CP301

Development Engineering Project

Micro Jet Engine for UAV Applications

Under the guidance of

Dr. Rajendra Munian & Dr. Sreekanth Shekar Padhee

Assistant Professor, Department of Mechanical Engineering

IIT Ropar

Submitted by

Kanniah Guptha Ainala 2022MEB1292

Karthikey 2022MEB1320

Lalam Gnana Deepak 2022MEB1322

YGSV Ashish 2022MEB1367

Semester II, AY 2024-25

ACKNOWLEDGEMENT

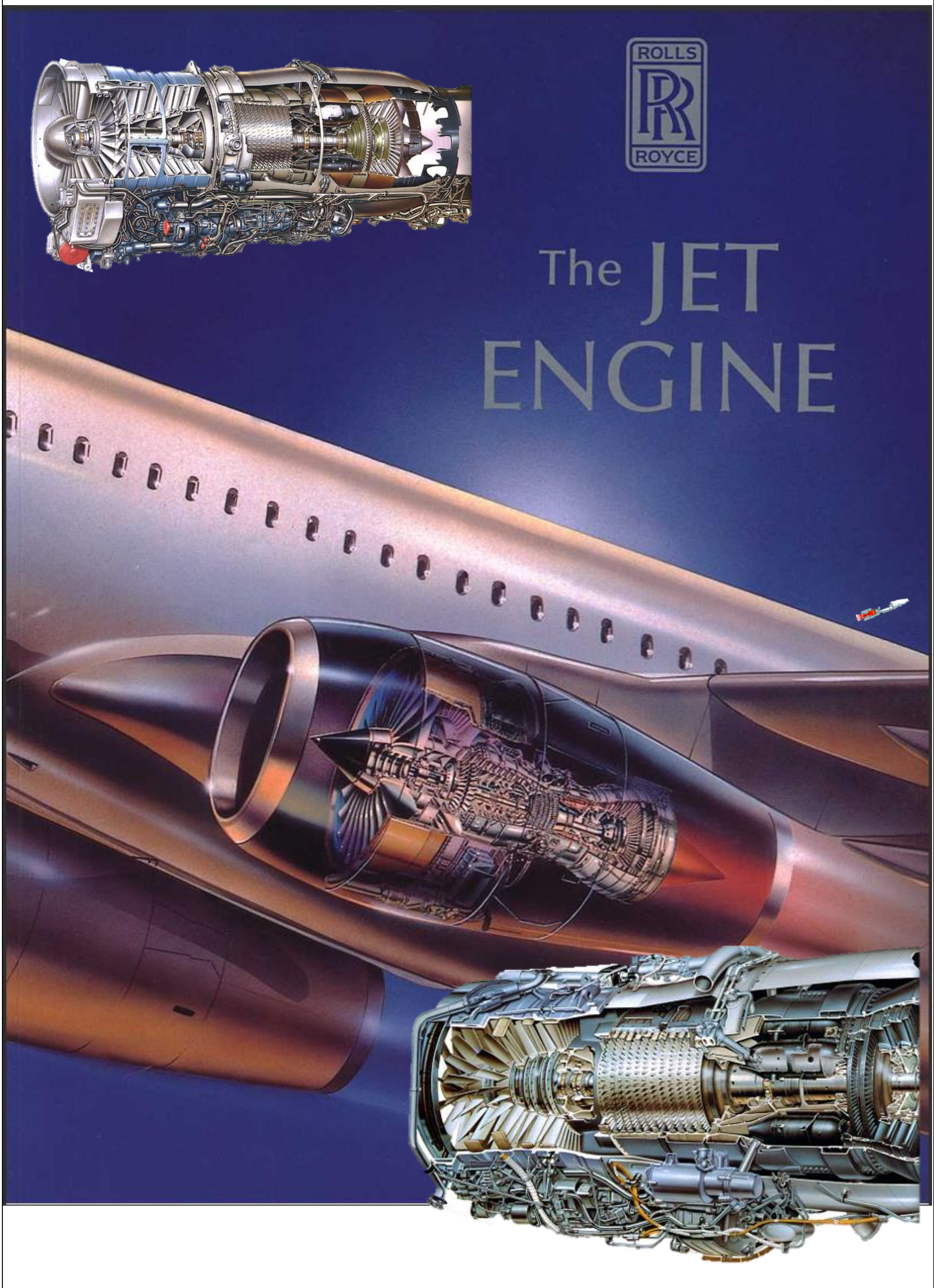
We would like to express our deepest gratitude to our esteemed guides, **Dr. Rajendra Munian** and **Dr. Sreekanth Shekar Padhee**, for their invaluable guidance, continuous support, and insightful suggestions throughout this project on **Micro Jet Engine for UAV Applications**. Their expertise and encouragement have played a crucial role in shaping our understanding and approach toward this research.

We also extend our sincere appreciation to the faculty and staff of the **Mechanical Engineering Department, IIT Ropar**, for providing the necessary resources and a conducive environment for our work. Their assistance has been instrumental in the successful completion of this project.

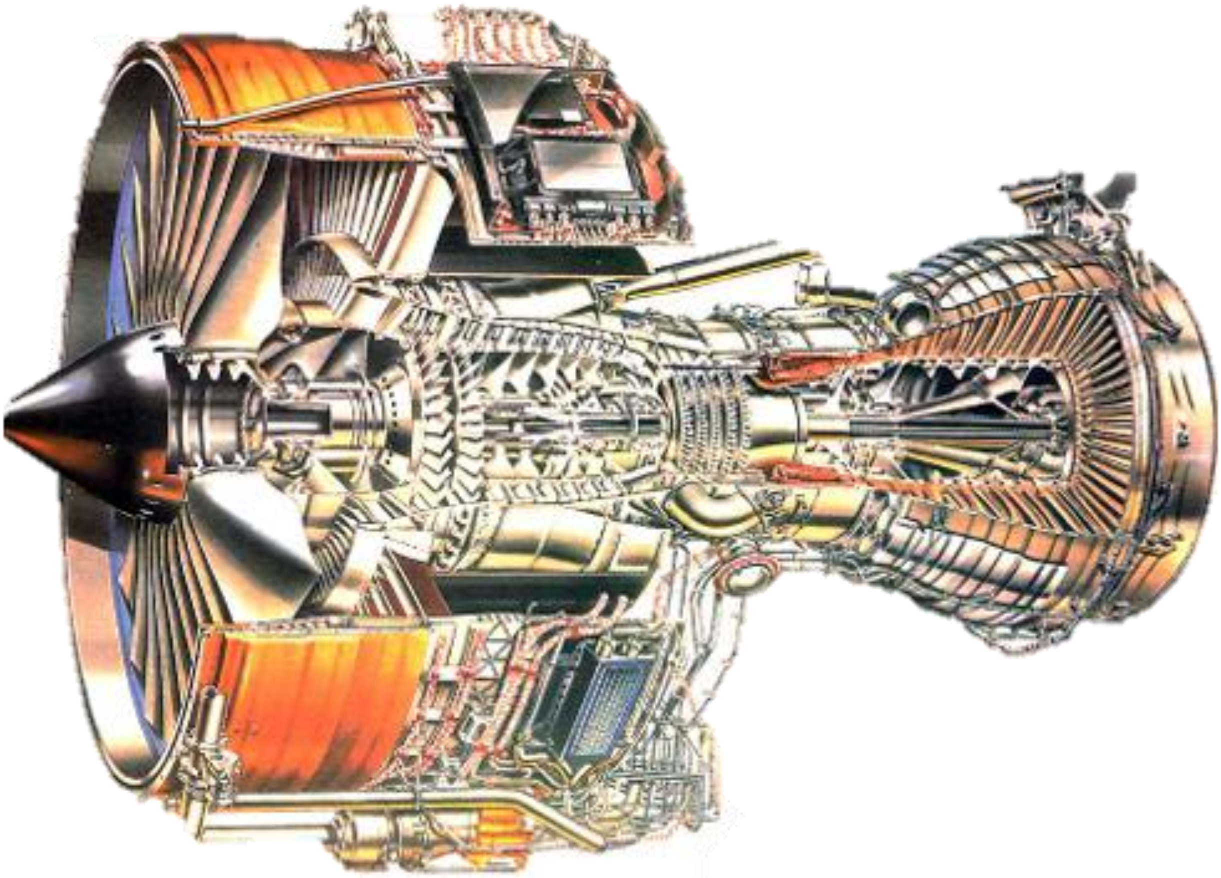
We would also like to acknowledge the various research papers, technical resources, and prior works in the field of micro jet engines, which have significantly contributed to our understanding of the subject. The knowledge gained from these references has greatly aided in developing the framework of our study.

A special thanks to our peers for their constructive discussions and collaborative spirit, which have enriched our learning experience. Their valuable input and critical feedback have helped refine our ideas and improve our work.

Lastly, we would like to extend our heartfelt gratitude to our families for their unwavering encouragement, patience, and motivation. Their constant support has been a source of strength throughout this journey.


ROLLS
ROYCE
The JET
ENGINE

Index:



Rolls-Royce Trent 800

Title of the Project:

Micro Jet Engine for UAV Applications

Aim:

To design, develop, and construct an efficient micro jet engine capable of generating 300 N of thrust, optimized for UAV applications, ensuring high performance and reliability.

Introduction:

The advancement in UAV technologies demands compact yet powerful propulsion systems. This project addresses the development of a micro jet engine that meets the thrust and efficiency requirements for UAVs. By leveraging contemporary design methods, cycle analysis, and state-of-the-art manufacturing techniques (including metal 3D printing), the project aims to deliver an engine that not only satisfies the performance criteria but also offers robustness and ease of integration.

Background and Literature Review:

Figure 1: Hero's Aeolipile (Source: Knight's American Mechanical Dictionary, 1876)

Historically, jet engines have evolved from simple reaction devices to complex systems incorporating multiple stages of compression, combustion, and expansion. Key literature [Benini and Giacometti, 2007; Fahlström and Pihl-Roos, 2016; Putra, 2020] details the challenges encountered in small-scale gas turbines, such as low Reynolds number effects, high rotational speeds (above 30,000 rpm), and critical thermal management issues. The use of an annular combustion chamber, for instance, provides advantages in terms of:

- Reduced combustion chamber length and weight.
- Lower pressure drop (approximately 5% loss).
- Improved flame stability and ignition reliability.

Cycle analysis typically follows an open Brayton–Joule model. The thermodynamic state at various stations is characterized by relations such as:

$$T_{02} = T_{01}\left(\frac{P_{02}}{P_{01}}\right)^{\frac{\gamma-1}{\gamma}},$$

$$\Delta T = \frac{Q}{\dot{m}_a c_p},$$

and the thrust equation:

$$T = \dot{m}_a\,(v_e - v_0),$$

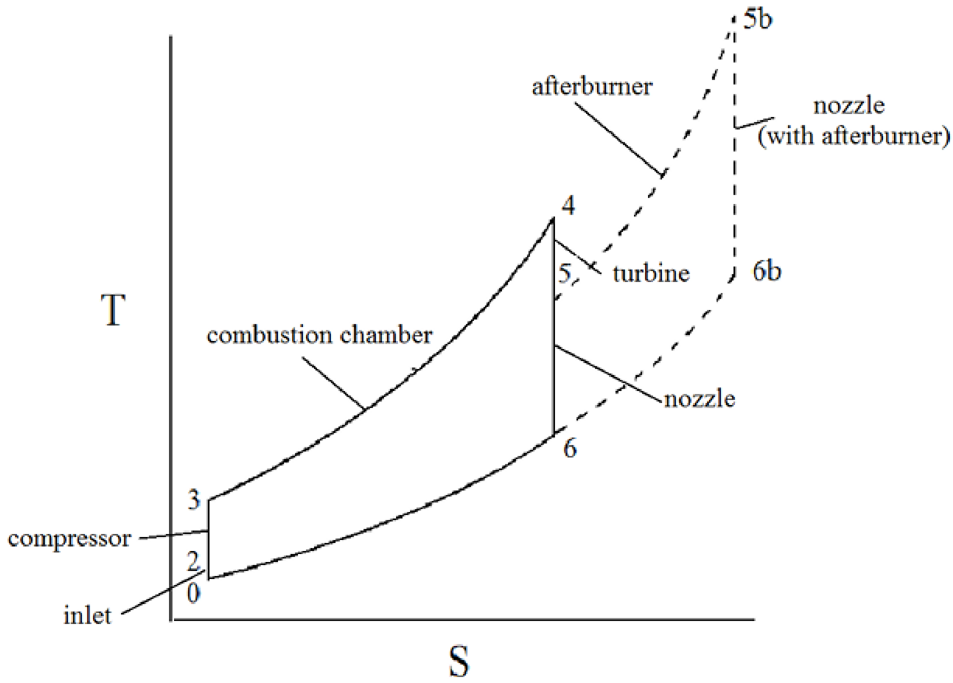


Figure 4: TS diagram of the ideal Brayton Cycle for Turbojet

where m˙a is the air mass flow rate, ve is the exhaust velocity, and v0 is the inlet velocity(assumed nearly zero for static thrust).

For the compressor, design calculations involve one-dimensional approximations (using continuity and energy equations) as well as three-dimensional CFD simulations to refine the impeller and diffuser geometry. For example, the compressor impeller is designed using:

$$\dot{m}_a = \rho\,A\,v,$$

with $A = \pi\left(\frac{D_{\rm out}^2 - D_{\rm in}^2}{4}\right)$. In one design case [Benini and Giacometti, 2007], an impeller with an outer diameter of 129 mm, an inlet diameter of 74 mm, and a tip speed of approximately 405 m/s at 60,000 rpm was used to achieve a compression ratio near 2.66.

Objective:

1. Perform a comprehensive thermodynamic cycle analysis for a 300 N thrust micro jet engine using the Brayton–Joule cycle.
2. Determine key design parameters:
3. Air mass flow rate, compressor pressure ratio, and cycle temperatures.
4. Geometric dimensions for the compressor, combustion chamber, and nozzle.
5. Select appropriate materials (e.g., aluminum alloys, stainless steel) that are manufacturable at the university level.
6. Validate the design using CFD and thermal analysis to predict off-design behavior.
7. Develop fabrication and testing strategies, including the use of CNC machining and metal 3D printing.

Technical Requirements:

A very rough preliminary estimation of parameters is provided to give a general idea of the overall requirements and specifications of the jet engine. A detailed analysis is followed in the design section.

- **Thrust Requirement:**
  $T = 300\,\mathrm{N}$
- **Thermodynamic Cycle Parameters:**
  Ambient conditions: $P_0 = 101.3\,\mathrm{kPa}$, $T_0 = 288\,\mathrm{K}$.
  Compressor exit conditions, based on an assumed pressure ratio of ~2.66, yield:

$$T_{02} = 288\,(2.66)^{\frac{0.4}{1.4}} \approx 416\,\mathrm{K}$$

- **Mass Flow Rate:**
  Using the thrust equation and assuming an exhaust velocity v_e (e.g., 500–860 m/s), the required mass flow rate is calculated as:

$$\dot{m}_a = \frac{T}{v_e} \quad \text{(for static conditions, with } v_0 \approx 0\text{)}$$

  For instance, if $v_e = 500\,\mathrm{m/s}$:

$$\dot{m}_a = \frac{300\,\mathrm{N}}{500\,\mathrm{m/s}} = 0.6\,\mathrm{kg/s}$$

- **Compressor and Combustion Chamber Sizing:**
  Compressor inlet area is defined by:

$$A = \pi\left(\frac{D_{\text{out}}^2 - D_{\text{in}}^2}{4}\right)$$

  With air density $\rho \approx 1.225\,\mathrm{kg/m^3}$ and an assumed velocity of $50\,\mathrm{m/s}$, the area is:

$$A \approx \frac{0.6\,\mathrm{kg/s}}{1.225 \times 50} \approx 0.0098\,\mathrm{m^2},$$

  yielding a compressor diameter of approximately 11.2 cm.

- **Combustion Chamber Temperature Control:**
  Lean combustion (e.g., fuel-to-air ratio ~1:50) is used to limit peak temperatures. For a kerosene LHV of ~43 MJ/kg, the temperature rise is:

$$\Delta T = \frac{\dot{m}_f\, LHV}{\dot{m}_a\, c_p},$$

where $\dot{m}_f$ is the fuel mass flow rate. For example, if $\dot{m}_f \approx 0.03\,\mathrm{kg/s}$ and $c_p \approx 1005\,\mathrm{J/kg}$K

$$\Delta T \approx \frac{0.03 \times 43 \times 10^6}{0.6 \times 1005} \approx 855\,\mathrm{K},$$

resulting in a combustion chamber exit temperature $T_{cc} \approx 416 + 855 \approx 1271\,\mathrm{K}$.

Material Properties Table

Material	Density (kg/m^3)	Yield Strength (MPa)	Applications
PLA	1240–1260	50–70	Prototyping, non-structural parts.
ABS	1010–1030	40–50	Prototyping, low-stress functional parts.
PEEK	1320	90–100	High-performance prototyping.
Aluminum (AlSi10Mg)	2680	200–300	Compressor housing, lightweight parts.
Titanium (Ti-6Al-4V)	4430	900–1200	Compressor blades, high-stress parts.
Stainless Steel (316L)	8000	500–800	Structural components.
Inconel (IN718)	8190	1000–1300	High-temperature components.

Theoretical details :

1. Conservation of Mass

$$\rho_c V_c A_c = \rho_o V_o A_o$$

- ρ_c, ρ_o: Inlet and exit densities.
- V_c, V_o: Inlet and exit velocities.
- A_c, A_o: Inlet and exit cross-sectional areas.

2. Ideal Gas Law

At the inlet and exit:

$$P_c = \rho_c R T_c \quad \text{and} \quad P_o = \rho_o R T_o$$

- P_c, P_o: Inlet and exit pressures.
- T_c, T_o: Inlet and exit temperatures.
- R: Specific gas constant.

3. Momentum Equation

For steady flow:

$$P_c A_c + \rho_c V_c^2 A_c = P_o A_o + \rho_o V_o^2 A_o$$

This accounts for pressure forces and momentum flux.

To calculate the required mass flow rate for the given thrust we related all pressure, area, velocity, density and temperature with conservation of mass, momentum, energy and ideal gas equations.

4. Energy Equation (Including Heat Addition)

$$C_p(T_o - T_c) + \frac{V_o^2 - V_c^2}{2} = q$$

- C_p: Specific heat at constant pressure.
- q: Heat added per unit mass from combustion.

Simplified Case: Constant Pressure Combustion ($P_o = P_c$)

a) Velocity Relation

$$V_o = V_c \frac{A_c}{A_o} \frac{T_o}{T_c}$$

Derived from mass conservation and the ideal gas law.

b) Density Relation

$$\rho_o = \frac{P_c}{RT_o}$$

From the ideal gas law ($P_o = P_c$).

c) Energy Equation

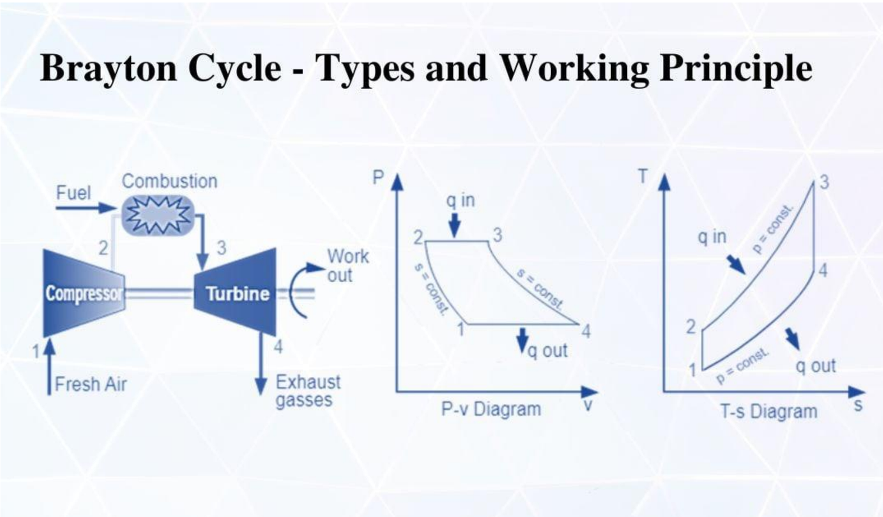
$$C_p(T_o - T_c) + \frac{V_o^2 - V_c^2}{2} = q$$

Solve for T_o if q is known.

7. Summary

Parameter	Equation
Exit Temperature	$T_o = \theta T_c$
Exit Velocity	$V_o = V_c \theta$
Exit Density	$\rho_o = \frac{\rho_c}{\theta}$
Temperature Ratio	$\theta = \frac{-C_pT_c + \sqrt{(C_pT_c)^2 + 2V_c^2(C_pT_c + \frac{V_c^2}{2} + q)}}{V_c^2}$

1. **Thermodynamic Cycle Analysis:**
   – Use a Brayton–Joule cycle simulator to compute the cycle parameters, ensuring maximum specific thrust.
   – Apply isentropic relations and energy balance equations to determine temperatures, pressures, and flow rates at compressor exit (station 2), combustion chamber exit (station 3), and turbine inlet (if applicable).



2. **Compressor Design:**
   – Start with one-dimensional approximations using:

$$\dot{m}_a = \rho A v,$$

   then refine the design through CFD analysis (e.g., using ANSYS CFX).
   – Design the impeller with radial blades, accounting for blade tip speed, using:

$$U_{\text{tip}} = \omega R_{\text{out}},$$

   and ensure proper diffuser design to convert rotational energy into static pressure.

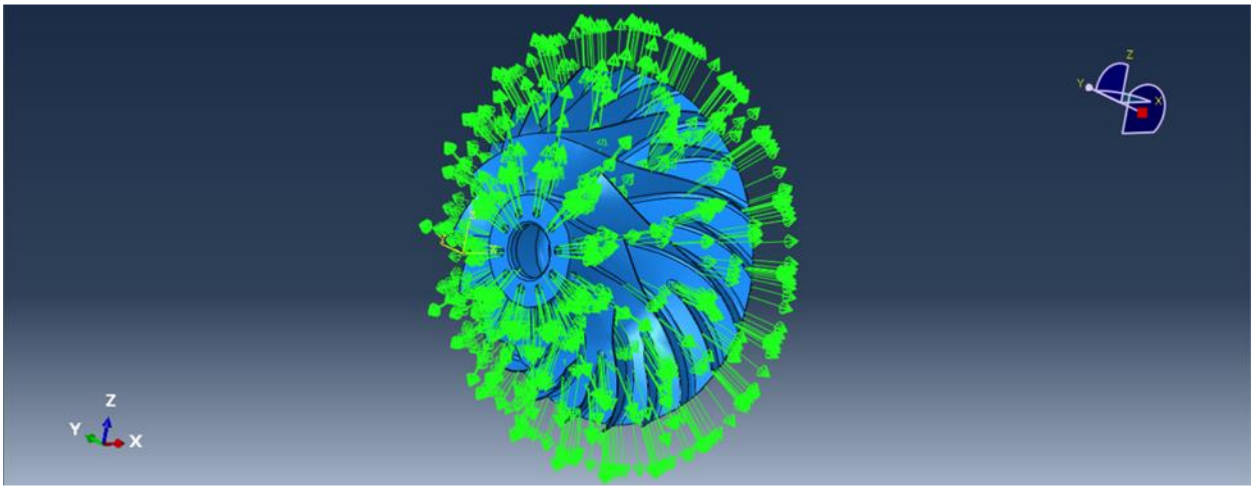
3. **Combustion Chamber and Diffuser Sizing:**
   – Design an annular combustion chamber with an optimized diffuser to minimize pressure losses.
   – Use empirical correlations (e.g., Aungier's loss correlations) to size the diffuser and predict pressure drop:

$$\Delta P = f\left(\frac{U_{\text{dif}}}{L}\right),$$

   where L is the chamber length and U_{dif} is the diffuser inlet velocity.
   – Determine the reference area A_{ref} of the combustion chamber using:

$$A_{\text{ref}} = \frac{\pi}{4}(D_{\text{out}}^2 - D_{\text{in}}^2).$$

- **Thermal Analysis with Ansys:**
  – Conduct thermal simulations to determine heat transfer coefficients and validate the temperature distribution in the combustion chamber.
  – Analyze the effect of lean fuel mixtures on flame stability and maximum operating temperatures.

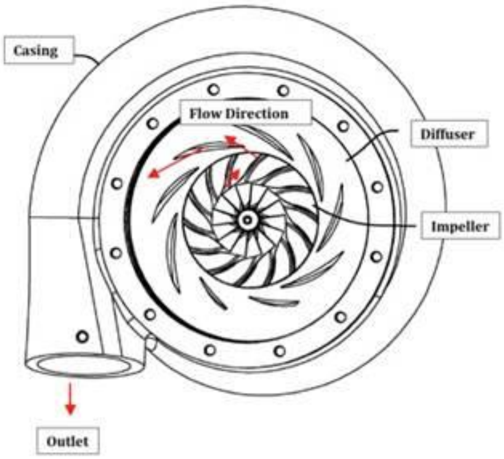


- **Manufacturing Considerations:**
  – We selected materials like PLA for compressor impeller and steel for the diffuser, considering machining capabilities and university-level fabrication resources.
  – Use CNC machining and metal 3D printing where applicable, ensuring tight tolerances (e.g., tip clearances around 0.2 mm) for high-speed components.

The Design:

Design of the Centrifugal compressor:

There are different types of compressors for different purposes, and for our requirement, two relevant compressor types are the Axial compressor and the Centrifugal compressor. Centrifugal compressors are used for low to medium pressure applications, smaller flow rates, and compact designs, and axial compressor is used for High-flow, high-efficiency applications where continuous high-speed air intake is needed. So, we have selected the centrifugal compressor finally as we are working on a micro-jet engine with a low to medium compression ratio and mass flow rate requirement. We decided to use single stage in our first iteration.



A centrifugal compressor contains three important components:

3) Impeller – Increases air velocity using rotating blades.
3) Diffuser – Converts kinetic energy into pressure by slowing down the flow.
3) Volute (Casing) – Collects and directs the compressed air efficiently to the next stage.

Designing of the Impeller:

We have two approaches to model the Impeller. First, if we already know the pressure, mass flow rate requirements and rpm of the motor, we can refer to the following book(Drive link: https://drive.google.com/file/d/1oO2kliFEsvatDqueC-3FjJUyfQSRKskw/view?usp=drive_link) and find out all the design parameters with their expressions given in the book. This is a very tedious task and does not suit our situation. So, we have opted for the second approach. That is, we download a standard model of the impeller with standard blade profiles, scale it up or down as per our requirements, find out all the important design parameters, and then calculate mass flow rate and compression ratio by changing the rpm. We have downloaded a standard model and imported it into SolidWorks.

We have fixed the maximum diameter of the jet engine as 15cm and max diameter of impeller as 10cm. So we have scaled downed the model such that max diameter becomes 10cm. Then we have measured the remaining design parameters of the impeller also:

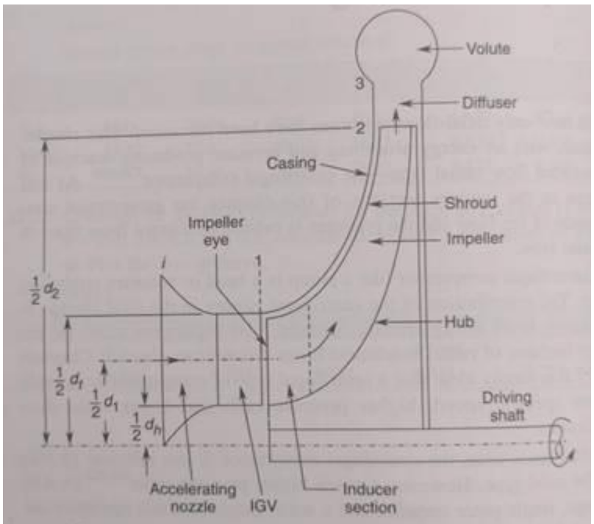


Fig. 12.1 Elements of a centrifugal compressor stage

The important design parameters of the impeller are:

r_1 = radius of blade at inlet = 0.0243m

r_2 = radius of blade at inlet = 0.0497m

r_t = Maximum inlet radius = 0.0319m

r_d = Minimum Inlet Radius = 0.0166m

b_1 = Blade width at inlet = 0.0152m

b_2 = Blade width at outlet = 0.00535m

t_1 = Thickness of blade = 0.000223m

t_2 = Thickness of Flow Separators = 0.000891m

n_1 = Number of blades = 11

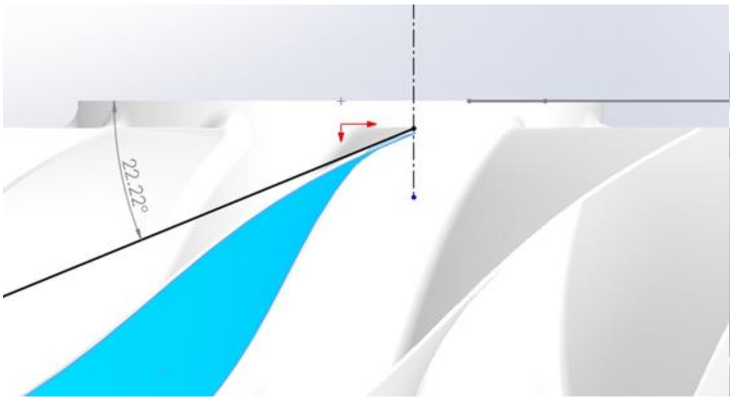
n_2 = Number of Flow separators = 11

$Alpha_1$ = Absolute velocity of air at inlet

$Alpha_2$ = Absolute velocity of air at outlet

$Beta_1$ = Relative velocity of air at inlet = 22.22 degrees

$Beta_2$ = Relative velocity of air at outlet = 30.05 degrees



Beta$_1$ Beta$_2$

Theory related to the compressor:

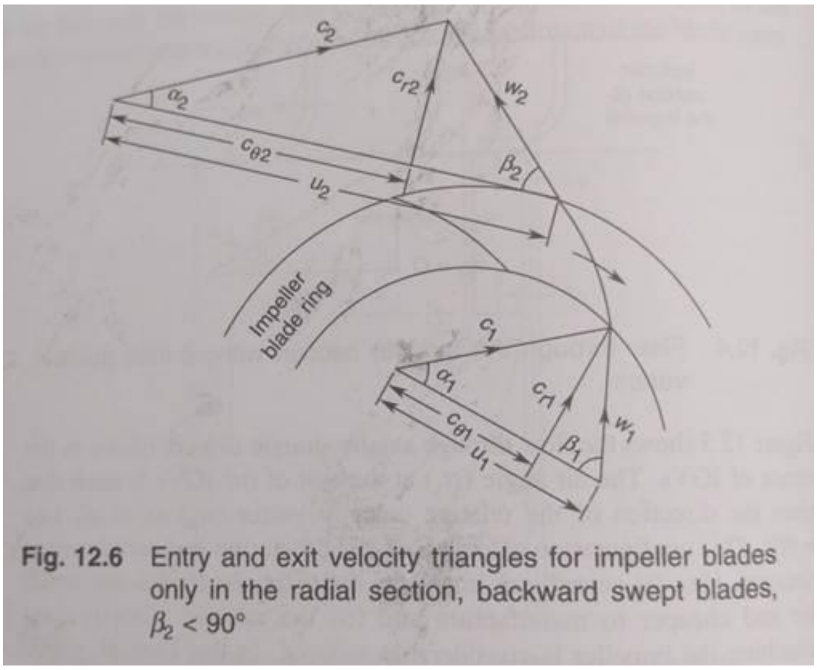


Fig. 12.6 Entry and exit velocity triangles for impeller blades only in the radial section, backward swept blades, $\beta_2 < 90°$

The above figure shows the velocity triangles of the impeller at inlet and exit. The velocity triangle at inlet will be located in a plane's normal which is perpendicular to the axis of the impeller. Whereas at exit, the velocity triangle is present in a plane perpendicular to the axis of impeller.

Relationship between different Velocities and derivation of mass flow rate and pressure ratio with them:

$$U_1 = \frac{2\pi N r_1}{60} \rightarrow \text{blade tipvelocity at inlet}$$

$$U_2 = \frac{2\pi N r_2}{60} \rightarrow \text{blade tip velocity at exit}$$

$$W_1 = \frac{C_{r1}}{\sin\beta_1} \rightarrow \text{relative velocity of air at inlet}$$

$$W_2 = \frac{C_{r2}}{\sin\beta_2} \rightarrow \text{relative velocity of air at exit}$$

$$C_{r1} = \frac{\dot{m}}{\rho_{in}(2\pi r_1 b_1)} \rightarrow \text{Radial velocity components at inlet.}$$

$$C_{r2} = \frac{\dot{m}}{\rho_{out}(2\pi r_2 b_2)} \rightarrow \text{Radial velocity component at exit.}$$

$$\left.\begin{aligned} C_{\theta 1} &= U_1 - W_1 \cos\beta_1 \\ C_{\theta 2} &= U_2 - W_2 \cos\beta_2 \end{aligned}\right\} \text{Tangential velocity Components}$$

$$\left.\begin{aligned} C_1 &= \sqrt{C_{\theta 1}^2 + C_{r1}^2} \\ C_2 &= \sqrt{C_{\theta 2}^2 + C_{r2}^2} \end{aligned}\right\} \text{Absolute velocities.}$$

$$\alpha_1 = \tan^{-1}\left(\frac{C_{r1}}{C_{\theta 1}}\right)$$

$$\alpha_2 = \tan^{-1}\left(\frac{C_{r2}}{C_{\theta 2}}\right)$$

From stannitz formula, $\underbrace{\sigma}_{\text{Stannitz}} = 1 - \frac{(0.63)(\pi)}{\underbrace{n}_{\text{number of blades}}}$

Slip in a centrifugal compressor refers to the reduction in tangential velocity of the air at the impeller exit due to fluid dynamics effects. It occurs because the air does not perfectly follow the motion of the impeller blades and instead lags behind. This is primarily caused by flow separation, friction, and secondary flows inside the impeller passages. The slip factor, which quantifies this effect, is typically less than 1 and reduces the actual work done on the air. As a result, the pressure rise and efficiency of the compressor are slightly lower than ideal predictions.

At exit, $C_{\theta 2} = \sigma U_2 \rightarrow$ Slip factor.

from euler's turbine equation, $\Delta h = U_2 C_{\theta 2} - U_1 C_{\theta 1}$

Assumption : $C_{\theta 1} \approx 0$ (axial entry assumption).

$$\Rightarrow \quad \Delta h = U_2 C_{\theta 2}$$

$$\Delta T = \frac{\Delta h}{C_p}$$

$$T_{out} = T_{in} + \Delta T_{out} \quad \cancel{\Delta T_{in}}$$

$$r_c = \left(1 + \frac{\eta \Delta h}{C_p T_{in}}\right)^{\frac{\gamma}{\gamma - 1}}$$

from Continuity, $\dot{m} = \rho A_2 C_{r2} \rightarrow$ radial component.

$$\tan\beta_2 = \frac{C_{r2}}{\cancel{\sigma}\, U_2 - C_{\theta 2}} \Rightarrow C_{r2} = (1-\sigma) U_2 \tan\beta_2$$

$$\Rightarrow \quad \boxed{\dot{m} = (1-\sigma)\rho A_2 U_2}$$

$$V_{in} = \frac{\dot{m}}{\rho_{in} \cdot \pi (r_t^2 - r_h^2)}, \quad V_{out} = \frac{\dot{m}}{2\pi r_2 b_2 \rho_{out}}$$

The Power required can be calculated as:

$$\text{Power Consumed} = \dot{m}\,(\text{work done per unit mass})$$
$$= \dot{m} U_2 C_{\theta 2}$$
$$= \sigma \dot{m} U_2^2$$

For Easier calculations, we have put all the formulas and expressions into a Python script file(Drive link: https://drive.google.com/file/d/1pnz5lT6474Bi9Py3sC91GTbF7lxP1Z_g/view?usp=drive_link).

At 50000 rpm, these are the output results:

```
RPM of Motor : 50000 rpm
Power : 4771.071492157195 Watt
Torque : 0.9112075341859711 N-m
mass flow rate : 0.06867072438160561 Kg/s
Total Pressure Ratio : 1.3685269310279065
Velocity of blade tip at outlet of impeller : 260.3400824045053 m/s
Velocity of blade tip at intlet of impeller : 127.25143041296452 m/s
Area of exit : 0.0016717490241359538 m^2
Velocity of Inlet : 24.855086568957812 m/s
Velocity of exit : 26.67348915703552 m/s
Absolute air inlet angle: 0.3627596236739675 degrees
Absolute air inlet angle: 0.1729267486406216 degrees
```

The theoretical formulas may not give accurate results because of the following reasons:

- The used theory did not take viscous and turbulence losses into account.
- At speeds near to Mach speeds, there will be different type of losses which we did not consider.
- Actually, the efficiency of a centrifugal compressor shows a peak when plotted against either mass flow rate or compression ratio. But we have considered it as constant.
- We don't know the operating point of the compressor.

So, the only way we find the operational point is to plot the performance curve for the compressor.

Theory related to the Performance curves and non-dimensional numbers related to the centrifugal compressor:

Compressor Performance characterstics:-

Output Parameters:-

$$\left.\begin{array}{c}\dfrac{T_{3t}}{T_{1t}} \\[2ex] \dfrac{P_{3t}}{P_{1t}}\end{array}\right| \; f[N, D, \dot{m}]$$

$$F(N, D, \dot{m}, P_{1t}, P_{3t}, RT_{1t}, RT_{3t}) = 0$$

Density is not included because it is just $\frac{P_{1t}}{RT_{1t}}$ (or) $\frac{P_{3t}}{RT_{3t}}$

It is included implicitly. fundamental dimentions = 3 (MLT)

No. of π terms = 7-3=4

repeating variables = D, P_{1t}, RT_{1t}

$$\pi_1 = \frac{ND}{\sqrt{RT_{1t}}}, \quad \pi_2 = \frac{\dot{m}\sqrt{RT_{1t}}}{D^2 P_{1t}}, \quad \pi_4 = \frac{T_{3t}}{T_{1t}}, \quad \pi_3 = \frac{P_{3t}}{P_{1t}}$$

$$\Rightarrow F(\pi_1, \pi_2, \pi_3, \pi_4) = 0$$

Physical significance of π_1: $\quad N.D \propto U_1$

$\sqrt{RT_{1t}} \propto a$

$$\pi \propto \frac{U}{a} = M_R$$ (Mach number based on rotor speed)

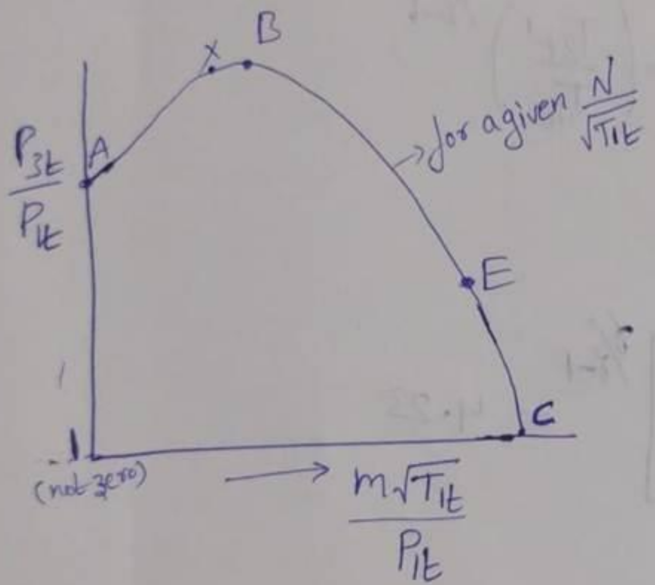
Physical significance of Π_2 :-

$$\frac{\dot{m}\sqrt{RT_{1t}}}{D^2 P_{1t}} = \frac{\rho A V_f \sqrt{RT_{1t}}}{D^2 P_{1t}}$$

$$\frac{P_{1t}}{\rho} \sim RT_{1t} \quad \propto \frac{AV_f}{D^2\sqrt{RT_{1t}}} \propto \frac{V_f}{a} = M_f$$ (Mach number based on flow speed).

$$\phi\left(\frac{N}{\sqrt{T_{1t}}}, \frac{m\sqrt{T_{1t}}}{P_{1t}}, \frac{P_{3t}}{P_{1t}}, \frac{T_{3t}}{T_{1t}}\right) = 0$$

→ for a given machine, for a given gas as D, R becomes Constants once we fix D, R for a machine



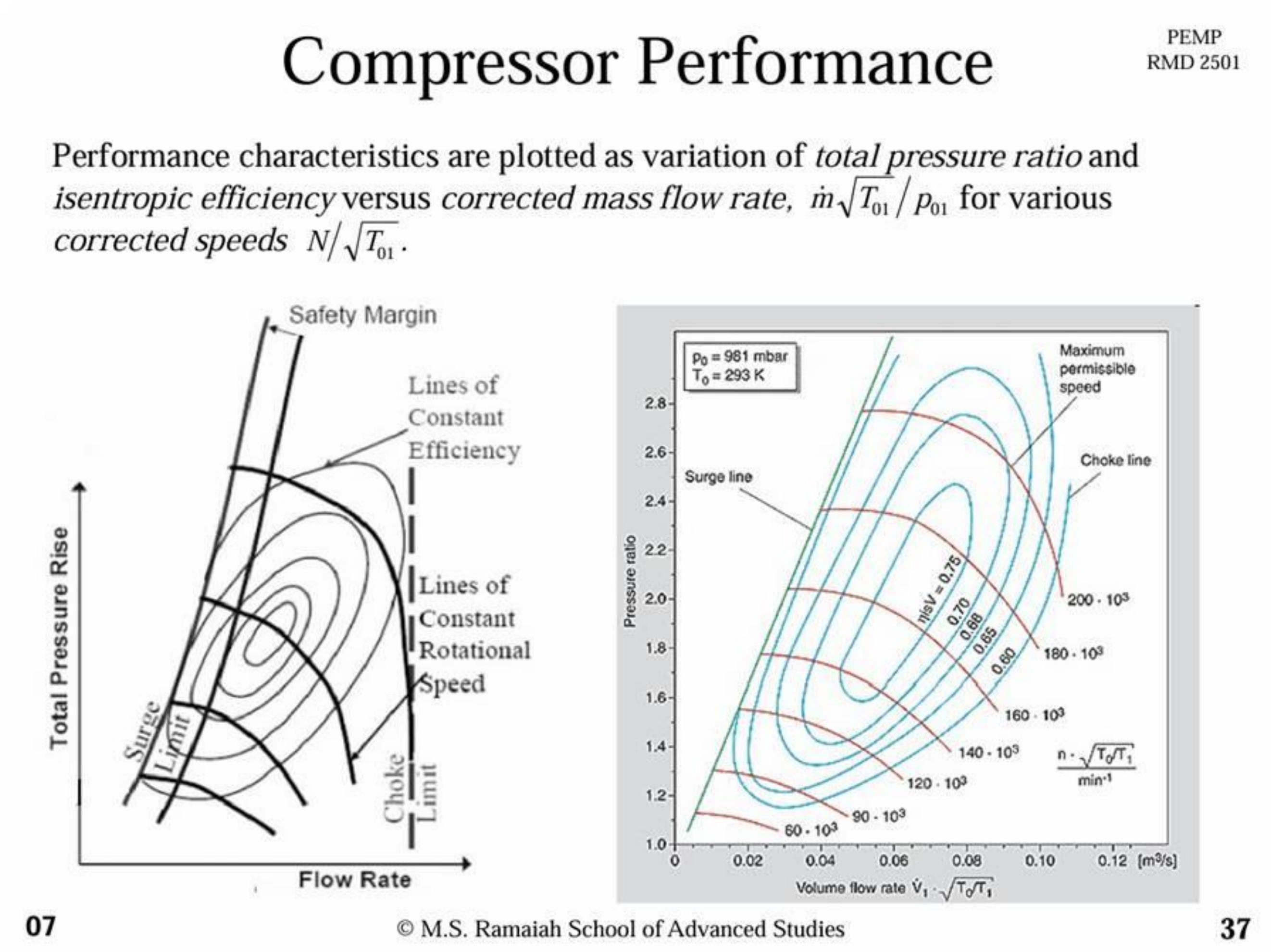
Operating point should be from A to ~~X~~ as it is unstable. Surging happens.

beyond E, choking occurs.

two performance curves required are:

P_{3t}/P_{1t} (vs) $\frac{\dot{m}\sqrt{T_{1t}}}{P_{1t}}$ at different $\frac{N}{\sqrt{T_{1t}}}$

T_{3t}/T_{1t} (vs) $\frac{\dot{m}\sqrt{T_{1t}}}{P_{1t}}$ at different $\frac{N}{\sqrt{T_{1t}}}$



The performance curves are compressor specific and we have to plot our own performance curve. We can follow any one of the two methods to plot it without performing the actual experiment:

- Method-1: To perform CFD analysis by varying different parameters and manually plotting the points or using inbuilt tools to directly plot the performance curve.
- Method-2: To refer the following theory(Drive link: https://drive.google.com/file/d/10lcSBEsbWEAHzYt4L-jiFkszAyGRLXum/view?usp=drive_link) and theoretically find the performance curve.

That will be our upcoming work regarding compression.

Design of the Diffuser: The diffuser decelerates the air by decreasing its kinetic energy and converting it into static pressure.

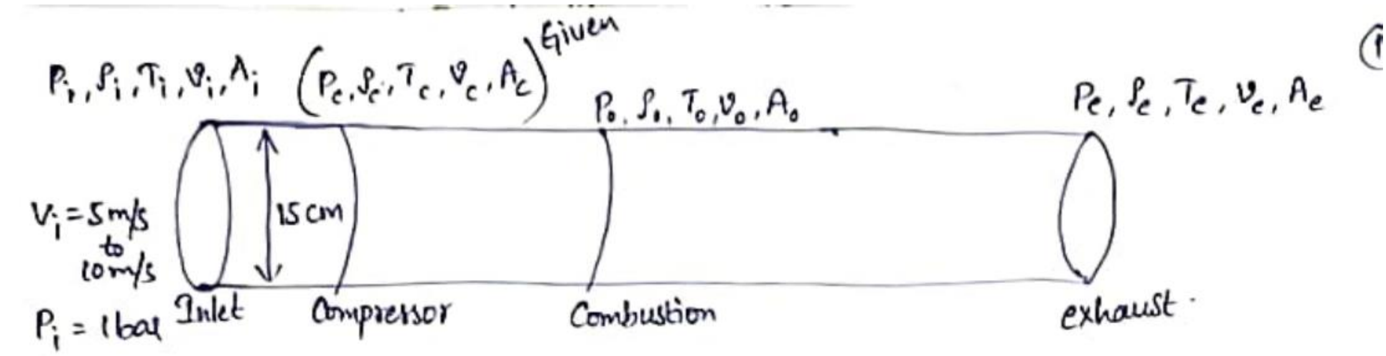
For optimal performance of the diffuser, the velocity of the air at the outlet should be equal to the inlet velocity of the compressior eye. The only important dimension inside a diffuser is an area of exit and the vane angles and dimensions. Once the operational point of the impeller is known, these two parameters can be calculated.

Coming to the Casing, it just follows the outer blade profile of the impeller with tight tolerance.

COMBUSTION CHAMBER

THERMODYNAMIC RELATIONS AMOUNG (P, Velocity, T, Area, Density)

(1)



$P_i = 1\,bar$

$T = 300\,K$ (or) $27^\circ C$

Assumptions: $A_0 = A_c$

Isobaric process in combustion chamber

i.e, $P_0 = P_c$

Inlet: $P_c, T_c, V_c, \rho_c, A_c$ (These values we will get from compressor outlet)

Exit: $P_0 = P_c$, $A_0 = A_c$

Heat added $(q) = \eta_c \cdot LHV \cdot FAR$

LHV ↓ latent heat of vapourisation

FAR ↳ Fuel-Air ratio

By using conservation of mass

$$\dot{m}_{in} = \dot{m}_{out}$$

$$\rho_c V_c A_c = \rho_0 V_0 A_0 \qquad [A_0 = A_c]$$

$$\rho_c V_c = \rho_0 V_0$$

$$\boxed{V_0 = \frac{\rho_c V_c}{\rho_0}} \rightarrow (1)$$

$$\cancel{V_0 = \rho_c V_c}$$

$$\rho_c A_c V_c = \rho_0 A_0 V_0$$

$$\rho_c A_c V_c = \left(\frac{P_c}{R T_0}\right) A_0 V_0$$

$$\rho_c \times \frac{A_c}{A_0} \times V_c \times \frac{R T_0}{P_c} = V_0$$

$$\frac{A_c}{A_0} \times V_c \times \frac{\rho_c R T_0}{P_c} = V_0$$

$$\boxed{\frac{A_c}{A_0} \times V_c \times \frac{T_0}{T_c} = V_0} \rightarrow (2)$$

Ideal Gas law:

$$P_c = \rho_c R T_c$$

$$P_0 = \rho_0 R T_0$$

R = specific gas constant.

$R_{air} = 287.7$

$R_{fuel} = 350$

$R_{fuel\ mixture} = 288.1$

$P_c = \rho_c R T_c$

$$P_c = P_0 \Rightarrow \rho_c \cancel{R} T_c = \rho_0 \cancel{R} T_0$$

$$\rho_0 = \frac{\rho_c T_c}{T_0}$$

$$\boxed{\rho_0 = \frac{P_c}{R T_0}} \rightarrow 3$$

from eq 2 $\Rightarrow \frac{A_c}{A_0} \times V_c \times \frac{T_0}{T_c} = V_0$

we know $\frac{A_c}{A_0} = 1$

So, $\boxed{V_0 = V_c \times \frac{T_0}{T_c}} \rightarrow 4$

from 1st law of Thermodynamics:

$$\Delta U = \dot{Q} - \dot{W} = \dot{m}\left(h_2 - h_1 + \frac{V_2^2}{2} - \frac{V_1^2}{2} + g(z_2 - z_1)\right).$$

$$\dot{Q} = \dot{m} C_p (T_0 - T_c) + \left(\frac{V_0^2 - V_c^2}{2}\right) \quad \left(\begin{matrix}\text{work done by} \\ \text{turbine} = 0\end{matrix}\right).$$

$$q = C_p (T_0 - T_c) + \left(\frac{V_0^2 - V_c^2}{2}\right)$$

By substituting eq-4 $\left(V_0 = V_c \times \frac{T_0}{T_c}\right)$

$$q = C_p (T_0 - T_c) + \frac{V_c^2}{2}\left(\frac{T_0^2}{T_c^2} - 1\right)$$

$$q = C_p T_c (\theta - 1) + \frac{V_c^2}{2}(\theta^2 - 1) \qquad \left[\theta = \frac{T_0}{T_c}\right)$$

$$q = C_p T_c \theta - C_p T_c + \frac{V_c^2}{2}\theta^2 - \frac{V_c^2}{2}$$

$$\frac{V_c^2}{2}\theta^2 + C_p T_c \theta - \left(C_p T_c + \frac{V_c^2}{2} + q \right) = 0$$

$$\theta = \frac{-C_p T_c \pm \sqrt{(C_p T_c)^2 + 4\left(\frac{V_c^2}{2}\right)\left(C_p T_c + \frac{V_c^2}{2} + q\right)}}{V_c^2}$$

Conservation of momentum:

$$P_c A_c + \rho_c V_c^2 A_c = P_o A_o + \rho_o V_o^2 A_o$$

Exit Temperature : $T_o = \theta T_c$

Exit velocity : $V_o = V_c \theta$

Exit Density : $\rho_o = \rho_c \frac{T_c}{T_o} = \frac{\rho_c}{\theta}$

Combustion Chamber:

Introduction:

The combustion chamber (Burner) is one of the most important components in the gas turbine engine. The combustion process is happened in there. The process increases and converts high-pressure airflows into high speed airflows. This component is located after the compressor and before the turbine. The component is consists of casing and liner.

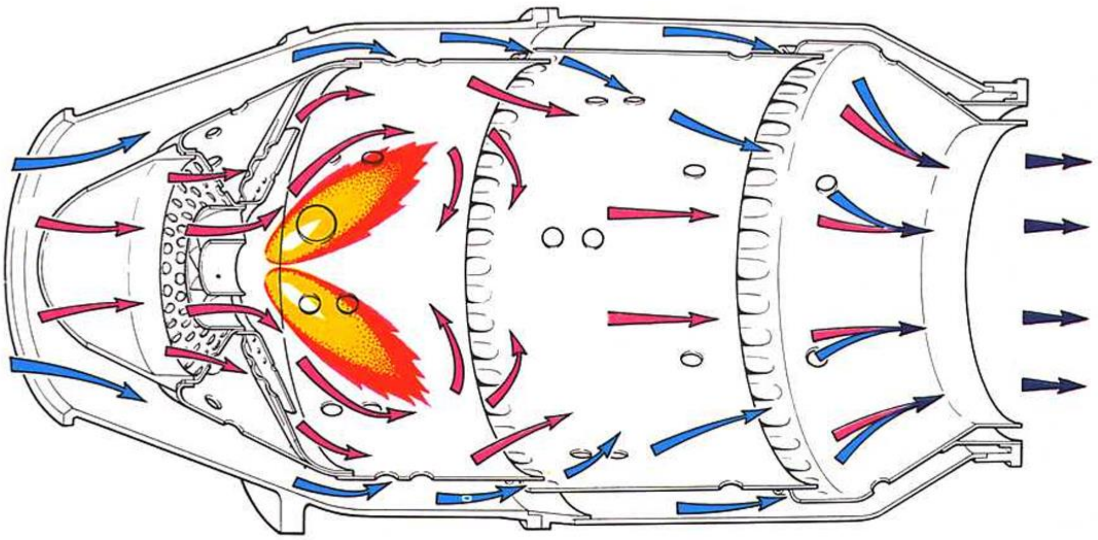


Fig. 4-3 Flame stabilizing and general airflow pattern.

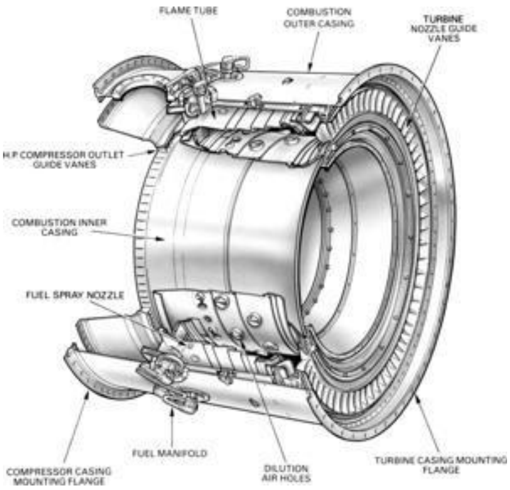
2. Design Procedures

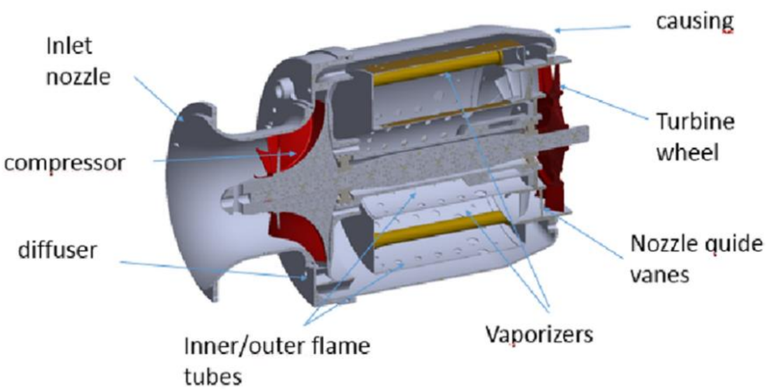
There are several procedures to design a combustion chamber. Melconian and Modak also Arthur H. Lefebvre are well-known ones. In this report, the preliminary design procedures follow Melconian and Modak. Below is the design procedures.

1. Design specification.
2. Select the combustion chamber type.
3. Select diffuser types.
4. Determine diffuser sizing.
5. Determine reference values and pressure drop parameters.
6. Determine liner sizing.
7. Determine airflow distribution.
8. Determine the size of the orifices (holes)

Combustion chamber type Selection:

For a micro turbojet engine, we need as small as possible combustion chamber dimension. From all of the combustion chamber type, the annular type is the best because of its minimum cross section area, length, and weight.





INPUTS :

- ma = Air mass flow rate
- Tt3 = Comp. exit total temp.
- pt3 = Comp. exit total press.
- mf = Fuel mass flow rate
- Tt4 = Turb. inlet total temp.

- pt4 = Turb. inlet total press.

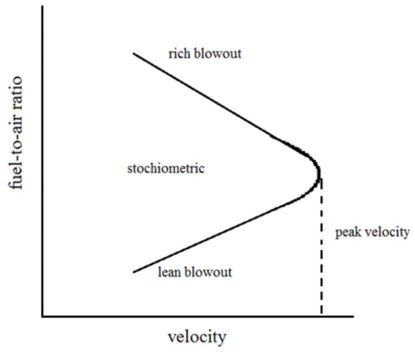


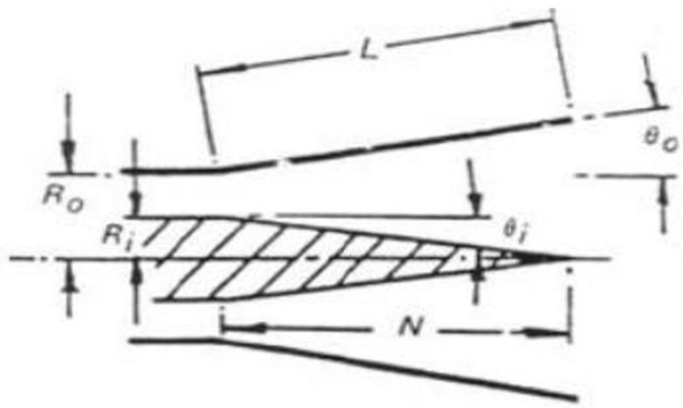
Figure 8: Fuel-air ratio vs velocity.

-

2.4. Diffuser Types Selection:

There are two types of the diffuser, which are aerodynamic or faired and annular dump diffuser. The aerodynamic gives low-pressure loss. However, it has a relatively long length, its performance susceptible to thermal distortion and manufacturing tolerances, and its performance and stability are sensitive to variation in inlet velocity. On the other hand, the dump diffuser has a relatively shorter length and insensitive to variations in inlet flow conditions. But it has higher pressure loss (about 50% higher than the aerodynamic type). In this preliminary design, the annular dump diffuser likely the fittest for this micro turbojet combustion chamber due to its relatively short and insensitive to variations in inlet flow conditions.

There are many types of annular dump diffuser. The best type of annular diffuser to use in this combustion chamber design is the equiangular annular types.



The diffuser design geometric parameters usually depend on its aspect ratio (AR), wall or axial length (L or N), divergence angle (θ) which usually between 7o and 12o. To find the aspect ratio, we can use the below equation.To

$$AR = 1 + 2\frac{L}{\Delta R}\frac{\sin\theta_i + (R_i/R_o)\sin\theta_o}{1 + R_i/R_o}$$
$$+ \frac{(L^2/\Delta R^2)(1 - R_i/R_o)(\sin^2\theta_i - \sin^2\theta_o)}{1 + R_i/R_o}$$

Start the design process, we use the geometry of the compressor exit (inner diameter and outer diameter as Ri and Ro). The wall or axial length becomes a length constraint on the combustion chamber. The divergence angle is chosen between the usual values. Then, put all the variables value to eq. (1) and solve for AR. After that, check the AR on the lines of the first stall graph. Make sure the AR from sizing is under the lines of the first stall.

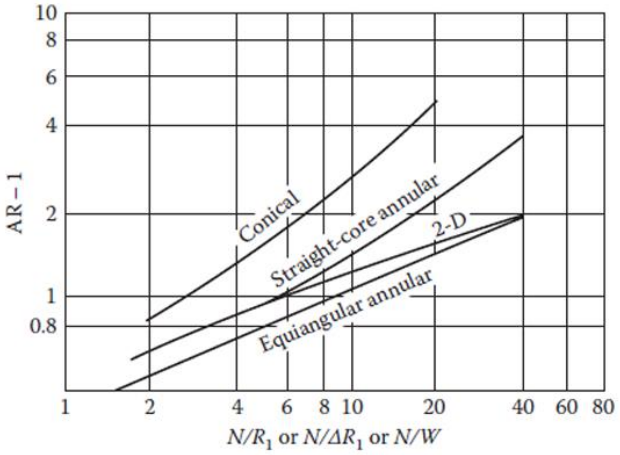


Figure 2 Lines of First Stall

Input

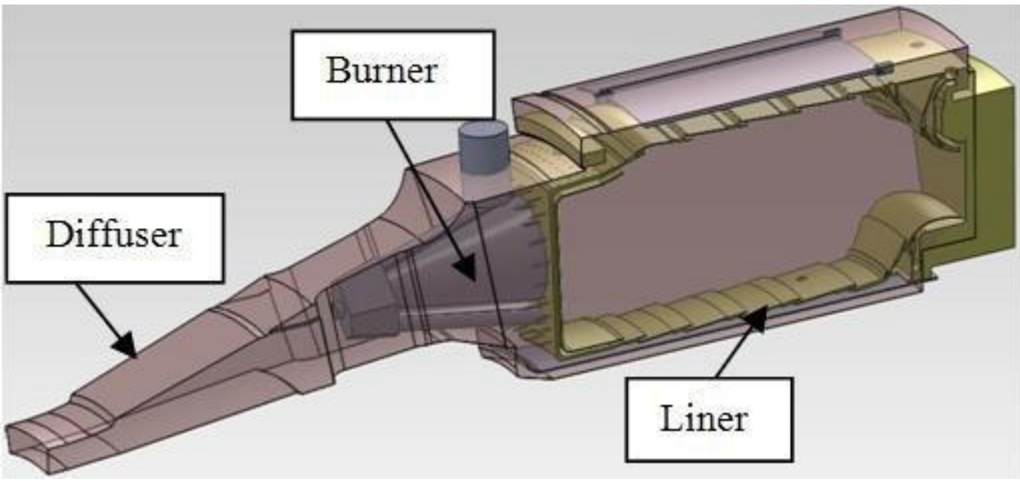
- Dout = Outer exit comp. dia.
- Din = Inner exit comp. dia.
- θo = Outer divergence angle
- θi = Inner divergence angle
- L = Wall length

Output

- AR = Aspect Ratio
- N = Axial length

After solving eq. (1), the AR value is 3.60. However, this sizing point lies above the lines of the first stall which indicates the flow is stalled. The stalled flow makes the pressure loss bigger and could not hold the flow stability. To lower the sizing point on the lines of the first stall graph, the wall length or divergence angle can be varied. Due to the limitation of divergence angle length, the wall length can be chosen to be varied. However, the wall length directly affects the length of the combustion chamber length. From this point, this AR still can be accepted as the sizing point, as a consequence, the diffuser will generate bigger pressure loss. Further, below is the table of additional variables.

- Pt3 = Exit comp. total density
- Udi-inle = Diffuser inlet velocity
- Udif-outle = Diffuser outlet velocity



2.6. Reference values and Pressure Drop :

The main combustion chamber sizing is started by determining the reference values including reference area, velocity, and dynamic pressure. Burner compression ratio (ratio of pt4 and pt3 or πB), air mass flow, and geometry of compressor exit and turbine inlet become the input of calculation.

Reference values : The reference area (Aref) is defined as the maximum cross-section area of casing exclude the liner. For the annular combustion chamber, the reference area can be obtained by the following eq. (2) below.

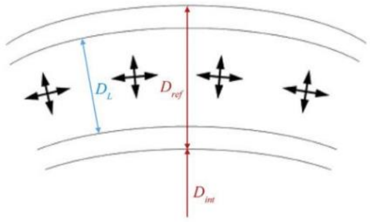


Figure 3 Geometry of Reference Area

Equation 2

$$A_{ref} = \frac{\pi}{4}\left[\left(2D_{ref} + D_{\text{int}}\right)^2 - \left(D_{\text{int}}\right)^2\right] \tag{2}$$

The reference diameter has the same size as the casing diameter. Further, the internal diameter is user-defined. We design the reference area thus we have the flow Mach number around 0.03 or below. The other reference values are velocity and dynamic pressure. Use the following equation to find reference velocity and reference dynamic pressure.

$$U_{ref} = \frac{\dot{m}_a}{\rho_{t3} A_{ref}}$$

$$q_{ref} = \frac{1}{2}\rho_{t3} U_{ref}^2$$

Input

- Dref = Ref. diameter
- Dint = Int. diameter

Output

- Aref = Ref. area
- Uref = Ref. velocity
- aref = Ref. speed of sound
- Mref = Ref. mach number
- qref = Ref. dynamic pressure

Pressure loss parameters calculation :

In the combustion chamber design process, there are two sources of pressure loss, from diffuser and liner (a place where the combustion process occurs). We can derive the pressure loss parameters by following the equation below. To find the pressure loss on the liner, we need to obtain the pressure loss on the diffuser. This can be accomplished by solving eq. (7) and eq. (8).

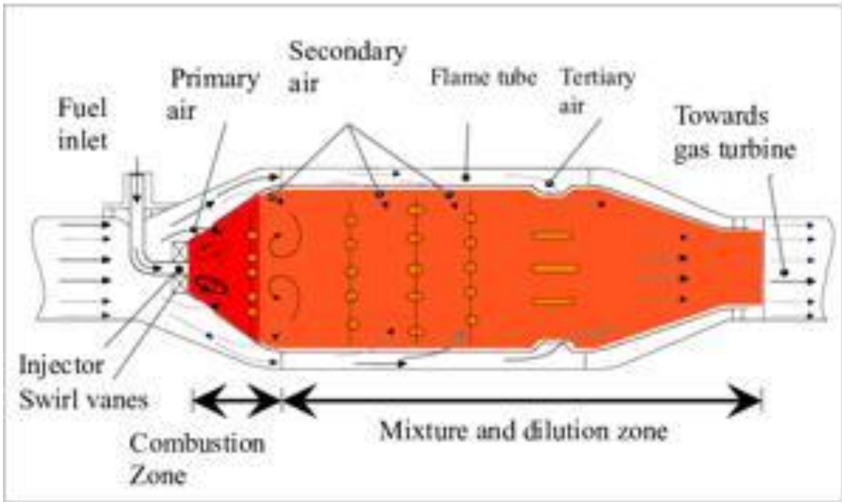
$$\pi_B = \frac{p_{t4}}{p_{t3}} = 1 - \frac{\Delta p_{t,3-4}}{p_{t3}} \tag{5}$$

$$\frac{\Delta p_{t,3-4}}{q_{ref}} = \frac{\Delta p_{t,diff}}{q_{ref}} + \frac{\Delta p_{t,L}}{q_{ref}} \tag{6}$$

$$\Delta p_{t,diff} = \bar{q}_1 \lambda \left(1 - \frac{1}{AR^2}\right) \tag{7}$$

$$\bar{q}_1 = \frac{1}{2} \rho_{t3} \overline{U}^2_{in,diff} \tag{8}$$

- $\Delta p_{t,3-4}/p_{t3}$ = Combustor press. loss
- $\Delta p_{t,3-4}/q_{ref}$ = Combustor press. drop factor
- $U_{in,diff}$ = Avg. velocity in diffuser
- λ = Dump diffuser constant
- $\Delta p_{t,diff}$ = Total press. loss in diffuser
- $\Delta p_{t,diff}/q_{ref}$ = Diffuser press. drop factor
- $\Delta p_{t,L}/q_{ref}$ = Liner press. drop factor
- $\Delta p_{t,L}$ = Total press. loss in liner



2.7. Liner sizing :

Liner is the place that undergoes the combustion process. The first step is to find the liner area by solving the equation below.

$A_L = kA_{ref}$ (9)

The constant k usually around 0.66 – 0.70. After finding the liner area, then we can calculate the liner diameter following eq. (2). Further, the liner length depends on liner pressure drop and allowable maximum temperature at the combustion chamber outlet. The equation is

$$L_L = D_L\left(A\frac{\Delta p_{t,L}}{q_{ref}} \ln\frac{1}{1-PF} \right)^{-1} \qquad (10)$$

Where

$$PF = \frac{T_{t,\max} - T_{t4}}{T_{t4} - T_{t3}} \qquad (11)$$

The liner is consists of three zones, primary, secondary, and dilution, respectively. The rule of thumb to calculate each zones length is

$$L_{PZ} = \frac{2}{3} \sim \frac{3}{4} D_L \qquad (12)$$

$$L_{SZ} = \frac{1}{2} D_L \qquad (13)$$

$$L_{DZ} = L_L - L_{PZ} - L_{SZ} \qquad (14)$$

In this preliminary design, we choose k equal to 0.70 to give the bigger size of the liner area, thus the air velocity inside the liner decreases. Then, we set the outlet maximum temperature to be at 1,210 K to avoid thermal stress failure on the material. Following eq. (9) to eq. (14), we will have our liner sizing calculation as listed

- K = Liner to reference area ratio
- DL,in = Inner liner dia.
- DL,out = Outer liner dia
- Tt,max = Maximum outlet local temp.

Output

- DL = Liner dia. or flame tube height
- PF = Pattern factor -
- LL = Liner length
- LPZ = Primary zone length
- LSZ = Secondary zone length
- LDZ = Dilution zone length

2.8. The Airflow Distribution:

Calculation of the required airflow rate by each zone is of prime importance for combustor design since the number location and size of the holes for a specific zone will be then calculated once the required

airflow rate is determined. Primary zone total airflow rate :
The total airflow needed for the primary zone can be calculated using the following formulae

$$\dot{m}_{PZ} = 14.77 \times \alpha_{PZ} \times \dot{m}_f \qquad (15)$$

In this zone, we need to ensure that the fuel is burned properly. To do that, in this zone, we will make a fuel-rich composition. Thus, the air to fuel ratio is set below 1.0. Further, the airflow could enter the primary zone through annulus or axially. We could set the percentage of airflow here.

Secondary zone total airflow rate :

Similar to the primary zone total airflow rate calculation, it can be calculated using the following formulae

$$\dot{m}_{SZ} = 14.77 \times \alpha_{SZ} \times \dot{m}_f \qquad (16)$$

In contrast to the primary zone, in this zone, we want to burn the fuel efficiently. Thus, we will make it lean composition. The air to fuel ratio is set more than 1.0. Moreover, the only way the airflow enters the secondary zone is through the annulus. Hence we do not need any flow division.

Dilution zone total airflow rate:

The dilution zone total airflow rate can be calculated as the rest of the airflow that still exists after being taken in the primary and secondary zone. Thus to calculate the dilution zone total airflow rate, we can use the following equation.

$$\dot{m}_{DZ} = \dot{m}_a - \left(\dot{m}_{PZ} + \dot{m}_{SZ}\right) \qquad (17)$$

- ma = Air mass flow
- mf = Fuel mass flow
- αPZ = Primary zone AFR
- αSZ = Secondary zone AFR
- kPZ = Ratio of airflow that enter axially to total in primary zone
- mPZ,ax = Primary zone airflow - axial
- mPZ, ann = Primary zone airflow – annular
- mSZ = Secondary zone airflow
- mSZ,jet = Secondary zone airflow jet
- mDZ = Dilution zone airflow
- mDZ,jet = Dilution zone airflow – jet

2.9. Sizing Liner Holes:

The need for the liner holes is to provide enough air for every zone. The liner wall contains a row of n holes, each of which has an effective diameter dj. Then to calculate the size of the holes, we can use the following equation.

$$\dot{m}_j = \frac{\pi}{4} n d_j^2 \rho_{t3} U_j \quad (18)$$

$$U_j = \sqrt{\frac{2\Delta p_{t,L}}{\rho_{t3}}} \quad (19)$$

$$\frac{Y_{\max}}{d_j} = 1.25 \frac{\dot{m}_g}{\dot{m}_g + \dot{m}_j} \sqrt{\frac{\rho_j U_j^2}{\rho_g U_g^2}} \quad (20)$$

$$d_h = \frac{d_j}{\sqrt{C_D}} \quad (21)$$

Primary Zone

- dj,ax = Holes diameter, axial
- nax = A half no. of holes, axial
- dj,ann = Holes diameter, annular
- nann,1 = A half no. of holes, axial, row-1
- nann,2= A half no. of holes, axial, row-2

Secondary Zone

- dj,SZ = Holes diameter
- nSZ,1 = A half no. of holes, row-1
- nSZ,2 = A half no. of holes, row-2

Dilution Zone

- dj,DZ Holes diameter
- nDZ A half no. of holes

The hole's diameter and its number can be obtained by solving eq. (18). To find the effective diameter, we have to solve eq. (19) and (20) respectively. The Ymax is a maximum penetration depth (a ratio to liner diameter). Once the Ymax has decided, then the effective diameter can be obtained. After that, we have to find the real hole's diameter by using eq. (21). CD is the discharge coefficient.

We use typical value of CD = 0.6. After solving eq. (18) to eq. (21), we will get the liner holes size as listed above.

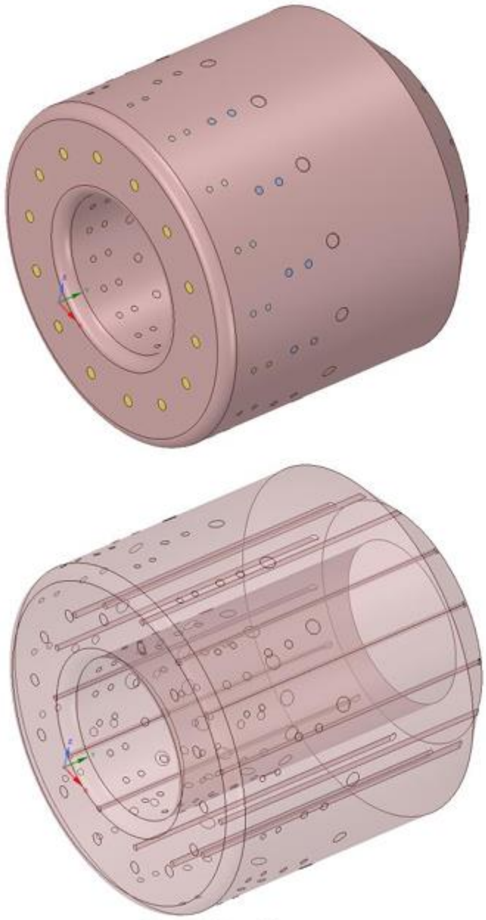


Figure 6 Liner Geometry

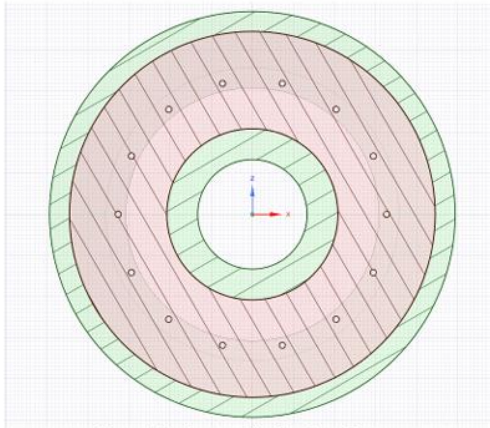


Figure 4 Combustion Chamber Cross-Section

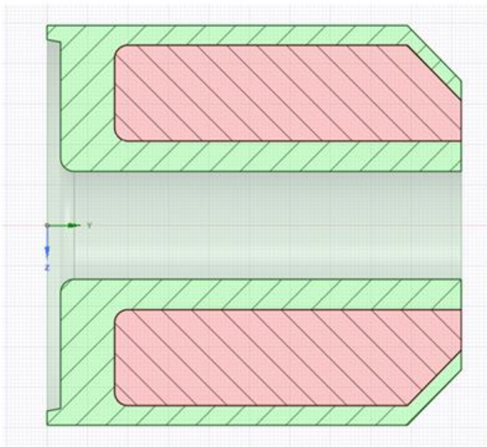


Figure 5 Combustion Chamber Meridian Cut

Result and observation:

Based on our finalized design methods and calculations, our approach indicates that a lean mixture (FAR ~1:50) should maintain combustion chamber temperatures in the range of 1200–1300 K. Although CFD simulations have not yet been performed, our analytical methods predict a uniform temperature distribution and an acceptable pressure drop (around 5% loss) across the annular combustion chamber. Our compressor design calculations indicate that a compression ratio of approximately 1.5, with an impeller tip speed around 260 m/s at 50,000 rpm, should provide adequate air mass flow 0.0686kg/s to achieve the target thrust. Future CFD analyses and experimental bench tests will be used to verify and refine these design parameters.

Conclusion:

The detailed technical analysis demonstrates that designing a micro jet engine with 300 N thrust is feasible using established thermodynamic principles and advanced simulation tools. The integrated approach—combining cycle analysis, CFD for compressor and combustion chamber design, and rigorous thermal analysis—ensures that key performance targets are met while maintaining material and manufacturing constraints suitable for a university-level project. Although challenges remain in achieving precise component tolerances and effective heat management, the proposed design meets the criteria for high performance and reliability in UAV applications.

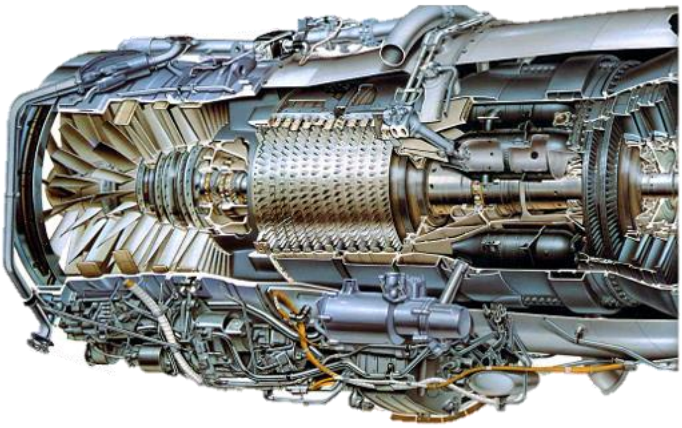
Further work to be done:

– Perform vibration and rotordynamic analysis of the compressor assembly.

-Perform rigorous CFD analysis and optimise dimensions and performance of compresser and combustion chamber
– Conduct extensive off-design performance testing and validate with experimental data.
– Optimize the fuel injection system to further improve combustion efficiency.
– Explore advanced cooling techniques (e.g., film or transpiration cooling) to enhance thermal management.
– Integrate real-time sensor feedback for adaptive control of the engine during flight.

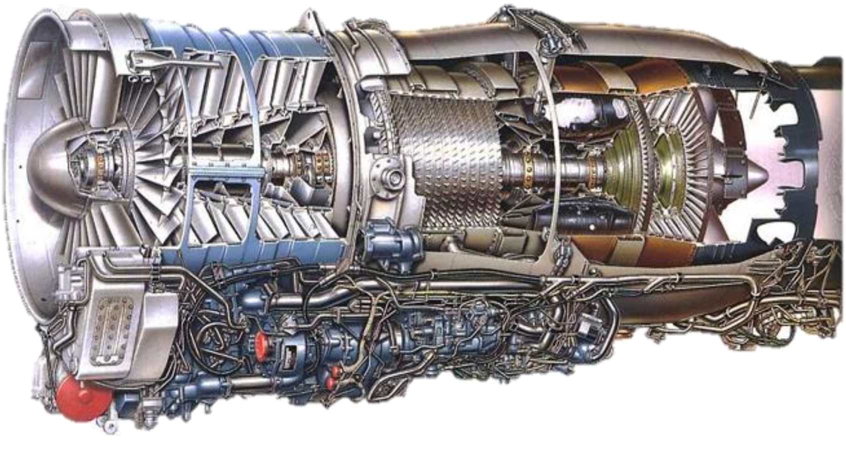
References:

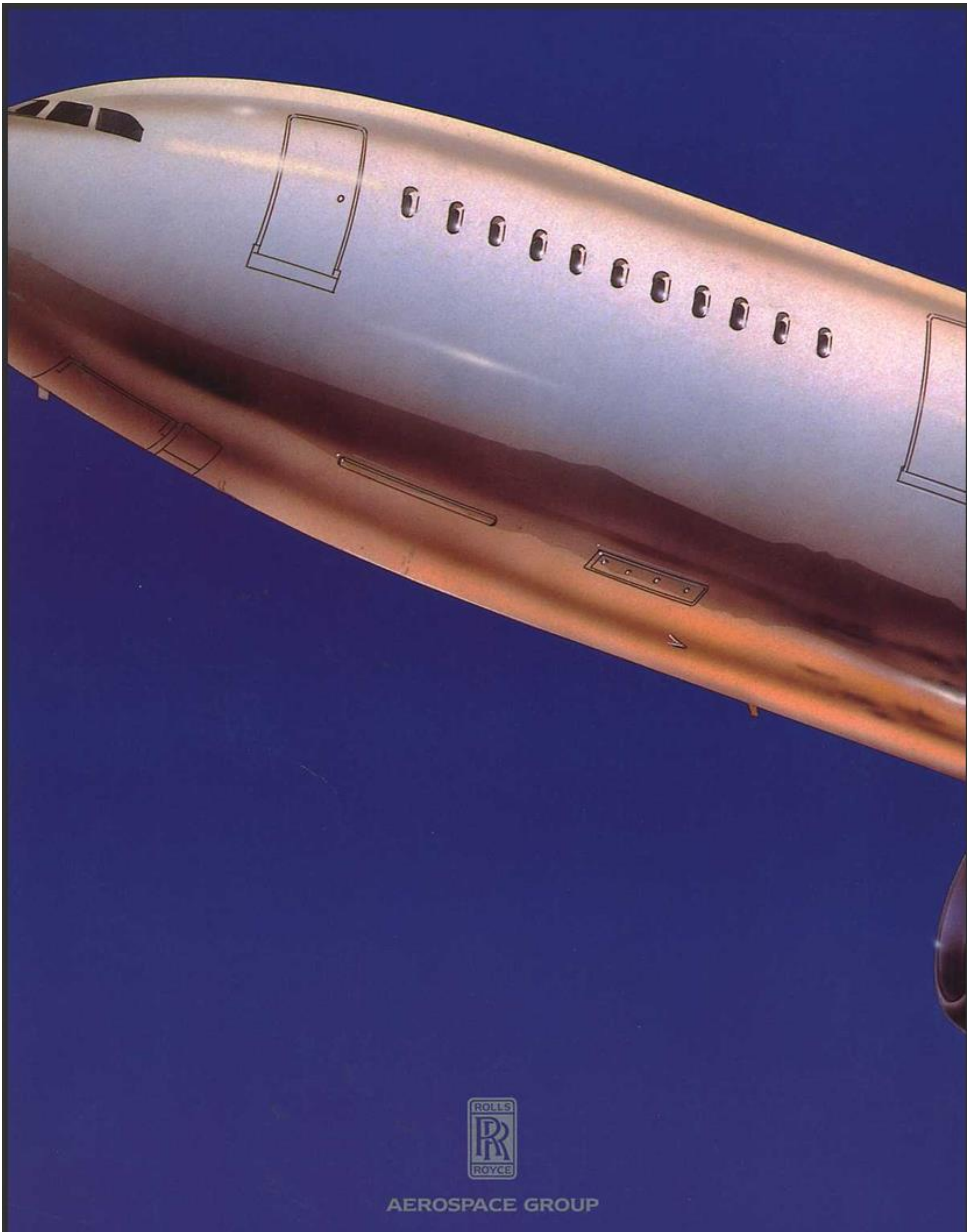
- Benini, E., & Giacometti, S. (2007). *Design, manufacturing and operation of a small turbojet engine for research purposes*. Applied Energy, 84, 1102–1116.
- Rashid S. E. Mohammed, (2019). Design and Analysis of Annular Combustion Chamber for a Micro Turbojet Engine. International Journal of Aerospace and Mechanical Engineering.
- Arthur H. Lefebvre, B. R. (1983). Gas Turbine Combustion Chamber. McGraw-Hill Book Company.
- Fahlström, S., & Pihl-Roos, R. (2016). *Design and construction of a simple turbojet engine*. Uppsala University.
- Putra, C. A. (2020). *Design and Analysis of Annular Combustion Chamber for a Micro Turbojet Engine*. Bandung Institute of Technology.
- Lefebvre, A. H. (1999). *Gas Turbine Combustion: Alternative Fuels and Emissions*.
- NASA Glenn Research Center. (n.d.). *Fundamentals of Jet Propulsion*. Retrieved from https://www.grc.nasa.gov
- Additional empirical correlations and CFD simulation data as referenced in the design documents.



Rolls-Royce RB183 Mk 555

Rolls-Royce RB168 Mk202/Mk203




ROLLS
RR
ROYCE
AEROSPACE GROUP