CHAPTER 6

AEROTHERMODYNAMICS OF INLETS, COMBUSTORS, AND NOZZLES

6.1 INTRODUCTION

In Chapter 5 we used the laws of thermodynamics and fluid mechanics to explain the behavior of aircraft jet engines. We treated the several engine components as “black boxes,” in the sense that we confined discussion to the inlet and outlet conditions of the propellant, without regard to the internal mechanisms that produce its change of state. Where necessary, we related the actual performance to some easily calculated or "ideal" performance by the definition of an appropriate component efficiency or stagnation pressure ratio. The purpose of this and the following chapter is to examine the internal mechanisms of the various components in order to describe the factors that impose practical limits on performance. We consider conditions required for high performance of components and, in some cases, present methods for quantitative prediction of their behavior.

For the ramjet, Eq. (5.34) indicated that a given percentage loss in stagnation pressure has the same effect on engine performance wherever it occurs through the engine. For turbine engines the same conclusion holds, though it is not so easily seen, since component performances are usually stated in terms of adiabatic efficiencies rather than stagnation pressure ratios. Hence high performance is of equal importance for all engine components. The attainment of high performance is generally more difficult in those regions requiring a rise in static pressure than in those where the pressure falls. This, as we pointed out in Chapter 4, may be attributed to boundary layer behavior and the tendency for separation in the presence of a rising static pressure. Thus inlets (which generally have rising
pressure gradients even for turbojets) may be more difficult to design for efficient operation than nozzles. Similarly, as we will see in Chapter 7, compressors are more difficult to design (and have lower ultimate efficiency) than turbines.

In this chapter we are concerned with the major components of turbine engines other than the turbomachinery (which is the subject of Chapters 7, 8, and 9). The common theme is steady flow in stationary ducts, and the chief interest lies in the arrangements that are necessary for high efficiency in inlets, burners, and nozzles.

### 6.2 SUBSONIC INLETS

An engine installed in an aircraft must be provided with an air intake and a ducting system. Figure 5.27 shows the inlet duct for the Rolls-Royce RB211-535E4 engine mounted in an underwing installation. We will see in Chapter 8 that for turbojet engines the airflow entering the compressor or fan must have low Mach number, in the range 0.4 to 0.7, with the upper part of the range suitable only for transonic compressors or fans. If the engine is designed for subsonic cruise at, for example, \( M = 0.85 \), the inlet must be designed to act as a diffuser with reasonably gentle diffusion from \( M = 0.85 \) to perhaps \( M = 0.6 \). Part of this deceleration occurs upstream of the inlet entrance plane. One can see in Fig. 5.27 the relatively small increase in area associated with the internal deceleration.

The inlet must be designed to prevent boundary layer separation, even when the axis of the intake is not perfectly aligned with the streamline direction far upstream of the inlet. In other words, the performance of the inlet must not be unduly sensitive to pitch (up-and-down) and yaw (side-to-side) motions of the aircraft. It is important that the stagnation pressure loss in the inlet be small. It is even more important that the flow velocity and direction leaving the inlet be uniform, since distortions in the velocity profile at the compressor inlet can severely upset the compressor aerodynamics and may lead to failure of the blades due to vibrations. Design of inlets that must operate efficiently in both supersonic and subsonic flight poses special problems; we consider these in Section 6.3.

**Flow Patterns**

Depending on the flight speed and the mass flow demanded by the engine, the inlet may have to operate with a wide range of incident stream conditions. Figure 6.1 shows the streamline patterns for two typical subsonic conditions and the corresponding thermodynamic path of an “average” fluid particle. During level cruise the streamline pattern may include some deceleration of the entering fluid external to the inlet plane [Fig. 6.1(a)]. During low-speed high-thrust operation (e.g., during takeoff and climb), the same engine will demand more mass flow and the streamline pattern may resemble Fig. 6.1(b), which illustrates external acceleration of the stream near the inlet. In both cases there is an external change of state that is essentially isentropic, since there are no walls on which friction may act. For given air velocities at stations (a) and (2), external accelera-
tion raises the inlet velocity and lowers the inlet pressure, thereby increasing the internal pressure rise across the diffuser. If this pressure increase is too large, the diffuser may stall because of boundary layer separation; stalling usually reduces the stagnation pressure of the stream as a whole. Conversely, external deceleration requires less internal pressure rise and hence a less severe loading of the boundary layer. 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” $A_o$ is less than the inlet area $A_1$, and some flow is “spilled over” the inlet, accelerating as it passes over the outer surface. For high-Mach-number subsonic flight this acceleration (and subsequent deceleration) must not be too large or there would be danger of shock-induced boundary layer separation on the outer surface and excessive nacelle drag. For supersonic flight such “spilling” action would necessarily be accompanied by a shock system that reduces the relative velocity at inlet to subsonic values—so that the air may sense the presence of the inlet and flow around it.

**Internal Flow**

Qualitatively the flow in the inlet behaves as though it were in a “diffuser,” which is a common element in fluid machinery. A better term might be decelerator, since the device is not primarily concerned with molecular or turbulent diffusion, but we will retain here the traditional term diffuser and define it to mean any section of a duct in which fluid momentum decreases and pressure rises, no
work being done. Considerable experimental and analytical work has been done on cylindrical (especially conical [1] and annular [2]) diffusers, but little of this is directly applicable to subsonic aircraft inlets. The reason is that most work on diffusers focuses on the conditions that are related to maximum pressure recovery—which is usually associated with a highly nonuniform exit velocity profile and perhaps even with some flow unsteadiness. In typical subsonic aircraft inlets there is a stringent requirement that the flow velocity entering the compressor be steady and uniform. Consequently inlet design does not depend so much on the results of diffuser research as on potential flow calculations, coupled with boundary layer calculations and followed by wind tunnel testing to assess inlet performance under a wide range of test conditions.

In the actual engine inlet, separation can take place in any of the three zones shown in Fig. 6.2. Separation of the external flow in zone (1) may result from local high velocities and subsequent deceleration over the outer surface. We will discuss this possibility, which leads to high nacelle drag, subsequently.

Separation on the internal surfaces may take place in either zone (2) or zone (3), depending on the geometry of the duct and the operating conditions. Zone (3) may be the scene of quite large adverse pressure gradients, since the flow accelerates around the nose of the center body, then decelerates as the curvature decreases. In some installations it has not been possible to make the exit area of the intake more than about 30% greater than the inlet area without the incidence of stall and large losses. Reynolds number effects may also be important for large inlets and high-speed flow. At high angles of attack, all three zones could be subjected to unusual pressure gradients.

**External Flow**

As we already indicated, inlet design requires a compromise between external and internal deceleration. Both can lead to difficulties, and a balance is needed. To examine the effect of external deceleration on inlet design, methods are needed for calculating both potential flow (internal and external) and boundary layer growth on intake surfaces. References [3], [4], and [5] provide details on how such methods have been applied to subsonic engine intakes. These methods are not valid for separated flow, but they are able to warn the designer of the danger of separation, and so provide guidance on design modifications that may be needed to avoid separation. Expressed in computer codes, these methods can

FIGURE 6.2 Possible locations of boundary layer separation.
determine the pressure and velocity field at all points in the flow. They are able
to predict the momentum thickness (and the associated nacelle drag) due to the
boundary layer on the external surface of the nacelle. They are also able to define
the velocity distribution of the flow approaching the compressor.

We will not go into the details of these methods here but instead will focus on
a qualitatively valid and much simpler explanation of the fact that external decel-
eration must be limited to prevent excessive nacelle drag. Following the method
of Küchemann and Weber [6], we here ignore compressibility and suppose that
the only effect of the boundary layer is flow separation (if the deceleration pres-
sure coefficient becomes too large).

Figure 6.3 shows a typical streamline pattern for large external deceleration.
In flowing over the lip of the inlet, the external flow is accelerated to high veloc-
ity, much as the flow is accelerated over the suction surface of an airfoil. This
high velocity and the accompanying low pressure can adversely affect the bound-
ary layer flow in two ways: For entirely subsonic flow, the low-pressure region
must be followed by a region of rising pressure in which the boundary layer may
separate. Hence one might expect a limiting low pressure $p_{\text{min}}$ or, equivalently,
a maximum local velocity $u_{\text{max}}$, beyond which boundary layer separation can be
expected downstream. For higher flight velocities (or higher local accelerations),
partially supersonic flow can occur. Local supersonic regions usually end
abruptly in a shock, and the shock-wall intersection may cause boundary layer
separation. One might expect a limiting local Mach number that should not be
exceeded. Whatever the cause, boundary layer separation is to be avoided, since
it results in poor pressure recovery in the flow over the after portions of the air-
craft or engine housing. This, of course, results in a net rearward force or drag on
the body.

![Figure 6.3](image-url)  
*FIGURE 6.3* Control volume for the calculation of thrust on inlet surface.
To illustrate the major features of the external flow near the inlet, consider the simplified problem of an inlet on a semi-infinite body. Küchemann and Weber have shown how to relate the external flow over such a body to the extent of external deceleration of the flow entering the inlet. Suppose (see Fig. 6.3) that the external cross section of the inlet grows to a maximum area $A_{\text{max}}$ and that the body remains cylindrical from this point downstream. A control surface is indicated that extends far from the inlet on the sides and upstream end, crosses the inlet at its minimum area $A_i$, passes over the inlet surface, and extends downstream far enough for the external fluid velocity to return essentially to the upstream or flight velocity $u_a$ (neglecting boundary layer effects). Thus all the external flow enters and leaves the control volume with an axial velocity component $u_a$, if one assumes that the sides of the control volume are sufficiently removed from the inlet. The internal flow enters the control volume with velocity $u_i$ and leaves with velocity $u_i$ (assuming, for simplicity, one-dimensional flow in the inlet). The net momentum flux out of the control volume is then, ignoring changes in the air density,

$$\dot{m}_i u_a + \rho u_i^2 A_i - \rho u_a^2 A_{\text{max}}.$$ 

From continuity, the side flow rate is $\dot{m}_s = \rho u_a A_{\text{max}} - \rho u_i A_i$, so that the net momentum flux can be expressed as $\rho A_i (u_i^2 - u_i u_a)$. The net force in the axial direction on the control volume is

$$p_a A_{\text{max}} - p_i A_i - F_x,$$

where $F_x$ is the axial component of the force on the control volume due to the forces on the external surface of the inlet. If we neglect friction,

$$F_x = -\int_{\text{inlet}} px \cdot n \, dA = \int_{A_i}^{A_{\text{max}}} p \, dA_x,$$

where $p = \text{pressure on surface},$

$x = \text{unit vector along axis in the flow direction},$

$n = \text{outward (from the inlet surface) pointing unit vector},$

$dA = \text{increment of external surface area},$

$dA_x = \text{increment of external surface area normal to } x \ (2\pi r \, dr \text{ for an axisymmetric inlet}).$

Combining these expressions, the momentum equation requires

$$p_a A_{\text{max}} - p_i A_i - \int_{A_i}^{A_{\text{max}}} p \, dA_x = \rho A_i (u_i^2 - u_i u_a)$$

or

$$\int_{A_i}^{A_{\text{max}}} (p_a - p) \, dA_x = \rho A_i (u_i^2 - u_i u_a) + (p_i - p_a) A_i.$$  \hspace{1cm} (6.1)

\(^\dagger\text{An inlet followed by a cylindrical portion several diameters in length would behave similarly, but few practical engine housings are actually cylindrical over any appreciable length.}\)
The equation is arranged in this form because we can consider the integral a component of thrust \( \Delta \mathcal{T}_i \), which acts on the front external surface of the inlet due to reduction of local surface pressure. Applying Bernoulli’s equation to the external deceleration of internal flow, we obtain
\[
p_i - p_a = \rho \left( \frac{u_a^2 - u_i^2}{2} \right).
\]
Thus
\[
\int_{A_i}^{A_{max}} (p_a - p) \, dA_x = \rho A_i (u_i^2 - u_a u_a) + \rho A_i \left( \frac{u_a^2}{2} - \frac{u_i^2}{2} \right)
\]
or
\[
\frac{\Delta \mathcal{T}_i}{\frac{1}{2} \rho u_a^2 A_i} = \left( 1 - \frac{u_i}{u_a} \right)^2.
\]
(6.2)
This shows that the greater external deceleration (i.e., the smaller the ratio \( u_i/u_a \)), the larger must be the “thrust” increment:
\[
\Delta \mathcal{T}_i = \int_{A_i}^{A_{max}} (p_a - p) \, dA_x.
\]
On the outer surface of the nacelle, the pressure must rise from some minimum value \( p_{min} \) (at the point where the local free-stream velocity is \( u_{max} \)) to the ambient value \( p_a \) associated in this simplified case with straight parallel flow downstream. Here we neglect the boundary layer except to say that the pressure coefficient
\[
C_p = \frac{p_a - p_{min}}{\frac{1}{2} \rho \, u_{max}^2}
\]
must not be too large or the boundary layer will separate. If, still for the outer surface of the nacelle, we define an average pressure difference
\[
\frac{p_a - p}{A_{max} - A_i} = s(p_a - p_{min}),
\]
where \( s \) is a factor between 0 and 1, we can write
\[
\Delta \mathcal{T}_i = s(p_a - p_{min}) (A_{max} - A_i),
\]
so that Eq. (6.2) becomes
\[
\frac{s(p_a - p_{min})(A_{max} - A_i)}{\frac{1}{2} \rho u_a^2 A_i} = \left( 1 - \frac{u_i}{u_a} \right)^2.
\]
We can rewrite this as
\[
\frac{A_{max}}{A_i} = 1 + \frac{\left( 1 - \frac{u_i}{u_a} \right)^2}{s C_{p_{max}} \left( \frac{u_{max}}{u_a} \right)^2}.
\]
or

\[
\frac{A_{\text{max}}}{A_i} = 1 + \frac{\left(1 - \frac{u_i}{u_a}\right)^2 (1 - C_{p\text{max}})}{sC_{p\text{max}}}.
\] (6.3)

The value of \(s\) will depend on the shape of the nacelle. Taking \(s = 0.5\) for purposes of illustration, we can show, as in Fig. 6.4, the dependence of the size of the external surface necessary to prevent external boundary layer separation, for any given value of \(u_i/u_a\).

The main point here is that the larger the external deceleration (i.e., the smaller the value of \(u_i/u_a\)), the larger must be the size of the nacelle if one is to prevent excessive drag. Even in the absence of separation, the larger the nacelle, the larger the aerodynamic drag on it. But if the external deceleration is modest (e.g., \(u_i/u_a > 0.8\)), its effect on minimum nacelle size is quite small. The use of partial internal deceleration is, of course, doubly effective in reducing maximum diameter because it permits a reduction in both \(A_i\) and \(A_{\text{max}}/A_i\).

With an aircraft flight Mach number of 0.85, an engine with a transonic fan will require little deceleration of the incoming air, since the allowable absolute Mach number at entry to the fan may be as high as 0.6 or even higher. Still the inlet must be carefully designed for this case. With a flight Mach number of 0.85, the maximum velocity near the external surface could easily be supersonic, so that there is the possibility of shock-induced boundary layer separation. Full allowance for compressibility effects is therefore necessary for inlet design.

To summarize: This analysis pertains to a simplified picture of the real flow around inlets. Nevertheless, it shows that the performance of an inlet depends on the pressure gradient on both internal and external surfaces. The external pressure rise is fixed by the external compression and the ratio, \(A_{\text{max}}/A_i\), of maximum

![Diagram](image_url)

**FIGURE 6.4** Minimum frontal area ratio. Various pressure coefficients, \(C_{p\text{max}}\) (Eq. 6.3, with \(s = 0.5\)).
area to inlet area. The internal pressure rise depends on the reduction of velocity between entry to the inlet diffuser and entry to the compressor (or burner, for a ramjet). Nacelle size required for low drag can be quite strongly dependent on the degree of external deceleration. In realistic analyses one must consider compressibility effects.

**Inlet Performance Criterion**

As Chapter 5 showed, one may characterize the differences between actual and ideal performance of aircraft engine inlets by a “diffuser efficiency” or by a stagnation pressure ratio. We define these as follows:

**a. Isentropic efficiency.** Referring to Fig. 6.5, we can define the isentropic efficiency of a diffuser in this form:

\[ \eta_d = \frac{h_{02a} - h_a}{h_{0a} - h_a} \approx \frac{T_{02a} - T_a}{T_{0a} - T_a}. \]

State \((02a)\) is defined as the state that would be reached by isentropic compression to the actual outlet stagnation pressure. Since

\[ \frac{T_{02a}}{T_a} = \left( \frac{p_{02}}{p_a} \right)^{(\gamma - 1)/\gamma} \quad \text{and} \quad \frac{T_{02}}{T_a} = 1 + \frac{\gamma - 1}{2} M^2, \]

the diffuser efficiency \(\eta_d\) is also given by

\[ \eta_d = \frac{(p_{02}/p_a)^{(\gamma - 1)/\gamma} - 1}{[(\gamma - 1)/2] M^2}. \]

(6.4)

**b. Stagnation pressure ratio, \(r_d\).** The stagnation pressure ratio,

\[ r_d = \frac{p_{02}}{p_{0a}}, \]

(6.5)

**FIGURE 6.5** Definition of inlet states.
is widely used as a measure of diffuser performance. Diffuser efficiency and stagnation pressure ratio are, of course, related. In general,

\[
\frac{p_{02}}{p_a} = \frac{p_{0a}}{p_a} = \frac{p_{02}}{p_{0a}} \left( 1 + \frac{\gamma - 1}{2} M_a^2 \right)^{\gamma(\gamma-1)/2},
\]

and, with Eqs. (6.4) and (6.5),

\[
\eta_d = \frac{\left( 1 + \frac{\gamma - 1}{2} M_a^2 \right) (r_d)^{\gamma-1} - 1}{[(\gamma - 1)/2] M_a^2}.
\]  

(6.6)

Because \( \eta_d \) will be primarily affected by the internal deceleration ("diffusion"), it is unfortunate that these criteria are based on overall deceleration rather than on internal deceleration. The relationship between internal and external deceleration depends on engine mass flow rate as well as flight Mach number \( M \). But for illustrative purposes Fig. 6.6 gives typical values of stagnation pressure ratio \( r_d \). The diffuser efficiency \( \eta_d \) was calculated from \( r_d \), with the use of Eq. (6.6).

### 6.3 SUPersonic INLETS

Even for supersonic flight it remains necessary, at least for present designs, that the flow leaving the inlet system be subsonic. Compressors capable of ingesting a supersonic airstream could provide very high mass flow per unit area and, theoretically at least, very high pressure ratio per stage. However, the difficulty of passing a fully supersonic stream through the compressor without excessive shock losses (especially at off-design conditions) has so far made the development of fully supersonic compressors a possibility that is somewhat remote. As we will see in Chapter 8, the Mach number of the axial flow approaching a subsonic
compressor should not be much higher than 0.4; for a transonic stage it can be about 0.6. The term transonic refers here to the flow velocity relative to the blade tip, not to the absolute entrance velocity.

For a ramjet there is no such limitation based on compressor aerodynamics. Also it is possible to have combustion in a supersonic stream without prohibitive aerodynamic losses [7]. The acronym SCRAMJET denotes the supersonic-combustion ramjet, a concept that has long been under study for flight Mach numbers so high that the ramjet should be superior to the turbojet in propulsion efficiency. The SCRAMJET concept has not yet found application in a flight vehicle; ramjets developed to date require subsonic airstreams to provide stable combustion without excessive aerodynamic losses. We do, however, briefly discuss supersonic combustion in Section 6.5. Section 6.6 will give some insight into why a typical Mach number at inlet to a subsonic combustor is 0.4.

Here we focus attention on means of decelerating a supersonic flow to subsonic speeds tolerable by existing compressors or fans (or ramjet combustors).

**Reverse Nozzle Diffuser**

Figure 3.8 shows that deceleration from supersonic to subsonic flow speeds can be done by a simple normal shock with small stagnation pressure loss if the upstream Mach number is quite close to 1. For high Mach number the loss across a single normal shock would be excessive; in this case it would be better to use a combination of oblique shocks. Isentropic deceleration would be still better. From a simple one-dimensional analysis, it might appear that a supersonic (converging-diverging) nozzle operated in reverse, would be the ideal device to produce nearly isentropic deceleration. There are formidable difficulties, however, in implementing such a concept. These have chiefly to do with the problem of successful operation of a reversed-nozzle inlet over a range of flight Mach numbers and with the serious problem of the interactions of internal shocks with boundary layers. There is also the problem of flow instability; under certain conditions the flow can become quite violently oscillatory.

**The Starting Problem**

Internal supersonic deceleration in a converging passage (of nonporous walls) is not easy to establish. In fact, as we will now show, design conditions cannot be achieved without momentarily overspeeding the inlet air or varying the diffuser geometry. This difficulty is due to shocks that arise during the deceleration process, and it need not be related to boundary layer behavior. Therefore let us neglect boundary layer effects for the moment, while we examine the starting behavior of a converging-diverging diffuser that is one-dimensional and isentropic except for losses that occur in whatever (normal) shocks may be present. This simplified analysis contains the essential features of the phenomena, and it could be a valid representation of a real flow from which the wall boundary layer fluid was carefully removed by suction through porous walls.
Figure 6.7 illustrates successive steps in the acceleration of a fixed-geometry converging-diverging inlet. To isolate the inlet behavior from that of the rest of the propulsion device, we assume that whatever is attached to the diffuser exit is always capable of ingesting the entire diffuser mass flow. Thus mass flow rate is limited only by choking at the minimum diffuser area $A_i$.

Condition (a) illustrates low subsonic speed operation, for which the inlet is not choked. In this case the airflow through the inlet, and hence the upstream capture area $A_a$, is determined by conditions downstream of the inlet. In condition (b), though the flight velocity is still subsonic, the flow is assumed to be accelerated to sonic velocity at the minimum area $A_i$, and the inlet mass flow rate is limited by the choking condition at $A_i$. Since the flow is assumed isentropic, then $A_i = A^*$ and the upstream capture area $A_a$ is given by

$$\frac{A_a}{A^*} = \frac{A_a}{A_i} = \frac{1}{M} \left[ \frac{2}{\gamma + 1} \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{\frac{(\gamma + 1) / 2}{\gamma - 1}} \right]. \quad (3.15)$$

For sufficiently high subsonic values of $M$ [see point (b) in Fig. 6.8], we have

$$\frac{A_a}{A^*} = \frac{A_a}{A_i} < \frac{A_i}{A_i}.$$

Thus for these conditions the capture area $A_a$ must be less than the inlet area $A_i$, and therefore spillage will occur around the inlet.

For sonic or supersonic flight speeds the spillage mechanism is necessarily nonisentropic. That is, in order to “sense” the presence of the inlet and flow around it, the spilled air must be reduced to subsonic velocity upstream of the

![Figure 6.7](image-url) Successive steps in the acceleration and overspeeding of a one-dimensional supersonic inlet.
inlet plane. The mechanism for this deceleration is a detached “bow wave” that stands sufficiently far upstream to allow the required spillage. One can imagine the process of establishing the detached shock wave as follows: Suppose that when the air first reached supersonic velocity there were no shock. Then flow would have to enter, without deviation, the entire inlet area, in effect making $A_i$ act as the capture area $A_s$. But for low supersonic Mach numbers (see Fig. 6.8, where $A_i/A_f > A_*/A^*$), the allowable capture area as limited by choking at $A_i$ is less than $A_i$. Hence there would be an accumulation of mass and a rise in pressure in the inlet. This pressure rise would build up rapidly until a shock of sufficient strength moved upstream against the supersonic flow and became established at a position that would allow the required spillage.

Once the shock is established, the flow entering the inlet is no longer isentropic. Hence when the design Mach number of the aircraft is first reached, as at (d) in Fig. 6.7, the “reversed isentropic nozzle” mass flow cannot pass through the throat area $A_t$. This follows from Eq. (3.14), which indicates that the choked mass flow through a given area ($A_s$) is proportional to $p_0$, and from the fact that the fluid suffers a stagnation pressure loss in traversing the shock. For the flow that does enter the inlet (assuming isentropic flow from a point just downstream of the shock to the throat), Eq. (3.15) gives

$$\frac{A_2}{A_t} = \left(\frac{A}{A^*}\right)_2 = \frac{1}{M_2}\left[\frac{2}{\gamma + 1}\left(1 + \frac{\gamma - 1}{2} M_2^2\right)^{(\gamma + 1)/2(\gamma - 1)}\right], \quad (6.7)$$

where the subscript 2 refers to conditions just downstream of a normal shock. We may assume that the slightly curved shock may be approximated over the capture area by a normal shock, so that $M_2$ may be expressed as a function of the upstream Mach number $M$ by the normal shock relation

$$M_2 = \sqrt{\frac{[2/(\gamma - 1)] + M^2}{[2\gamma/(\gamma - 1)]M^2 - 1}}, \quad (6.8)$$
These two expressions may be combined to give the ratio of capture area to throat area, \( A_a/A_t \), as a function of flight Mach number for the flow that includes a detached normal shock. The result, plotted in Fig. 6.8, indicates that the inlet area \( A_t \) will remain too large and spillage will continue even beyond the design Mach number \( M_D \), unless the inlet can be overspeeded to a Mach number \( M_0 \). At this Mach number [or just below it, as at (e)], the inlet is capable of ingesting the entire incident mass flow without spillage. The shock position will be just on the lip of the inlet, as in Fig. 6.7(e), and a slight increment in speed, as to \( (e') \), will cause the shock to enter the convergence. Since a shock cannot attain a stable position within the convergence, it will move quickly downstream to come to rest within the divergence, at a position determined by downstream conditions. The flow to the throat is now isentropic, and the area ratio \( A_a/A^* \) (now greater than \( A_a/A_t \), since \( A_t > A^* \)) would be given by \( (e') \) of Fig. 6.8. The incoming flow is decelerated from \( A_t \) to \( A_r \), whereupon it is reaccelerated supersonically in the divergence. Having thus attained isentropic flow in the inlet, the Mach number may be reduced from \( M_0 \) to \( M_D \), as at (f). At exactly the design speed, the throat Mach

\[ \dot{m}_i = p_0 \left[ \frac{A_t}{\sqrt{RT_0}} \sqrt{\frac{2}{\gamma + 1}} \right] ^{(\gamma + 1)/(2\gamma - 1)} = \epsilon \rho_{02}, \]

where, for constant inlet velocity (flight speed), \( \epsilon \) is a constant and \( p_{02} \) is the stagnation pressure downstream of the shock. Now suppose that a small disturbance moves the shock slightly upstream where the shock Mach number is slightly higher. A greater stagnation pressure loss will occur across the shock, lowering the downstream stagnation pressure and the throat mass flow. Since conditions upstream of the shock are not affected, the mass flow through the shock remains constant and there results an accumulation of mass and a rise in pressure behind the shock. As the static pressure ratio across the shock increases, its propagation speed relative to the fluid increases and the shock moves farther upstream. This further increases the stagnation pressure loss, of course, so that the mass flow imbalance increases and the shock continues to move upstream and out of the inlet. The final position of the shock is far enough outside the inlet area to allow the correct "spilling" of air. By similar arguments one can see that slight motion downstream by the shock is continued until the shock is carried through the throat.

If the flow downstream of \( A_t \) is not choked, an appropriate downstream boundary condition might be that at some exit area \( A_e \), a constant static pressure \( p_e \) is maintained. In this case, for steady flow,

\[ \dot{m}_i = \dot{m}_e = p_e u_e A_e = \frac{A_e p_e}{RT_e} \sqrt{2C_p(T_e - T_r)}. \]

Assuming isentropic flow downstream of the shock, we can express this as

\[ \dot{m}_i = \frac{A_e p_e}{\sqrt{RT_e}} \sqrt{\frac{2\gamma}{\gamma - 1} \left( \frac{p_{02}}{p_e} \right)^{\gamma - 1} - 1}. \]

Thus it can be seen that, since \( \dot{m}_i \) is dependent only on \( p_{02} \), changes in shock position result in a mass flow imbalance similar to that for the choked case, and the same conclusions hold: Since any shock would exist in the convergence by moving into it, it follows that no shock can reach a stable position within the convergence.
number would be just unity and isentropic deceleration from supersonic to subsonic flow would exist. Even for this simplified model, however, this condition would be unstable. A slight decrease of flight speed or increase of back pressure would require spillage, and a shock would move rapidly out through the inlet to reestablish condition (d). Thus it might be better to maintain the throat Mach number slightly greater than unity while reaching subsonic velocities through a very weak shock just downstream of the throat.

This simplified description contains the essential feature of the starting problem associated with an internally contracting passage. That is, an inlet having $A_t/A_l$ greater than 1 will always require spillage upon reaching supersonic flight velocities, since $A_o/A_l$ will always pass through a minimum of 1 just as sonic flight velocity is attained. It is necessary to perform some operation other than simply accelerating to the design speed in order to “swallow” the starting shock and establish isentropic flow. Overspeeding is one such operation, but there are others.

If overspeeding is not feasible (note that, except for very modest design Mach numbers, substantial overspeed would be required), it might be possible to swallow the shock by a variation of geometry at constant flight speed. The principle is easily seen in terms of simple one-dimensional analysis. Suppose the inlet is accelerated to the design Mach number $M_D$ with the starting shock present, as at point (d) in Figs. 6.8 and 6.9. Now, if the actual area ratio can be decreased from $A_t/A_l$ to the value that can ingest the entire inlet flow behind the shock, the shock will be swallowed to take up a position downstream of the throat. This variation would normally involve a momentary increase of throat area from $A_t$ to a new value that we will call $A_t'$ (see Fig. 6.9). Having thus achieved isentropic flow within the convergence, the throat Mach number $M_D'$ is greater than 1, and a relatively strong shock occurs farther downstream. Completely isentropic flow

![Diagram](image)

**FIGURE 6.9** Shock swallowing by area variation.
can then be achieved by returning the area ratio to its original value, while the operating conditions move from (d) to (f).

A geometric variation such as that shown schematically in Fig. 6.9 would be difficult, mechanically, for axisymmetric flow. However, geometries that permit the axial motion of a center plug between nonparallel walls can be used. A somewhat similar effect can be had through the use of porous convergent walls. With the shock external, the high static pressure within the convergence causes considerable “leakage” through the walls, thus effectively increasing the throat area to permit shock swallowing. Once this occurs, the lower static pressure within the convergence somewhat decreases the flow loss. But since the total porosity required is even greater than the throat area [8], there remains a high mass flow loss under operating conditions. This is desirable to the extent that it removes boundary layer fluid, but the lost flow is more than is needed for this purpose. Flow that is decelerated but not used internally contributes substantially to the drag of the propulsion device.

From this discussion it would seem that one could avoid the starting problem altogether by using a simple divergent inlet, that is, one for which \( A_i/A_t = 1 \), as shown in Fig. 6.10. For supersonic flight speeds and sufficiently low back pressure, it is possible to accelerate the internal flow within the divergence before decelerating it in a shock. To reduce stagnation pressure losses, it is desirable to have the shock occur at the minimum possible Mach number, which, for this geometry, is the flight Mach number. One can achieve this condition by adjusting the back pressure (by varying the engine exhaust area, for example) so that the shock is positioned just on the inlet lip.

A slight improvement, called the Kantrowitz-Donaldson inlet by Foa [9], is also illustrated in Fig. 6.10. This configuration uses the maximum internal convergence that will just permit shock swallowing at the design flight Mach number. Referring to Fig. 6.9, we can see that this is just \( A_i/A_t \) for a design Mach number.

**FIGURE 6.10** Fixed-geometry diffuser with intentional normal shocks at the design Mach number \( M_D \). (Refer to Fig. 6.9.)
of $M_D$. As in the simple divergent inlet, it is necessary to adjust the back pressure to assure that the shock occurs at the minimum possible Mach number. The advantage of the internal convergence is, of course, that this minimum Mach number is less than the free-stream value. A disadvantage is that there is an abrupt change in performance just at the design Mach number, since the shock will not be swallowed below this value and it will be immediately disgorged if the speed falls off slightly from the design value. Further, since shocks of any origin are unstable within the convergence, such an inlet would be quite sensitive to changes in angle of attack.

The condition in which a shock just hangs on the inlet lip is called the critical condition. Operation with the shock swallowed is called supercritical, whereas that with a detached shock and spillage is called subcritical. Note that subcritical operation may occur as the result of choking in the inlet, as discussed here for inlets alone; or, for a complete engine, as the result of any downstream flow restriction that cannot accept the entire mass flow $\rho_u u A_i$.

### The Shock–Boundary Layer Problem

As we noted earlier, sketches such as Figs. 6.9(d') and 6.10 are realistic only if the boundary layer fluid is removed from the walls between which the shock is located. Across a shock wave of appreciable strength, the boundary layer separates, and this separation may have a large effect on the structure of the shock. We might suppose, in accord with the boundary layer concepts discussed in Chapter 4, that a turbulent boundary layer will separate when the shock-induced velocity reduction is between 20% and 30% (0.36 < $C_p$ < 0.51). Using the normal shock relationships of Chapter 3, we can see that this would correspond to upstream Mach numbers of 1.15 (20% reduction in velocity) or 1.25 (30% reduction).

To begin, we consider the interaction of a weak shock with a boundary layer. Figure 6.11 is a series of schlieren photographs of the interaction of a shock wave and a boundary layer for upstream free-stream Mach numbers ranging from 1.30 to 1.55. Schlieren photographs show regions of high-density gradient [7], so they reveal sharply the presence of stationary shock waves as well as the presence of boundary layers in a compressible flow. In Fig. 6.11(a) we can see what is nearly a plane normal shock wave (with an upstream Mach number $M$ of 1.3) and a wall boundary layer whose thickness is growing perceptibly in the flow direction. In Fig. 6.11(d), for $M = 1.4$, the shock wave near the boundary layer has taken up a “lambda” shape, with an oblique shock reaching ahead of the main shock location. Because a large fraction of the boundary layer is subsonic, the pressure rise due to the shock is sensed (near the wall) some distance ahead of the main shock wave. This upstream pressure gradient causes the boundary layer to grow rapidly, as one can see from close inspection of Fig. 6.11(d). The increase in displacement thickness affects the flow in the same way as would a wedge in inviscid flow; hence an oblique shock is formed ahead of the main wave.

As the free-stream Mach number increases above perhaps 1.25, the boundary layer thickens very rapidly under this lambda shock system (see, e.g., Fig. 6.11(i)) and causes the boundary layer to separate. The pressure gradient near the wall
FIGURE 6.11 Interaction of a normal shock wave and a boundary layer. Numbers indicate upstream Mach number. (Courtesy C. J. Atkin and L. Squire of Cambridge University.)
has become too large for the slow-moving fluid near the wall to continue moving in the main flow direction.

Figure 6.12 shows the effect of a strong shock in a diverging duct. Here the main stream-flow adjustment is very far from the simple normal shock picture in Fig. 6.11. Instead the interaction between the “shock” and the boundary layer results in at least five major pressure adjustments (shown by the X-like dark regions in Fig. 6.12). Large separation zones cause a highly distorted, and probably unsteady, flow field that may require an axial distance of 10 duct widths or more to return to reasonable uniformity of flow. The flow field disturbances and distortions shown in Fig. 6.12 would have seriously harmful effects on the behavior of a compressor or combustor placed immediately downstream.

The chief lesson here is that unless one makes a strenuous effort to remove the wall boundary layer, strong shocks may have disastrous effects on duct flow. If a shock wave must be placed in a supersonic stream of given Mach number, then:

a. An oblique shock is much better than a normal one because the pressure rise is less;

b. The shock should interact with the wall at the point where the boundary layer is thinnest—preferably at the leading edge for the simple diverging inlet of Fig. 6.10.

**External Deceleration**

The absence of a starting problem for the normal shock inlet is offset by the accompanying stagnation pressure loss at all but low flight Mach numbers (less than about 1.5). For example, Fig. 3.8 indicates a 28% stagnation pressure loss at $M = 2 (\gamma = 1.4)$. If we wish to obtain reasonable performance while maintaining the starting characteristics of a simple divergent inlet, then it is clear that some external deceleration must occur upstream of the inlet plane in order to reduce the Mach number of the normal shock to a suitable value.

The simplest and most practical external deceleration mechanism is an oblique shock or, in some cases, a series of oblique shocks. Though such shocks are

![FIGURE 6.12 Shock-boundary layer interactions in a duct. Flow from left to right. (Courtesy M.I.T. Gas Turbine Laboratory.)](image)
not isentropic, the stagnation pressure loss in reaching subsonic velocity through a series of oblique shocks followed by a normal shock is less than that accompanying a single normal shock at the flight velocity. The losses decrease as the number of oblique shocks increases, especially at high flight Mach numbers.

In the external compression process, shocks and boundary layers may interact strongly, so that it is highly desirable to locate the oblique shocks at points where boundary layers are absent. This can be arranged easily if one uses a center body (primarily for axisymmetric flow), as in Fig. 6.13. These schematics, taken from Oswatitsch [10], illustrate the typical single oblique shock system and two double oblique shock systems. Although the double shock systems theoretically give better performance, one can see that several problems may arise in their use. In the configuration of Fig. 6.13(b), the second shock, generated by a turn of the wall, occurs at a point where the boundary layer has had time to develop, and separation may result. The configuration of 6.13(c) avoids this at the point of shock generation, but the second shock still intersects the boundary layer on the center body.

With (if need be) boundary layer removal, we can appreciate the performance gain to be expected through the use of multiple oblique-shock deceleration by looking at a two-dimensional example. Consider the diffuser in Fig. 6.14, in which the flow is deflected through two 15° angles before entering a normal shock. Using Fig. 3.12, we can show that the Mach numbers in regions (2), (3), and (4) are 2.26, 1.65, and 0.67, respectively. Then, using Fig. 3.11, we can obtain the stagnation pressure ratios as follows:

\[
\frac{p_{02}}{p_{01}} = 0.895, \quad \frac{p_{03}}{p_{02}} = 0.945, \quad \frac{p_{04}}{p_{03}} = 0.870.
\]

Thus the overall stagnation pressure ratio is approximately \(p_{04}/p_{01} = 0.735\). If the deceleration had been achieved by a single normal shock, the overall stagnation pressure ratio would have been only 0.33. Remember that these estimates do not include losses due to boundary layer effects, which may be especially important in the subsonic diffuser.

This example does not necessarily employ the best arrangement of three shocks, of course, since a variation of their relative strengths might provide a higher overall stagnation pressure ratio. For the simple two-dimensional case, in

![FIGURE 6.13 Typical configurations for oblique shock diffusers. (Adapted from Oswatitsch [10].)](image)
which conditions downstream of each shock are uniform, Oswatitsch [10] has shown by theoretical analysis that, for a given flight Mach number and a given number of oblique shocks followed by one normal shock, the overall stagnation pressure ratio will be maximized if all the oblique shocks have equal strength, that is, if the Mach numbers of the velocity components normal to each shock, and incident to it, are equal. It follows, of course, that the stagnation pressure ratios will also be equal. This result will not seem unreasonable if we recall that for a normal shock the stagnation pressure loss rises very quickly with incident Mach number. Oswatitsch found that the Mach number of the final normal shock should be less than the normal component of the oblique shocks (about 0.94 times as great). Figure 6.15 shows the best performance using $n$ oblique shocks of equal strength followed by one normal shock.

For axisymmetric inlets and conical shocks, the best arrangement is not so easily determined, since the downstream flow of a conical shock is not uniform. In fact, fluid properties are constant along conical surfaces emanating from the shock vertex, and the streamlines downstream of the shock are curved [11]. The effect of the streamline curvature is further diffusion, which can be very nearly isentropic, so that subsequent shocks occur at reduced (but nonuniform) Mach numbers. It is even possible to achieve deceleration to subsonic velocities behind a conical shock without subsequent shocks, as shown for a rather low Mach number in Fig. 6.16.

The performance advantage of multiple conical shocks is qualitatively similar to that of the multiple-plane shocks shown in Fig. 6.15. Going to the limit (for
either axisymmetric or two-dimensional geometries) of an infinite number of infinitesimal shocks, generated by continuous wall curvature, one could, theoretically at least, achieve isentropic external deceleration to sonic velocity. Such an inlet, indicated qualitatively in Fig. 6.17, would seem to provide the ideal geometry to achieve low losses, while at the same time avoiding the starting problems of an internal convergence. However, several practical difficulties would be encountered in the operation of such an inlet. This geometry, like that of the isentropic internal flow diffuser, would function properly at only one Mach number, and performance would be very sensitive to angle of attack. Furthermore, the boundary layer along the curved surface, unlike that along plane or conical surfaces, would be subject to a high adverse pressure gradient, which might cause separation. Finally, for high flight Mach numbers it would be necessary that the flow turn through large angles before reaching sonic velocity. The resultant large cowl angle would at least exhibit high drag and perhaps even interfere with the inlet flow.

It will be clear from the foregoing that variable geometry is an almost inescapable requirement for an engine inlet that must operate at both subsonic and supersonic speeds. Fig. 6.18 shows the design of the two-dimensional intake adopted for the Concorde aircraft, whose design flight Mach number is 2; the Concorde is also required to cruise over certain land areas at subsonic speeds.

Figure 6.18 shows the intake geometry during takeoff; here the ramp assembly is raised to allow as much air as possible to the engine. Shock waves are of
The Flow Stability Problem

The operation of external shock diffusers is divided into subcritical, critical, and supercritical modes, which depend on the external and internal shock configuration. The three modes are shown schematically for a typical case in Fig. 6.19. With entirely diverging internal flow such as this, the normal shock position is determined by a downstream flow restriction rather than by the inlet geometry. Hence the operating mode is sensitive to variations in exhaust-nozzle area and fuel flow rate. Subcritical operation entails "spilling" of flow and a normal shock upstream of the inlet. "Low" and "high" subcritical operations differ only in the extent of spilling. Supercritical operation occurs at the same mass flow as critical operation, but with increased losses, since the normal shock occurs at a higher Mach number.

Numerous investigators have tested the performance of experimental supersonic diffusers, owing to the substantial importance of inlet performance, especially at high Mach numbers. A wide variety of geometries have been considered, each depending on a particular application. Special attention is paid to the off-design performance of an inlet; this is of great importance in an actual flight application but is not so amenable to analysis as the design performance. Various adjustable inlets have been considered to extend the favorable operating range of an inlet.

The data of Dailey [13] are typical of many investigations of an important instability that occurs during the subcritical operation of most supersonic inlets. This phenomenon, known as "buzz," consists of a rapid oscillation of the inlet
shock and flow pattern; the resultant internal disturbance is very detrimental to engine performance. In a ramjet, for instance, the onset of buzz usually extinguishes combustion. Although the pulses of the shock system are similar, the interval between pulses is not constant [13]; hence buzz cannot be considered a periodic phenomenon. Although it is not thoroughly understood at present, buzz has been shown to be a function of conditions only at, and immediately downstream of, the inlet. In some cases boundary layer bleed from the center body can delay the onset of buzz. In other cases the use of a length of nondivergent subsonic passage just downstream of the inlet can have a beneficial effect. This latter case could be attributed either to the establishment of a healthy boundary layer (as compared with a separated one just downstream of the normal shock) or to the establishment of more uniform free-stream conditions (see below) before subsonic diffusion is attempted.

Figure 6.20 is a plot of typical diffuser performance as a function of mass flow, expressed as the ratio of actual to critical (or supercritical) mass flow rate $\dot{m}_c$, or the equivalent ratio of actual to critical capture area. Ratios of $\dot{m}/\dot{m}_c$ less than 1 signify subcritical operation. One can see that for mass flows slightly less than critical (the so-called high-subcritical flow) the stagnation pressure ratio increases slightly with decrease in mass flow rate. This is explained by the presence of lower aerodynamic losses in the internal passages due to reduced mass flow and velocity. Overall performance, of course, does not increase with subcritical operation, since spilling is accompanied by both increased drag and decreased thrust. As the flow is further reduced, the normal shock is pushed farther stream and the stagnation pressure ratio decreases. A portion of the air entering the cowl may travel through a single strong shock (see “low subcritical,” Fig. 6.19)
rather than the two weaker shocks it would pass through during critical operation. The entrance of this low-stagnation pressure (or high-entropy) air causes a reduction in performance. In supercritical operation the stagnation pressure ratio drops rather rapidly at constant mass flow rate.

We can see by comparing Figs. 6.20(a) and (b) that the ingestion of low-stagnation pressure air may or may not correspond to the onset of buzz (although it usually does).

For both subsonic and supersonic intakes, the best balance of external and internal deceleration is a matter of concern because of the need to keep nacelle drag as low as possible and to supply uniform flow to the engine. For the subsonic intake, fixed geometry can generally provide acceptable boundary layer behavior over all flight speeds from takeoff to cruise. For supersonic flight, shock waves are a practical necessity for deceleration of the inlet air to subsonic speeds; the inlet geometry required to minimize shock and boundary layer losses depends strongly on Mach number. This means that to cope with a wide range of flight Mach numbers, variable geometry is needed. The control of the intake geometry must be done carefully to avoid flow instability. Large perturbations in the geometry of the inlet shock pattern could disturb the engine intake flow sufficiently to cause serious problems for the compressor or burner.
bustor to achieve high efficiency. For both possibilities LeFebvre [15] describes the characteristic dependency of combustor efficiency on pressure, liner pressure drop, mass flow rate, entrance temperature, and combustor cross-sectional area and length.

**Chamber Geometry**

Over the years combustion chamber geometry has evolved considerably with respect to the objectives of:

a. Improving flame stability (both at sea-level and altitude conditions);

b. Reducing chamber size (while still burning all the fuel and maintaining reasonable pressure drop);

c. Reducing emissions of the oxides of nitrogen, carbon monoxide, and unburned hydrocarbons;

d. Increasing chamber life;

e. Controlling the temperature distribution at inlet to the turbine.

Combustion stability, intensity, and efficiency all depend heavily on the fluid flow and turbulence distribution within the combustor. So does the heat transfer to the chamber walls. For these reasons much combustor development has focused on internal flow patterns. Much of the historical development was preceded by experimental trials of prototype combustors or studies of flow patterns in geometries similar to those in combustors. Only in recent years have analytical tools been developed for detailed mathematical simulation of combustion flow fields. These, in conjunction with experiments, can assist designers in optimizing combustor geometry. In principle these analytical methods should be able to predict turbulence intensity, reaction rates (including rates of formation of pollutants), and both convective and radiative heat transfer rates. Much, however, remains to be done to develop their full potential. Combustor development still relies heavily on laboratory testing.

Early models of gas turbine combustors (e.g., the pioneering Whittle engines developed in the late 1930s in the United Kingdom) had reverse-flow combustors (shown schematically in Fig. 6.27) mainly to keep turbine and compressor close together. At the time, this appeared to be necessary to prevent whirling vibrations of the shaft. Subsequent improvements in shaft design made it possible to eliminate whirling vibrations in long shafts so that straight-through flow combustion chambers could be used. These chambers were easier to develop for acceptable flow distribution and combustion efficiency, though reverse-flow combustors are still used in small turboshift engines where overall cross-sectional area is not a serious concern.

In each type of combustor the chemical reaction is confined to the interior of a perforated metal liner mounted within the outer casing. Airflow between the liner and the casing keeps the metal cool enough to retain its strength despite the very much higher temperatures of the central combustion zone.

Figure 6.28 shows a typical configuration for a can-annular combustor in which 10 cylindrical combustion zone liners are enclosed within an annular space between the inner and outer combustion casings. At the entrance to each can is
a passage of diverging area to reduce the velocity from a typical compressor outlet value (100–150 m/s) to the bulk flow average velocity in the combustion zone (20–30 m/s). Shown also are the swirl vanes used to set up high swirl within the combustor. The chief value of the swirl is in establishing a region of backflow or recirculation within the central and forward part of the combustion chamber; the flame is stabilized within this region of low and reversing velocity. Swirl also helps to distribute the fuel droplets leaving the atomizer and to intensify the turbulence needed for rapid combustion. The combustion liners, or flame tubes, as they are denoted in Fig. 6.28, have many small holes for transpiration cooling of
the liner, larger holes for dilution of the combustion products, and interconnector passages through which the pressure is communicated from one combustion zone to another. The interconnectors help to maintain circumferentially uniform outlet conditions.

Figures 6.29 and 6.30 show examples of fully annular combustion chambers in which the combustion zone itself occupies an annular space. Figure 6.29 shows also the annular diffuser at the entrance to the combustor. Air dilution holes on the inner surface of the combustion chamber liner are indicated, as well as annular slots on the outer surface, which provide both cooling and dilution air. The liner geometry shown in Fig. 6.30 is quite different. It has a large number of small holes through which air enters in small jets to provide a stable combustion zone and cooling air for the walls. Shown are 18 fuel and air inlets to the individual combustion initiation zones within the annular chamber.

LeFebvre [15] and Odgers and Kretschmer [16] discuss the relative advantages of can, can-annular, and fully annular combustion chambers. The tubular or can-type chamber has the advantages of ease of control of the fuel–air ratio and simplicity and low cost of replacement of a damaged liner, as well as the important advantage of requiring a relatively small air supply for combustion chamber testing during the development period. Though interconnectors have been commonly used for tubular designs, each chamber can be designed and tested as though it behaved independently of the other chambers. The disadvantages of this configuration for large engines include the relatively large size and weight of the chamber components, the relatively large pressure drop, and the need to provide numerous igniters. It is not easy with the tubular configuration to provide nearly simulta-

![Diagram of Annular Combustion Chamber](image-url)
neous ignition of all chambers. The disadvantages have outweighed the advantages, and tubular chambers are now seldom used for large gas turbines for aircraft applications.

The can-annular-chamber has the advantages of providing ease of ignition, minimum total cross-sectional area, minimum pressure drop, and minimum length and weight. The disadvantages include difficulties in development to obtain circumferentially uniform fuel–air ratio and outlet temperature. A failure of the liner in one spot means replacement of a relatively expensive component. There tends to be a heavy buckling load, due to thermal expansion on the outer surface of the chamber liner.

Both the annular and the can-annular chambers require large airflow rates during testing. The latter requires somewhat more volume and poses more difficult ignition problems than the former. Also it will typically have higher pressure drop. It can be mechanically more robust, however, because of the geometry of the combustion liner. Also it allows better control of the circumferential distribution of the fuel–air ratio and the outlet temperature.

To create the high turbulence necessary for intense combustion, there must be considerable pressure drop across the small holes in the chamber liner through which most of the air enters the combustion zone. Other causes of pressure drop in combustors include the bulk acceleration of the hot gas as its density decreases because of combustion (in a roughly constant area channel), and friction on the channel walls. For best engine performance the sum of these pressure losses should not be greater than a few percentage points of the combustor inlet pressure. LeFebvre [15] quotes typical overall combustor pressure drops as follows:

<table>
<thead>
<tr>
<th>Type</th>
<th>Pressure Drop</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tubular</td>
<td>7%</td>
</tr>
<tr>
<td>Can-annular</td>
<td>6</td>
</tr>
<tr>
<td>Annular</td>
<td>5</td>
</tr>
</tbody>
</table>