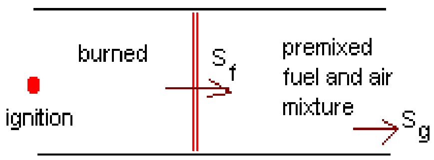
One aspect of flame stability can be studied with the Bunsen burner:

If the gas velocity > burning velocity ==> blow off

If the burning velocity > gas velocity ==> flash back

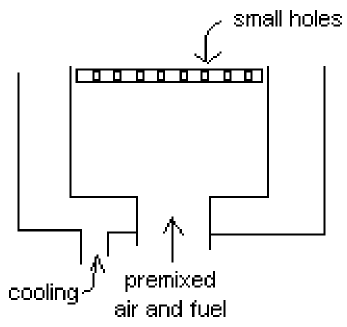
The burning velocity measured with a Bunsen burner is not the true adiabatic burning velocity because of heat transfer from the flame to the environment.

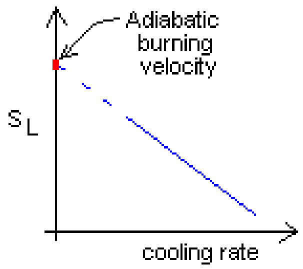
2. cylindrical tube



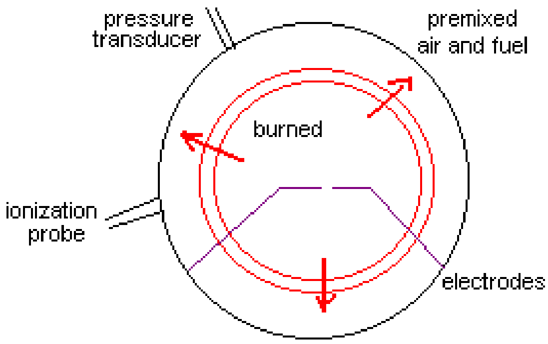
$$S_L = S_f - S_g$$

3. flat flame burner





4. closed spherical bomb



Analyzing data from the bomb requires corrections for unsteady heat conduction and varying temperature at the wall. The ionization probe is an open circuit that flame radicals will close. This senses the flame arrival time. The probe can be moved to different locations to see if any buoyancy effects are present.

Flame velocity of gas-air mixtures at 20 C (Gunther, 1974) (Vol % of fuel)

fuel	v_{fmax} **cm/s**	**[fuel]** $_{max}$ **vol %**	v_{fst} **cm/s**	**[fuel]**$_{st}$ **vol %**
methane	43.0	10.17	42.0	9.5
ethane	48.7	5.99	47.6	5.64
propane	47.2	4.27	46.0	4.07
butane	45.2	3.38	43.4	3.13
n-pentane	55.0	2.64	54.8	2.56
ethylene	78.0	7.0	77.8	6.55
propylene	54.7	4.64	54.4	4.51
1-butylene	53.3	3.48	53.0	3.38
acetylene	168.0	9.3	155.0	7.75
hydrogen	364.0	42.5	237.0	29.58
CO	19.5	41.5	17.4	29.58
0.9 CH_4 **+ 0.1** N_2	41.0	12.6	39.0	11.7
0.8 CH_4 **+ 0.2** N_2	40.2	14.0	39.0	13.1
gas 15.6 MJ/m^3	103.5	25.2	86.0	21.4
gas 17.3 MJ/m^3	91.0	23.1	81.0	20.1

Note that the flame velocities in the above table are higher when the mixture is richer than stoichiometric. The maximum flame temperature occurs in stoichiometric mixtures. But when the mixture is slightly fuel rich, the concentrations of free radicals - which play a significant role in flame propagation - are at a peak. (There is enough fuel present to consume all the oxygen, which results in peak power if the combustion occurs in a SI engine.)

Two types of **flame stability limits** exist:

1. the ability of the mixture to support flame propagation; this defines the flammability limits of the fuel
2. flow conditions that lead to blow off, or to flash back (mentioned above)

The **lower flammability limit** (also called the weak or lean limit) is the minimum percentage of fuel in the fuel/air mixture that can be flammable, or the minimum Φ.

The **upper flammability limit** (also called the rich limit) is the maximum percentage of fuel in a fuel/air mixture that can be flammable, or the maximum Φ.

Within the flammability limits between fuel-rich and fuel-lean conditions, flame propagation is possible once ignition occurs. Safe installations to prevent explosion hazards use this data.

As temperature and pressure increase, flammability limits generally widen. The fuel rich limit may increase significantly at higher temperatures due to "cool flames." Cool flames ignite at low temperatures, 300 - 400 C, and propagate best at high fuel equivalence ratios. Their presence may cause violent explosions within the normal flammability limits.

Effect of pressure on flammability limits (O_2 partial pressure in %)

fuel in air	**10 atm**	**50 atm**	**125 atm**
methane	6.0 - 17.1	5.4 - 29.0	5.7 - 51.6
hydrogen	10.2 - 68.5	10.0 - 73.3	9.9 - 74.8
CO	17.8 - 62.8	20.6 - 56.8	20.7 - 51.6

When inert gas is added to the flammable mixture, the flammability limits narrow. The narrowing happens mainly in the lowering of the rich limit. The lean limit remains mostly unaffected. Near the lean limit, oxygen is diluents just as nitrogen is. At the rich limit, it is fuel that is largely replaced by the diluents, as the fuel is present in excess in the mixture. Hence, the lowering of the rich flammability limit is due to a different mode of expressing the limits when diluents are added. The effectiveness of the diluents increases according to its molar heat capacity (CO_2> H_2O > N_2> He >Ar).

Maximum safe percentage of oxygen in combustible gas-air mixtures with carbon dioxide or nitrogen added

fuel	**CO_2 as diluent**	**N_2 as diluent**
methane	14.6	12.1
ethane	13.4	11.0
propane	14.3	11.4
ethylene	11.7	10.0
propylene	14.1	11.5
hydrogen	5.9	5.0
CO	5.9	5.6

The quenching thickness or quenching distance is another measurable property of a fuel. The quenching distance is the smallest diameter or minimal channel width in which a flame can propagate. Walls of the channel act as sinks for heat and free radicals and therefore quench the flame at their surface.

The quenching distance is used in the design of flame arrestors between the fuel tank and burner of outdoor grills, camping stoves, hydrogen tanks, etc. Flame quenching at non-combustible surfaces within IC engines accounts for the unburned HCs spewed out of the tailpipe. We will discuss this form of pollution later.

Physical and combustion properties of selected fuels

fuel	**quenching distance cm**	**E_{igmin} MJ**	**T_{ig} C**
methane	0.039	3.3	610
ethane	0.0354	6.5	-
propane	0.031	3.05	470
butane	0.0312	-	-
n-pentane	0.051	8.2	285
ethylene	0.019	0.96	-
propylene	0.031	2.82	-
acetylene	0.0118	0.3	425
hydrogen	0.00984	0.2	-

The flash point of a fuel is the lowest temperature for ignition but not sustained burning.The fire point of a fuel is the lowest temperature for ignition and sustained burning.Ignition temperatures depend strongly on the apparatus in which they are measured. Standardized tests are now used to give comparative values. Ignition is kinetically controlled and there is a temperature limit below which the rate of reaction is so low that the heat of reaction is dissipated by heat losses and the mixture temperatures cannot rise.Ignition temperatures are lower for a slightly fuel-rich mixture.The minimum ignition energy for a combustible mixture is the minimum amount of energy that will ignite the mixture. It is determined experimentally with spark ignition.