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

Kartikey 2022MEB1320

Lalam Gnana Deepak 2022MEB1322

YGSV Ashish 2022MEB1367

Semester II, AY 2024-25

ACKNOWLEDGEMENT

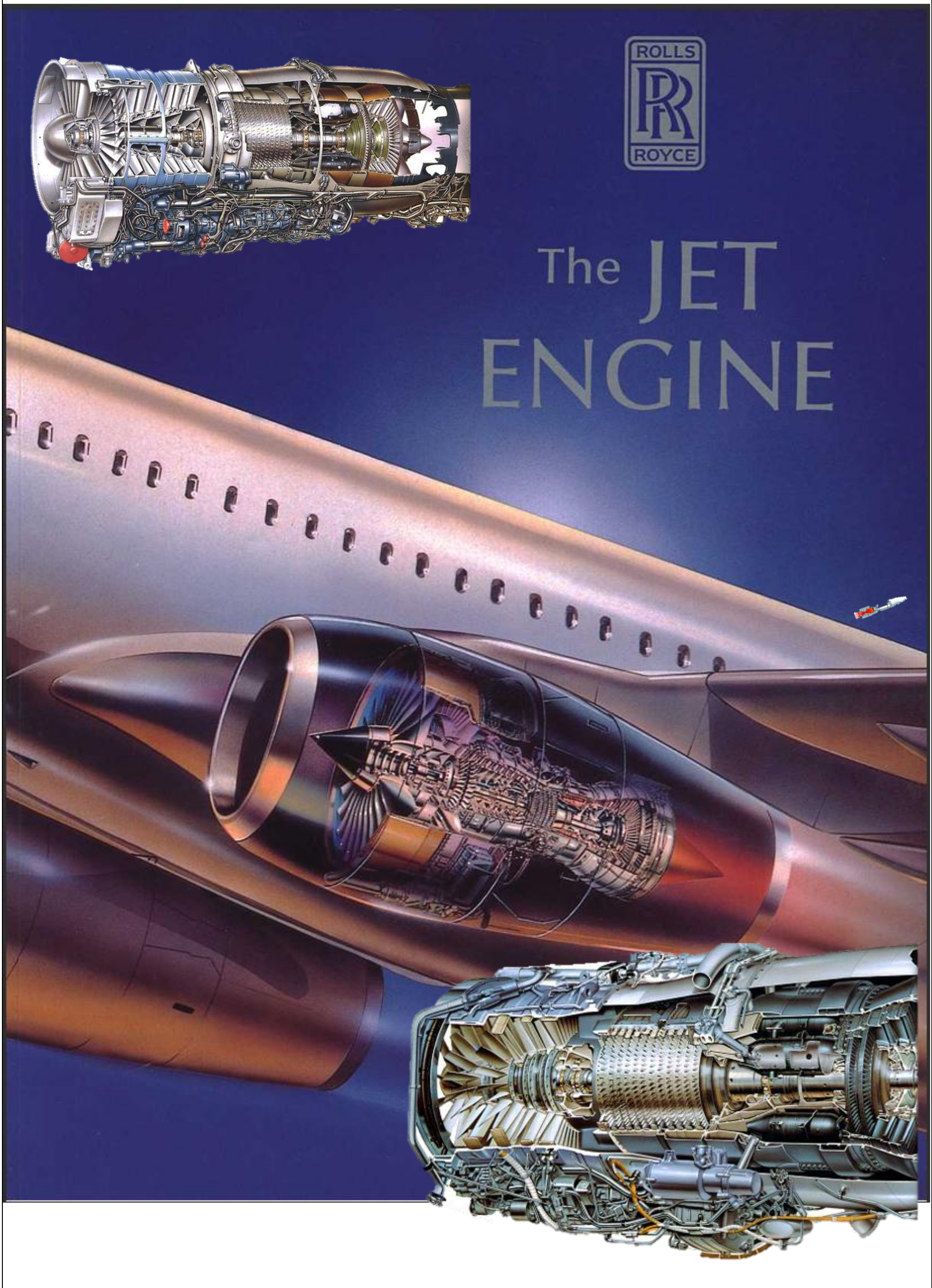
We would like to express our deepest gratitude to our esteemed guides, **Dr. Rajendra Munian** and **Dr. Sreekanth Shekar Padhee**, for their invaluable guidance, continuous support, and insightful suggestions throughout this project on **Micro Jet Engine for UAV Applications**. Their expertise and encouragement 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.

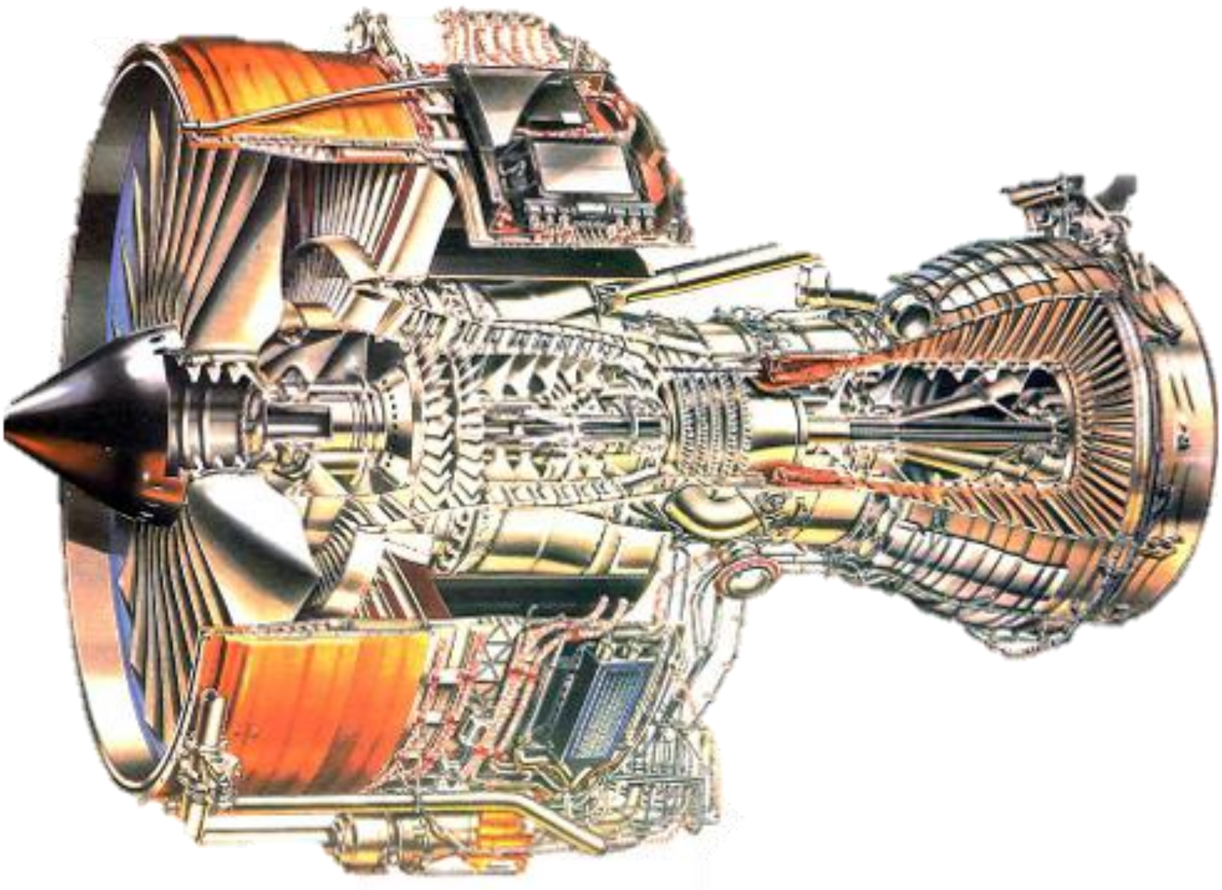
We would also like to acknowledge the various research papers, technical resources, and prior works in the field of micro jet engines, which have significantly contributed to our understanding of the subject. The knowledge gained from these references has greatly aided in developing the framework of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

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

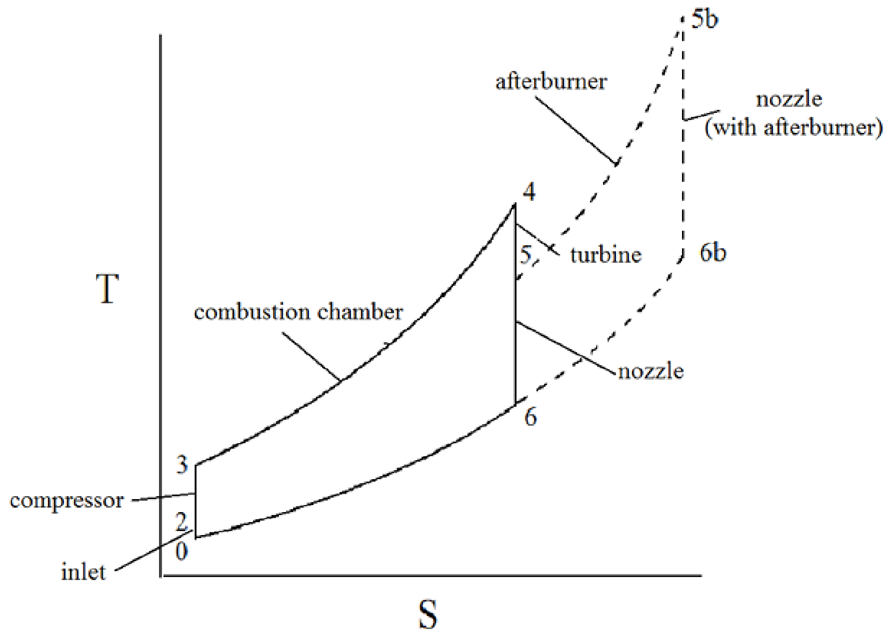


Fig. TS dig. For baryton cycle

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

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

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 Parameters:

Ambient: $P_0 = 101.3\,\mathrm{kPa},\ T_0 = 288\,\mathrm{K}$

Compressor exit (PR ≈ 2.66):

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

Mass Flow Rate (Static, $v_0 \approx 0$):

$$\dot{m}_a = \frac{T}{v_e}, \quad \text{e.g., } v_e = 500\,\mathrm{m/s} \Rightarrow \dot{m}_a = \frac{300}{500} = 0.6\,\mathrm{kg/s}$$

Compressor Inlet Area:

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

With $\rho = 1.225\,\mathrm{kg/m^3}$, $v = 50\,\mathrm{m/s}$:

$$A = \frac{0.6}{1.225 \times 50} \approx 0.0098\,\mathrm{m}^2 \Rightarrow D \approx 11.2\,\mathrm{cm}$$

• Combustion Chamber Temperature Control

For lean combustion (fuel-to-air ratio ≈ 1:50) and kerosene with lower heating value (LHV) ≈ 43 MJ/kg, temperature rise is:

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

Given:

$$\dot{m}_f = 0.03\,\mathrm{kg/s}, \quad \dot{m}_a = 0.6\,\mathrm{kg/s}, \quad c_p = 1005\,\mathrm{J/kg\,K}$$

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

Combustion chamber exit temperature:

$$T_{cc} = 416 + 855 = 1271\,\mathrm{K}$$

• Material Properties Table

Material	Density (kg/m^3)	Yield Strength (MPa)	Applications
PLA	1240–1260	50–70	Prototyping, non-structural
ABS	1010–1030	40–50	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:

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.

1. Conservation of Mass

$$\rho_e V_e A_e = \rho_0 V_0 A_0$$

- ρ_e, ρ_0: Exit and inlet densities
- V_e, V_0: Exit and inlet velocities
- A_e, A_0: Exit and inlet cross-sectional areas

2. Ideal Gas Law

At inlet and exit:

$$P_e = \rho_e R T_e, \quad P_0 = \rho_0 R T_0$$

- P, T: Pressure and temperature
- R: Specific gas constant

3. Momentum Equation (Steady Flow)

$$P_e A_e + \rho_e V_e^2 A_e = P_0 A_0 + \rho_0 V_0^2 A_0$$

Accounts for pressure forces and momentum flux.

4. Energy Equation (with Heat Addition)

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

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

Simplified Case: Constant Pressure Combustion ($P_0 = P_c$)

a) Velocity:

$$V_0 = V_c \frac{A_c}{A_0} \frac{T_0}{T_c}$$

b) Density:

$$\rho_0 = \frac{P_c}{R T_0}$$

c) Energy (rearranged):

Solve for T_0 from:

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

7. Summary

Parameter	Equation
Exit Temperature	$T_o = \theta T_e$
Exit Velocity	$V_o = V_e \theta$
Exit Density	$\rho_o = \rho_e \theta$
Temperature Ratio	$\theta = \frac{-C_p T_e + \sqrt{(C_p T_e)^2 + 2V_e^2\left(C_p T_e + \frac{V_e^2}{2} + q\right)}}{V_e^2}$

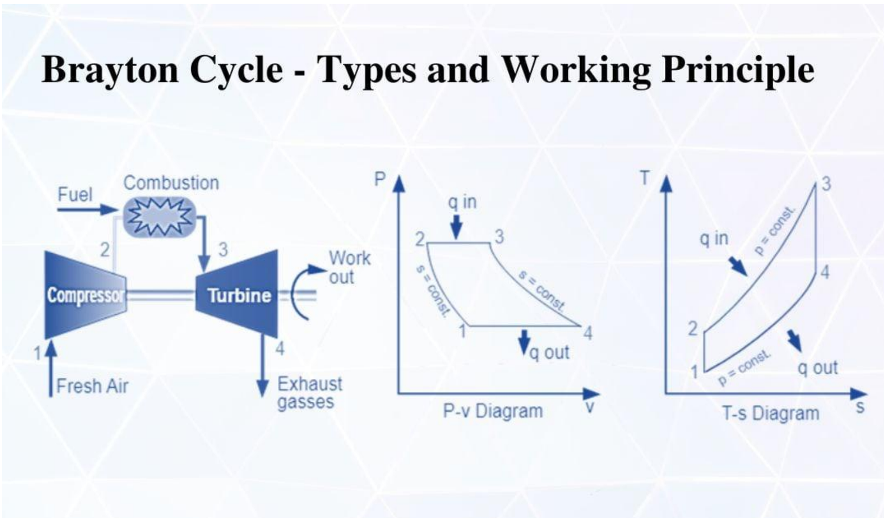


Fig3.Types of baryton cycle

1. Define Design Requirements

Objective: Design a combustion chamber liner for a turbojet engine producing 200N thrust at 0.55 kg/s air mass flow.

Key targets: Air mass distribution (Primary 20%, Secondary 30%, Dilution 50%), and allowable pressure drop (4%–10%).

2. Initial Parameters and Thermodynamic Calculations

Use compressor and turbine output conditions (temperatures, pressures, air mass flow) as inputs.

Calculate reference values like:

Reference area

Airspeed

Mach number

Dynamic pressure

3. Analytical and Empirical Design Calculations

Use Lefebvre's method to determine:

Liner dimensions (length, diameters)

Zone-wise liner lengths (Primary, Secondary, Dilution)

Cross-sectional areas

Hole size and number for each zone

Airflow distribution into zones

Pressure loss due to liner geometry

4. Liner Geometry Definition

Translate analytical results into a CAD design defining:

Liner body

Placement and size of holes in each zone

Flame tube dimensions and airflow pathways

5. Numerical Simulation (Cold Flow CFD Analysis)

Simulate non-combustive flow through the combustion chamber to validate:

Air mass flow distribution across zones

Pressure loss and recovery

Recirculation zones for fuel mixing (important for injector placement)

CFD setup includes:

k-ε turbulence model (extended wall) for accurate boundary layer modelling

Hexahedral mesh (~10 million cells)

Mesh independence study to ensure accuracy

6. Analytical vs CFD Validation

Compare results of analytical calculations with CFD simulation:

Primary zone

Secondary zone

Dilution zone

mass flow differences

Pressure drop also compared with analytical

And CFD analysis

Validate and refine geometry if discrepancies exceed tolerances.

7. Injector Placement Decision Based on CFD

Use recirculation zone data from CFD to identify optimal fuel injector positions.

Consider:

Location in primary zone

Flow velocity and direction

Adequate mixing potential for stable combustion

8. Final Design Consolidation

After validating all parameters and ensuring desired performance through cold-flow simulation:

Freeze the design for physical prototyping or further hot-flow simulation.

Future step could include combustion modeling or experimental validation.

Summary Flowchart:

1. Set requirements & thermodynamics
2. Analytical design using Lefebvre method
3. Define geometry
4. Cold flow CFD simulation
5. Compare & validate vs analytical
6. Decide injector placement using recirculation data
7. Finalize design 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.

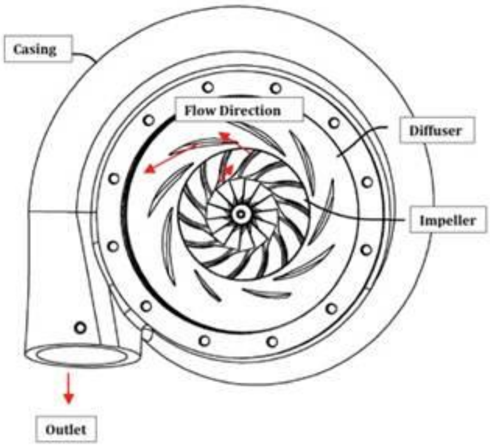


Fig4. Air flow of centrifugal compressor

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:

Fig5. Impeller Blades design

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.

Fig6. CAD model of blade design

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:

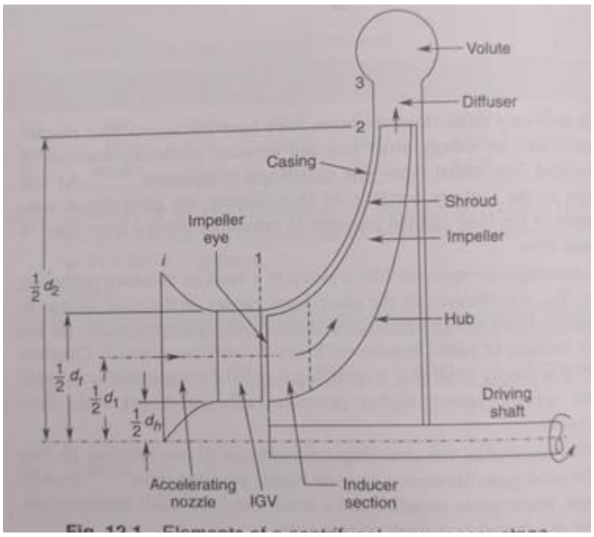


Fig7. Elements of a centrifugal compressor stage

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

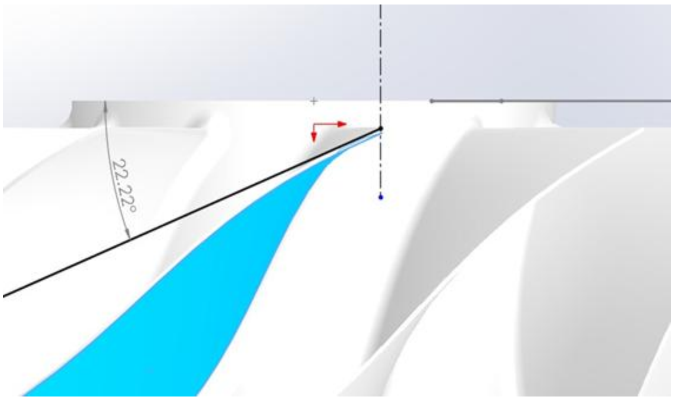
n_2 = Number of Flow separators = 11

$Alpha_1$ = 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:

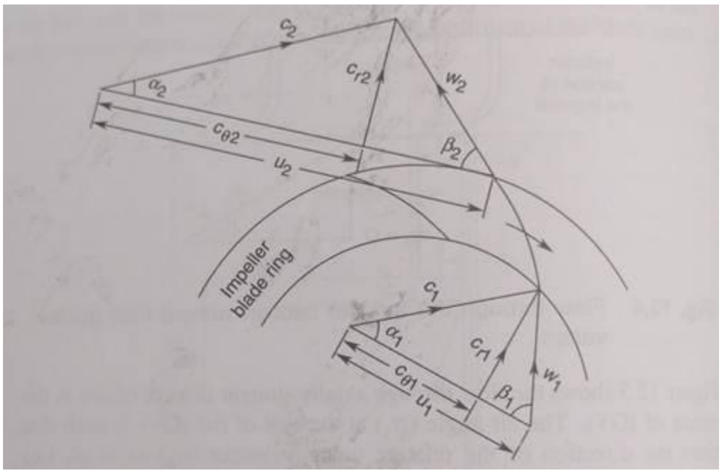


Fig8. Velocity triangles for impeller blades

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:

$u_1 = \frac{2\pi N r_1}{60}$ → blade tipvelocity at inlet

$u_2 = \frac{2\pi N r_2}{60}$ → blade tip velocity at exit

$w_1 = \frac{C_{r1}}{\sin\beta_1}$ → relative velocity of air at inlet

$w_2 = \frac{C_{r2}}{\sin\beta_2}$ → relative velocity of air at exit

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

$C_{r2} = \frac{\dot{m}}{\rho_{out}(2\pi r_2 b_2)}$ → 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, $\sigma = 1 - \frac{(0.63)(\pi)}{n}$

σ → Stannitz

n → 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, $\underbrace{C_{\theta 2} = \sigma U_2}$ → 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).

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

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

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

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

from Continuity, $\dot{m} = \rho A_2 C_{r2}$ → 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:-

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

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

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 $= D, P_{1t}, RT_{1t}$

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

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

Physical significance of π_1: $N.D \propto U_1$, $\sqrt{RT_{1t}} \propto a$

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

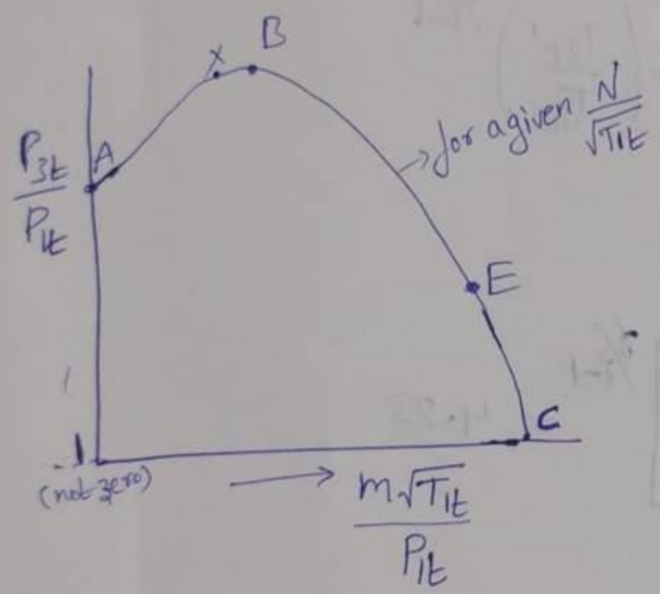
Physical significance of π_2 :-

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

$$\frac{P_{1t}}{\rho} \sim RT_{1t} \propto \frac{AV_f}{D^2\sqrt{RT_{1t}}} \propto \frac{V_f}{a} = M_f$$ (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 $\cancel{X}$ 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 $\frac{N}{\sqrt{T_{1t}}}$

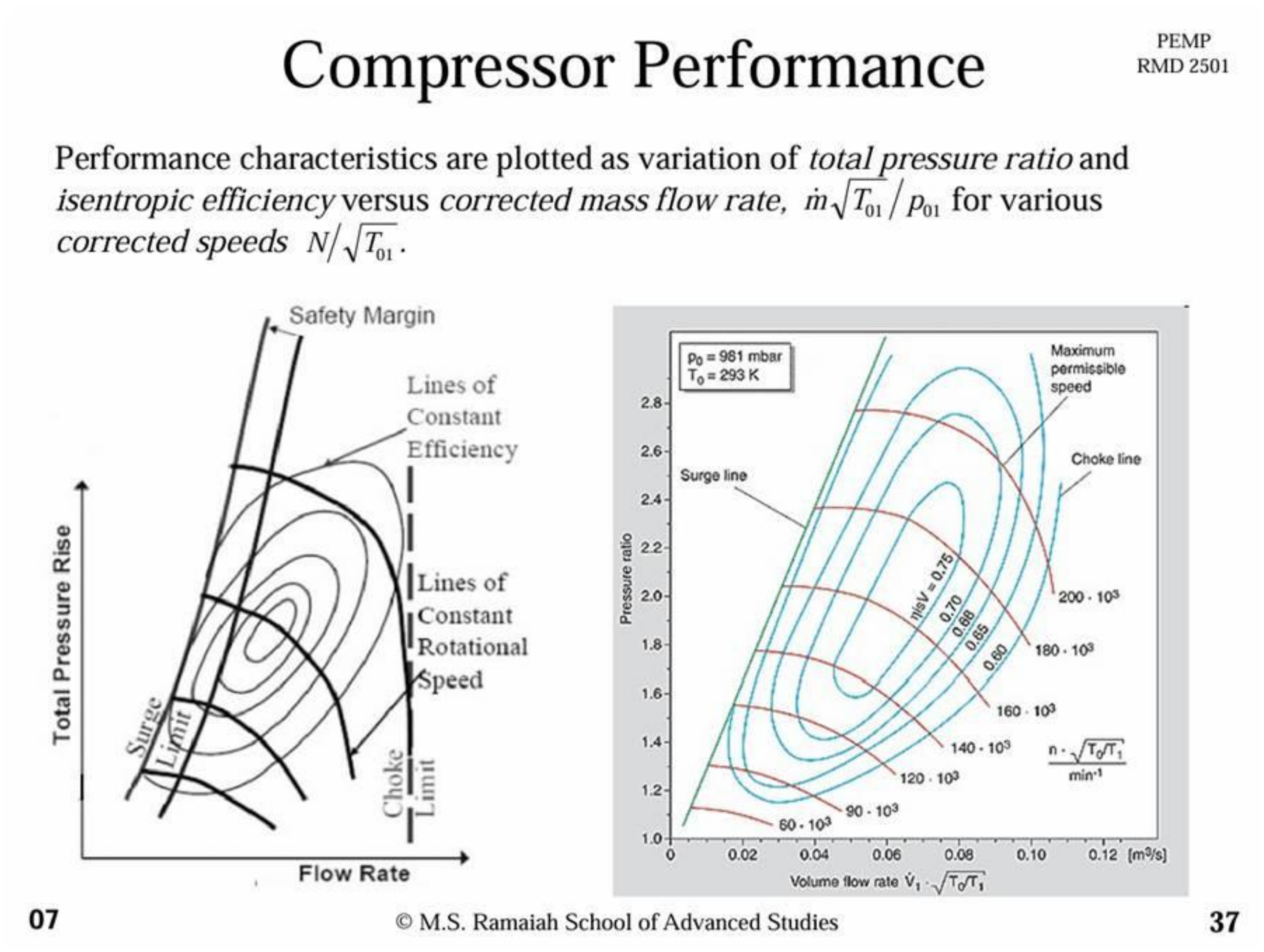


Fig9. Compressor performance graph

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.

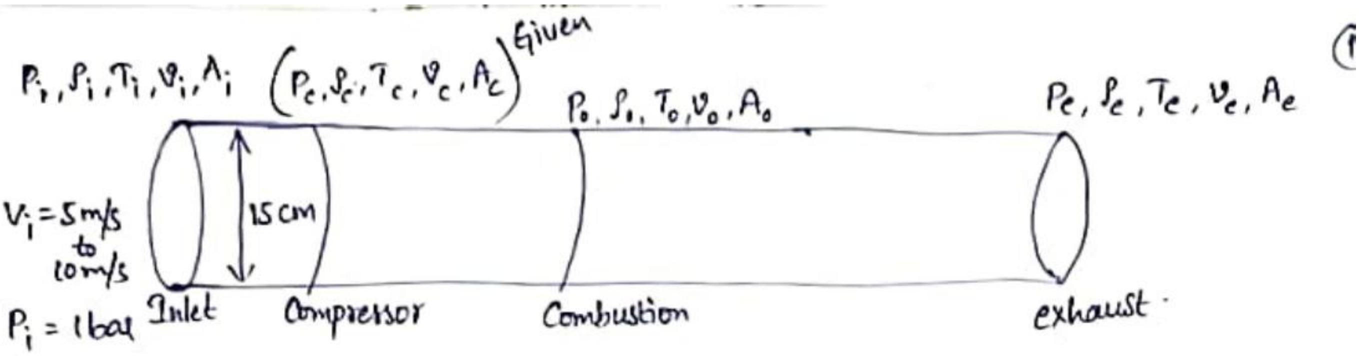
Design of the Diffuser: The diffuser decelerates the air by decreasing its kinetic energy and converting it into static pressure.

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)



$V_i = 5 m/s$ to $10 m/s$

$P_i = 1 bar$

$T = 300 K$ (or) $27°C$

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

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

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

$$\rho_c V_c = \rho_0 V_0$$

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

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

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

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

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

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

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

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.

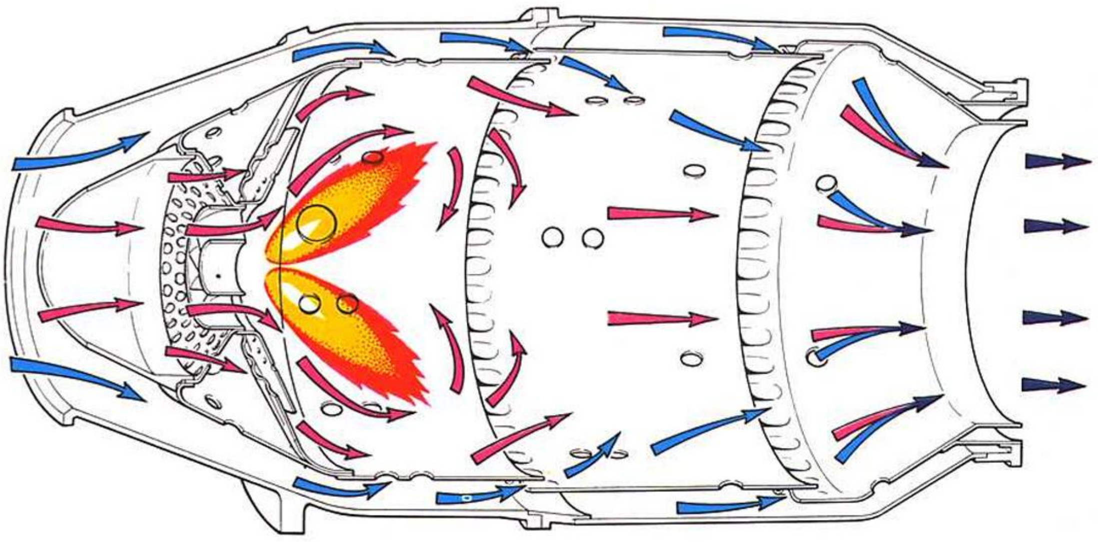


Fig10. Airflow pattern in combustion chamber

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.

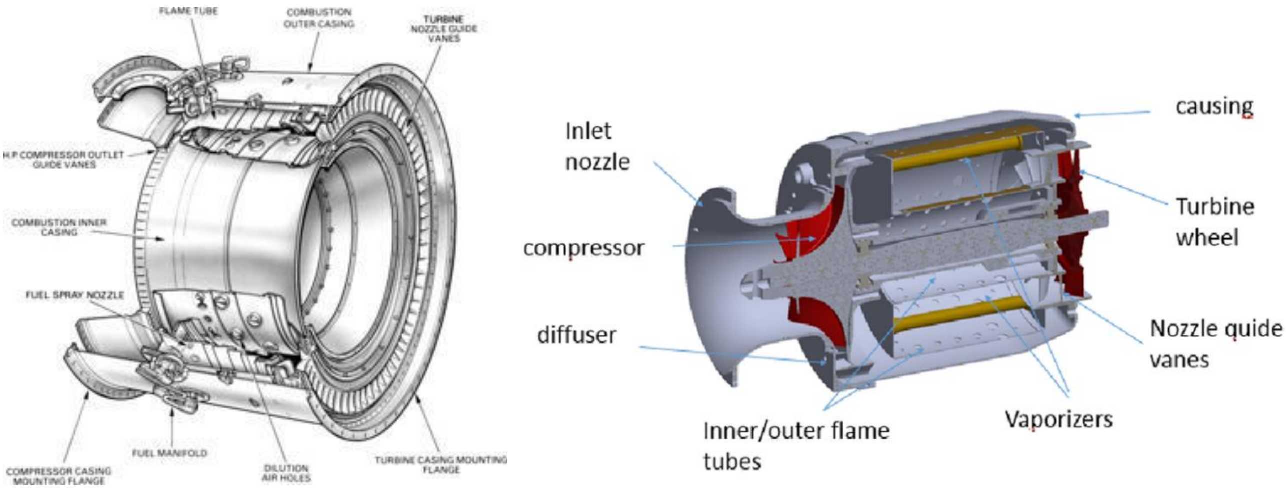


Fig11. Combustion chamber types requirements

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.

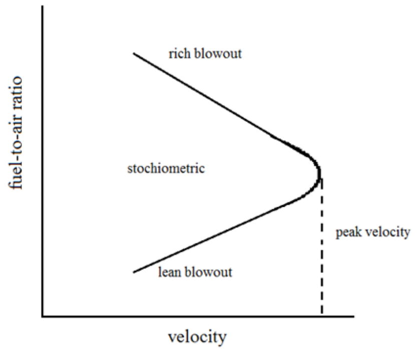


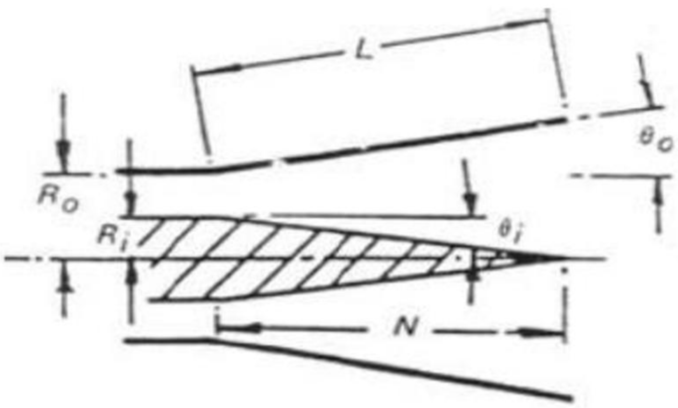
Figure 8: Fuel-air ratio vs velocity.

-

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 7^0 and 12^0. 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.

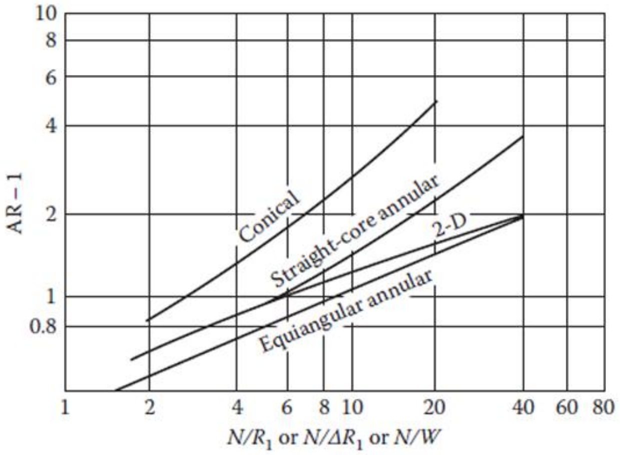


Figure 2 Lines of First Stall

Input

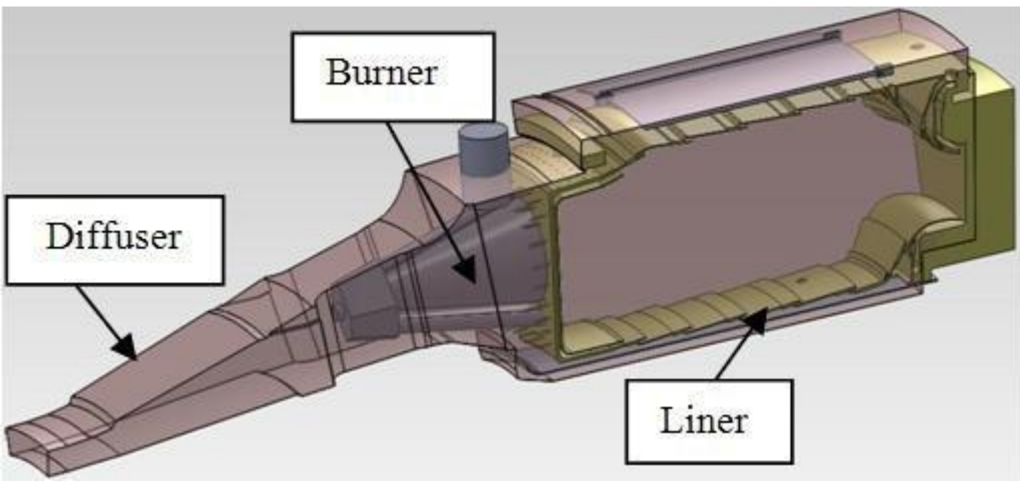
- Dout = Outer exit comp. dia.
- Din = Inner exit comp. dia.
- θo = Outer divergence angle
- θi = Inner divergence angle
- L = Wall length

Output

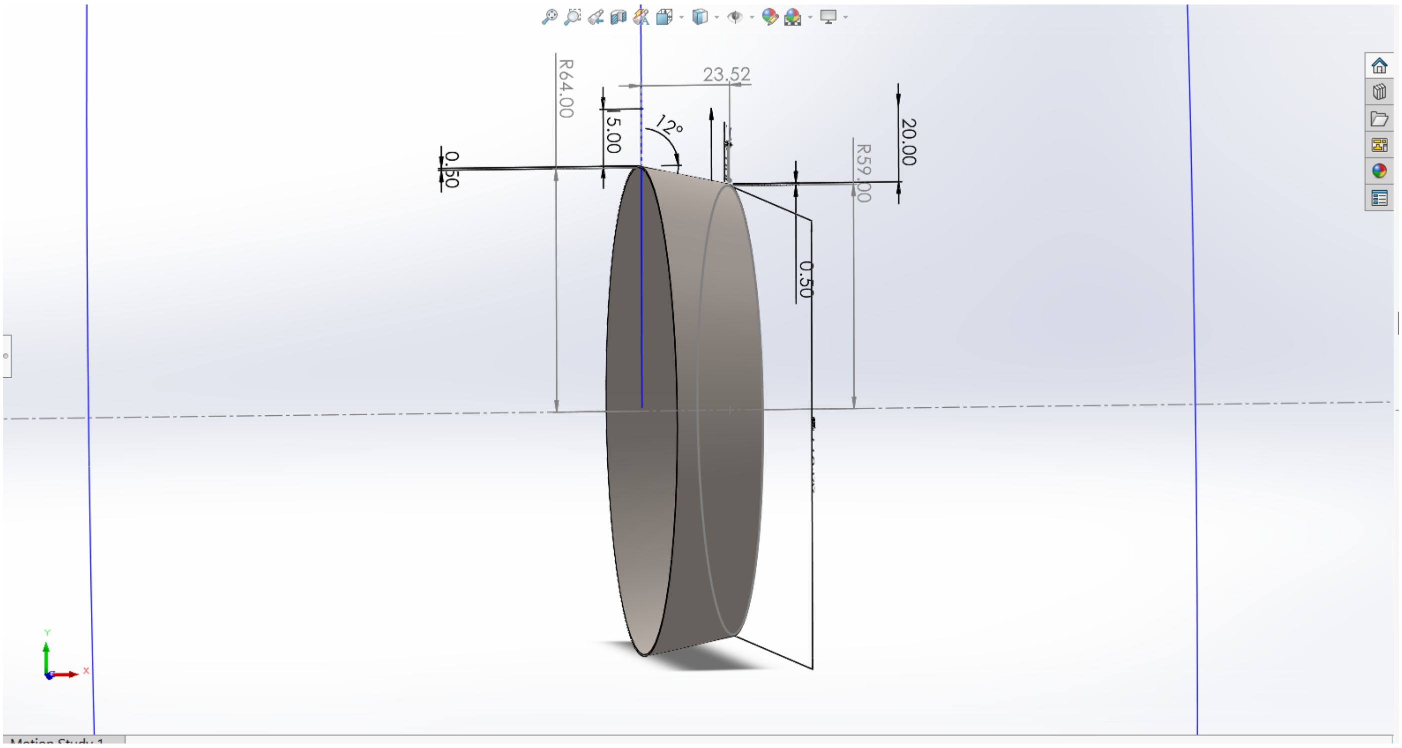
- AR = Aspect Ratio
- N = Axial length

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



FINAL PROTOTYPE DESIGN:



3. Governing Formulae

3.1 Non-dimensional length

$$N = \frac{L}{\Delta R}$$

3.2 Equiangular-Diffuser Aspect Ratio

From Desaindan & Analisis (Your Reference PDF):

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

Parameter	Symbol	Value	Notes
Axial length	L	23.52 mm	also N
Inner-wall divergence angle	θ_i	12°	
Outer-wall divergence angle	θ_o	0°	
Inner diffuser radius	R_i	59 mm	
Outer diffuser radius	R_o	64 mm	
Radial gap	$\Delta R = R_o - R_i$	5 mm	
Radius ratio	$\frac{R_i}{R_o}$	0.9219	
$\sin\theta_i$	—	0.2079	$\sin 12^\circ$
$\sin\theta_o$	—	0	
Term 2	$2\left(L\sin\theta_i + (R_i/R_o)\sin\theta_o\right)/\left(\Delta R(1 + R_i/R_o)\right)$	1.0175	middle term of AR formula
Term 3	$\frac{(L^2/\Delta R^2)(1-R_i/R_o)(\sin^2\theta_i - \sin^2\theta_o)}{1+R_i/R_o}$	0.0389	last term
Aspect ratio	AR	**2.06**	$1+$ Term 2 $+$ Term 3

AR = 2.06 and N/R = 4.7 [N=23.52 & R=5]

From Lines of First Stall

1.75

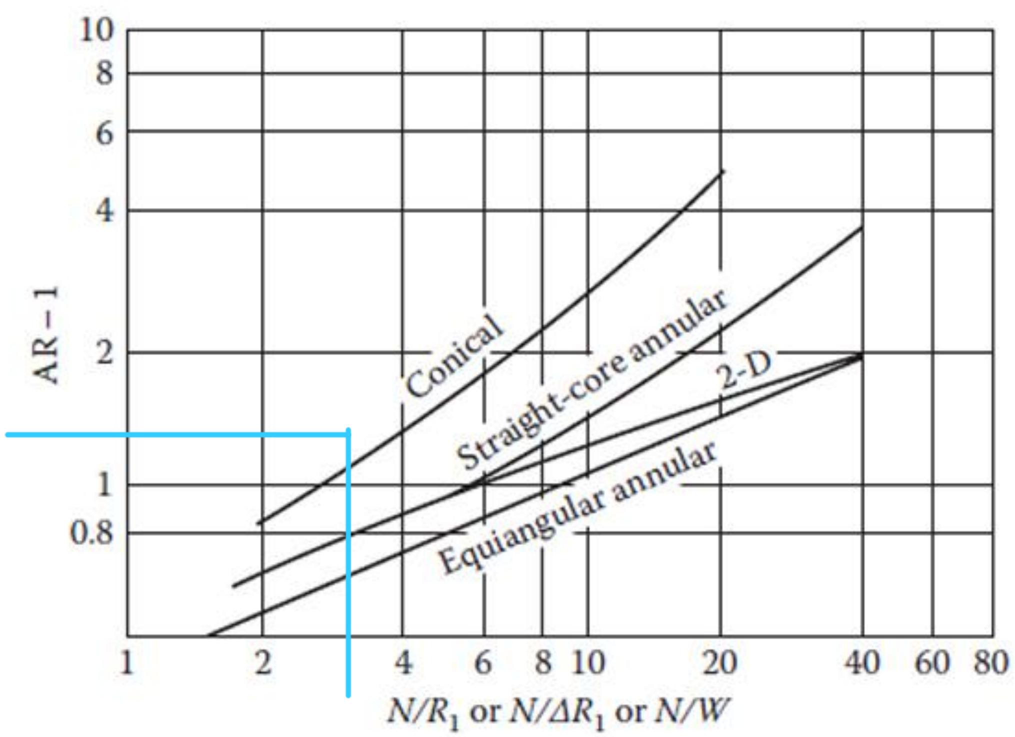


Figure 2 Lines of First Stall

AR= 2.75

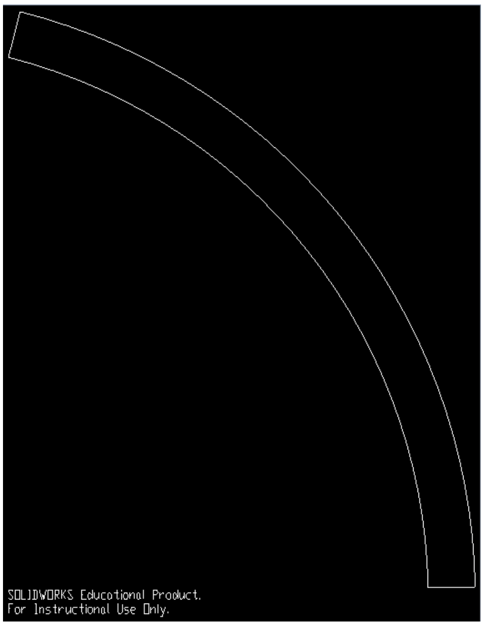
Dxf file path which is cut with laser and folded to give diverging duct

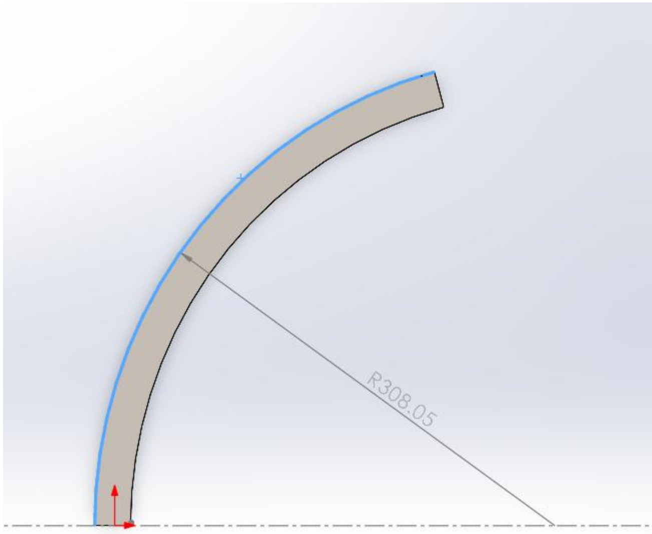
Our AR= 2.06 is < 2.75 so no stall will occur, and we minimized pressure losses with this dimension diverging duct.

METHOD OF FABRICATION:

We plan to make this conical shape of diverging by cutting it from a base of cone in solid works, converting it to sheet metal model, inserting bends and sheet metal parameters (K-factors, etc) and unfolding this

conical base will give shape of arc, which can be converted to DXF file and can be cut precisely using metal laser cutter on sheet metal. DXF FILE





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.

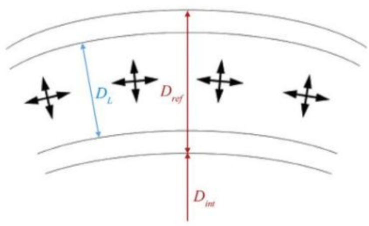


Figure 3 Geometry of Reference Area

Equation 2

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

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
-

CALCULATIONS :

Parameter	Symbol	Value	Units
Outer diameter	D_{out}	79.50	mm
Inner diameter	D_{in}	58.66	mm
Reference area	A_{ref}	2.26×10^{-3}	m²
Mass-flow rate	$\dot{m}_a$	0.065	kg/s
Reference Mach number	M_{ref}	0.03	–
Compressor-exit total density	ρ_{t3}	2.81	kg/m³
Reference velocity	U_{ref}	10.2	m/s
Reference dynamic pressure	q_{ref}	146.2	Pa

1. **Reference area**

$$A_{\text{ref}} = \frac{\pi}{4}\left((0.0795)^2 - (0.05866)^2\right) \approx 2.26 \times 10^{-3}\,\text{m}^2$$

2. **Reference velocity**

$$U_{\text{ref}} = \frac{\dot{m}_a}{\rho_{t3}\,A_{\text{ref}}} = \frac{0.065}{2.81 \times 2.26 \times 10^{-3}} \approx 10.2\,\text{m/s}$$

3. **Reference dynamic pressure**

$$q_{\text{ref}} = \tfrac{1}{2}\,\rho_{t3}\,U_{\text{ref}}^2 = 0.5 \times 2.81 \times (10.2)^2 \approx 146.2\,\text{Pa}$$

For mass flow rate of 0.065 kg/s and for reference velocity to be 0.03 mach, the reference velocity, reference dynamic pressure, and area are 10.2m/s, 146.2Pa and 2.26x10^-3 m^2.

The calculated value of D is not assumed randomly, it was the final D after several trails and iterations with existing dimensions of Micro jet engines, and we finally settled up with these dimensions as we are constrained by unavailability of desired sizes of pipes.

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).

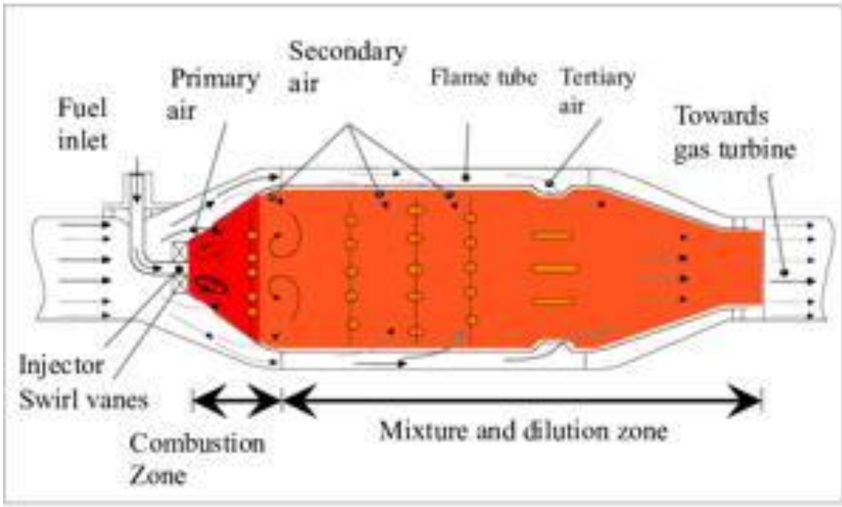
$$\pi_B = \frac{p_{t4}}{p_{t3}} = 1 - \frac{\Delta p_{t,3-4}}{p_{t3}} \tag{5}$$

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

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

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

- $\Delta p_{t,3-4}/p_{t3}$ = 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



CALCULATIONS :

Symbol	Quantity	Value
πB	Burner pressure ratio	0.90
Aref	Reference area	2,261.36 mm²
AR	Diffuser aspect ratio	2.04
Pt3	Compressor-exit total pressure	151,987.5 Pa
Δpt,3-4/pt3	Combustor pressure-loss ratio	0.10
Δpt,3-4	Total pressure loss in combustor	15,198.7 Pa
Δpt,3-4/qref	Combustor pressure-drop factor	103.4
Uin,diff	Avg. velocity in diffuser	92.83 m/s
λ	Dump-diffuser constant	0.45
Δpt,diff	Total pressure loss in diffuser	2,670.8 Pa
Δpt,diff/qref	Diffuser pressure-drop factor	18.2
Δpt,L	Total pressure loss in liner	12,528.0 Pa
Δpt,L/qref	Liner pressure-drop factor	85.2

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.

AL=kAref (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

CALCULATIONS:

2.7.1 Liner Cross-Section (Eq. 9)

$$A_L \;=\; k\,A_{\text{ref}}$$

where

- k = liner-to-reference-area ratio (choose 0.70 for low gas-velocity),
- A_{ref} = reference area (from diffuser sizing table).

Using

$$k = 0.70, \quad A_{\text{ref}} = 2.26136 \times 10^{-3}\,\text{m}^2,$$

yields

$$A_L = 0.70 \times 2.26136 \times 10^{-3} \;=\; 1.583 \times 10^{-3}\,\text{m}^2.$$

2.7.2 Flame-Tube Height (Eq. 2 of report)

In an annular liner the flow-area is

$$A_L = \frac{\pi}{4}\left(D_{L,o}^2 - D_{L,i}^2\right)$$

but we can instead express the **liner "height"** (radial shell thickness) as

$$D_L = \frac{D_{L,o} - D_{L,i}}{2}.$$

Using your compressor-exit geometry:
$D_{L,o} = 79.50\,\text{mm}, \;\; D_{L,i} = 58.66\,\text{mm}$

$$D_L = \frac{0.07950 - 0.05866}{2} = 0.01042\ \text{m}.$$

2.7.3 Pattern Factor (Eq. 11)

$$PF = \frac{T_{t,\max} - T_{t4}}{T_{t4} - T_{t3}} \tag{11}$$

with

$T_{t,\max} = 1210\ \mathrm{K},\ T_{t4} = 1175\ \mathrm{K},\ T_{t3} = 475\ \mathrm{K}$,

$$PF = \frac{1210 - 1175}{1175 - 475} = 0.05.$$

2.7.4 Liner Length (forced to 125 mm)

The original formula (Eq. 10) relates the liner length L_L to the allowable pressure drop and pattern factor. Here, **we set:**

$$L_L = 0.125\ \mathrm{m}.$$

2.7.5 Zone Lengths (Eqs. 12–14)

Divide the total liner length L_L into the three flame-tube zones:

1. **Primary-zone length**

$$L_{PZ} = 0.75\, D_L = 0.75 \times 0.01042 = 0.00782\ \mathrm{m} \tag{12}$$

2. **Secondary-zone length**

$$L_{SZ} = 0.50\, D_L = 0.50 \times 0.01042 = 0.00521\ \mathrm{m} \tag{13}$$

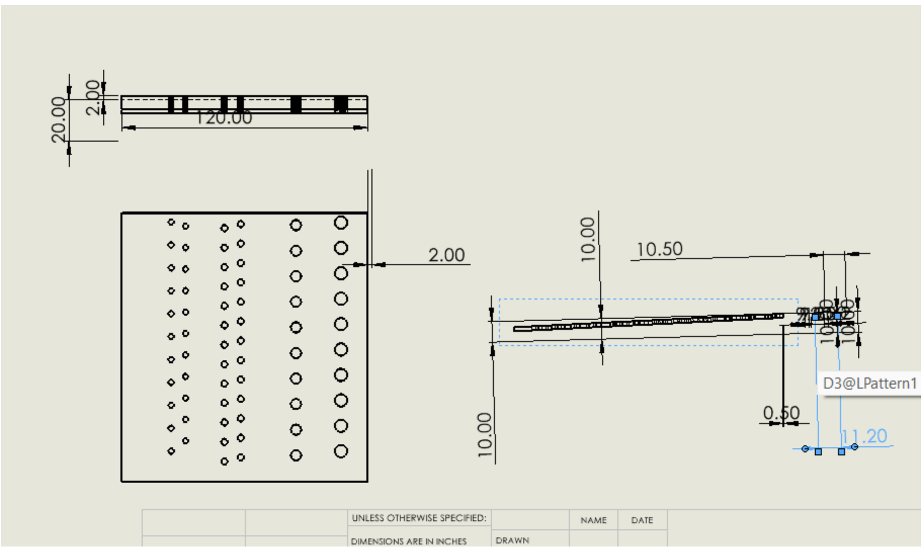
3. **Dilution-zone length**

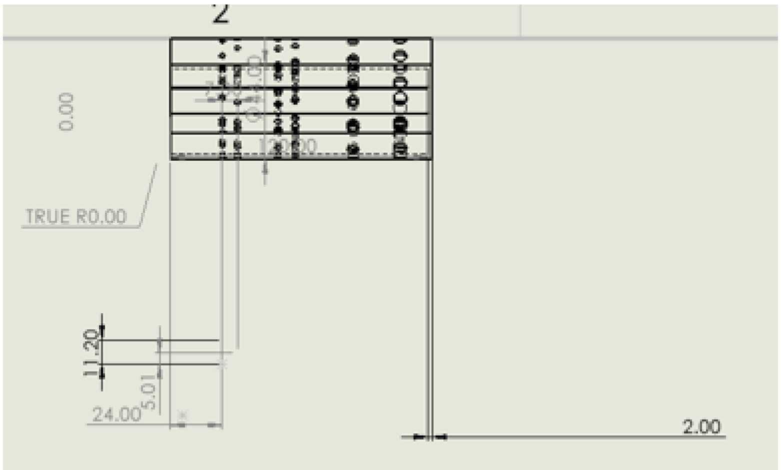
$$L_{DZ} = L_L - L_{PZ} - L_{SZ} = 0.125 - 0.00782 - 0.00521 = 0.11197\ \mathrm{m} \tag{14}$$

2.7.6 Summary Table

Symbol	Quantity	Value	Units
Inputs			
k	Liner-to-reference area ratio	0.70	–
A_{ref}	Reference area	2.26136×10^{-3}	m^2
$D_{L,i}$	Inner liner diameter	58.66×10^{-3}	m
$D_{L,o}$	Outer liner diameter	79.50×10^{-3}	m
$T_{t,\max}$	Maximum liner-exit temperature	1 210	K
Intermediate			
A_L	Liner flow-path area (Eq. 9)	1.583×10^{-3}	m^2
D_L	Flame-tube "height"	0.01042	m
PF	Pattern factor (Eq. 11)	0.05	–
Outputs			
L_L	Total liner length (forced)	0.125	m
L_{PZ}	Primary-zone length (Eq. 12)	0.00782	m
L_{SZ}	Secondary-zone length (Eq. 13)	0.00521	m
L_{DZ}	Dilution-zone length (Eq. 14)	0.11197	m

DESIGN AND FABRICATION:





Due to unavailability of desired pipe dimensions, we settled with k=0.9 instead of optimal k (0.6-0.77) , we will cut the inner liner, outer liner and outer casing MS pipes of 3mm thickness, to required liner lengths and , the front cap will be a annular disk which will be cut from 1mm SS sheet using metal laser cutter , for maximum heat endurance , as highest temperatures (<1250 K)will be generated near the point of fuel injection and recirculation zone .

Fig12. Sheet metal, mild steel and diesel and injection pipes for construction of combustion chamber

Fig13. Combustion chamber front part cut using Metal laser cutter

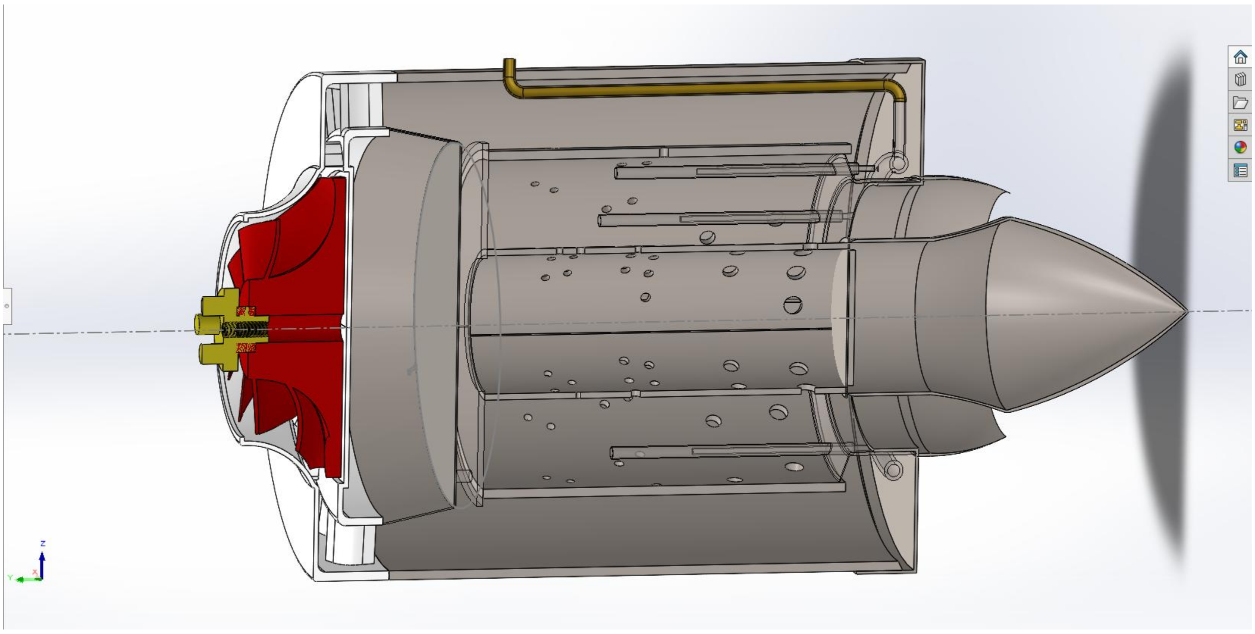


Fig14. Right plane cross sectional view

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.

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

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

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

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

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.

CALCULATIONS :

1. **Zone mass-flows**

   Compute $\dot{m}_{a,\mathrm{PZ}}$, $\dot{m}_{a,\mathrm{SZ}}$, $\dot{m}_{a,\mathrm{DZ}}$ from Eqs. (15)–(17).

2. **Choose hole-count n_j**

   Decide on a symmetric arrangement: each half-annulus has n_j holes, so total holes = $2\,n_j$.

3. **Mass-flow per hole**

$$\dot{m}_j = \frac{\dot{m}_{\mathrm{zone}}}{2\,n_j}$$

4. **Effective diameter** (Eq. 18)

$$\dot{m}_j = \rho_t\,U_j\,\frac{\pi\,d_{\mathrm{eff,j}}^2}{4} \quad \Longrightarrow \quad d_{\mathrm{eff,j}} = \sqrt{\frac{4\,\dot{m}_j}{\pi\,\rho_t\,U_j}} \tag{18}$$

5. **Pressure-drop / penetration** (Eqs. 19–20)

   (Used if accounting for local $\Delta p_{t,j}$ and entrainment. Here we assume the same Δp and neglect secondary-flow correction factors.)

6. **Real hole diameter** (Eq. 21)

   Apply a discharge-coefficient correction

$$d_j = \frac{d_{\mathrm{eff,j}}}{C_D} \quad \text{with } C_D = 0.6 \tag{21}$$

2.9.2 Estimated Hole Sizes and Counts

Zone	Symbol	Quantity	Value	Units
Primary	$d_{j,ax}$	Axial–row hole diameter	2.61	mm
	n_{ax}	Holes per half-annulus	12	–
	$d_{j,ann}$	Annular–row hole diameter	2.61	mm
	n_{ann}	Holes per half-annulus (row)	12	–
Secondary	$d_{j,SZ}$	Annular hole diameter	3.07	mm
	n_{SZ}	Holes per half-annulus (each row)	10	–
Dilution	$d_{j,DZ}$	Annular hole diameter	5.50	mm
	n_{DZ}	Holes per half-annulus	8	–

> **Notes:**
>
> - "ρ_t, U_j for each zone come from your diffuser/liner flow calculations."
> - "$\dot{m}_{\text{zone}}$ from your airflow-distribution (Table 7)."
> - "We targeted hole diameters around 2.6 mm, 3.1 mm, and 5.5 mm by adjusting n_j in the above procedure."

To derive the theory, formulas, and calculations behind the fuel injector placement in the primary zone (recirculation zone) as stated in your quote — involving horizontal (H) and vertical (V) distances, angular position (θ), and tangential velocities — the design follows a standard methodology based on Lefebvre's analytical rules for gas turbine combustion chambers. Here's a structured breakdown:

1. Theoretical Background (Source: Lefebvre & Ballal, 2010)

In Lefebvre's "Gas Turbine Combustion", critical factors for fuel injector placement include:

Stable flame anchoring via a recirculation zone formed using swirl or reverse flow in the primary zone.

Proper atomization and mixing of air-fuel mixture for ignition and sustained combustion.

Recirculation helps in reintroducing hot combustion products to incoming fuel-air mixture to sustain ignition.

Reference chapters:

Section 1.7 – Primary Zone (Pg. 15)

Section 4.14.2 – Size of Recirculation Zone (Pg. 144)

Section 5.9 – Mechanisms of Flame Stabilization (Pg. 179)

2. Injector Placement Criteria

From the journal you referenced (Santoso et al., 2023):

Angular Spread (θ): 32.72° to 360° with ~32.73° increments ensures symmetrical fuel injection in the annular chamber.

H–V Distances from Liner Wall

Horizontal: 34–40 mm

Vertical: 35–43 mm

Tangential Velocity Range: 43–60 m/s in outer recirculation zone

This geometry supports formation of a toroidal vortex which traps and recirculates hot gases for improved mixing and flame holding.

3. Key Design Equations and Calculations

As per Lefebvre and supported in the paper:

a. Airflow Split

Typical combustion chamber air distribution:

Primary zone: ~20% (for fuel mixing)

Secondary: ~30% (complete combustion)

Dilution: ~50% (temperature control)

b. Recirculation Diameter (d):

Used to define the vortex size and injector offset

$$d = \sqrt{4\mathrm{A}_{\mathrm{rec}}/\pi}$$

c. Jet Penetration (Z) and Lateral Coverage (H)

Used to ensure jets enter and stay within recirculation:

$$Z = K \cdot \left(\frac{\rho_j}{\rho_m} \cdot \frac{d_j}{D} \cdot \frac{V_j}{V_m} \right)$$

d. Tangential Velocity in Swirl

From swirler theory, tangential velocity at radius:

$$V_t = S . V_a$$

Designers set swirl such that recirculation strength gives ~50% reversal velocity, allowing injector placement at optimal zones to maximize mixing and ignition stability.

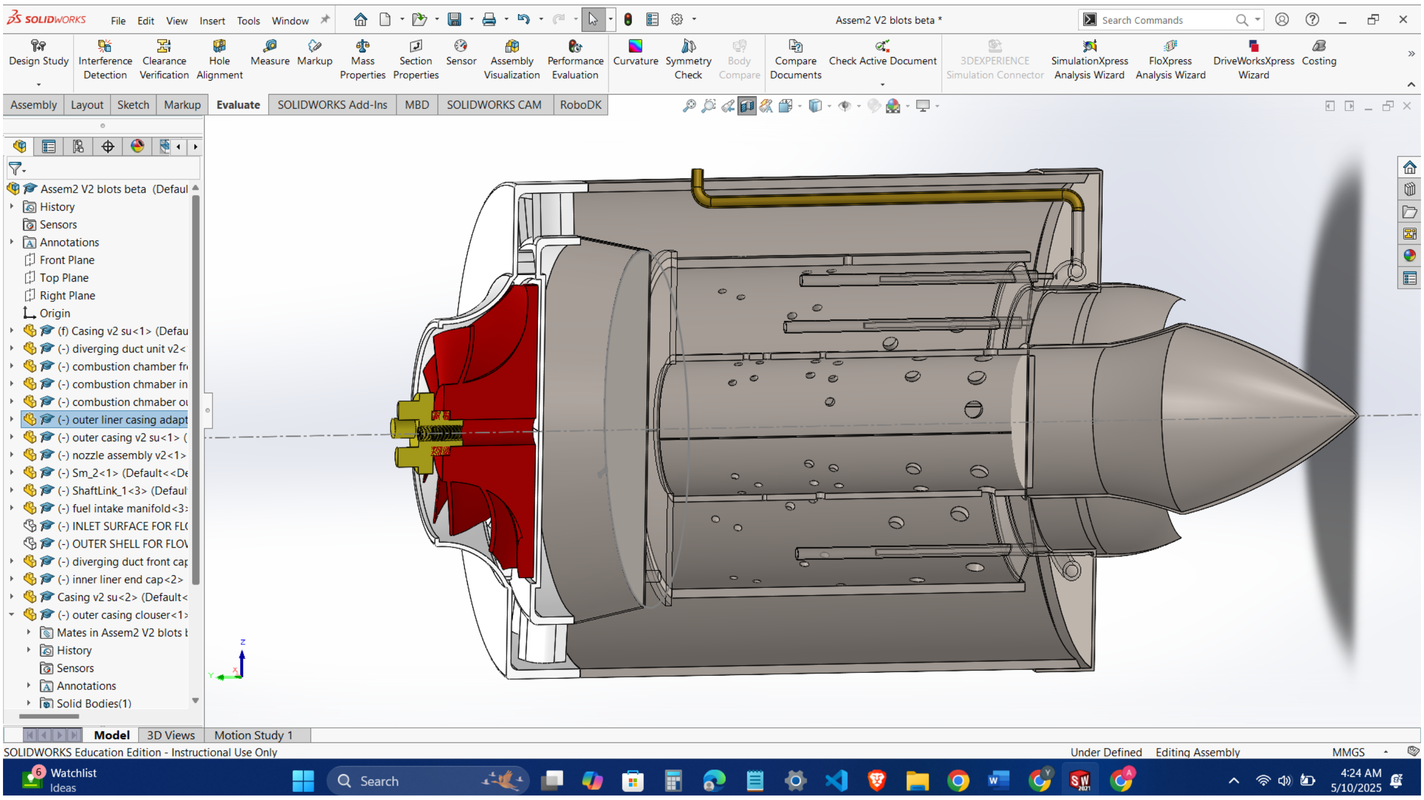
4. Empirical Design Rule (From Santoso et al.)

The empirical region (θ = 32.72° to 360° with increments of 32.73°) likely emerges from:

Liner hole arrangement strategy for uniform flame anchoring.

CFD validation of velocity streamlines showing recirculation strength optimal in these angular and radial bands.

CAD MODEL



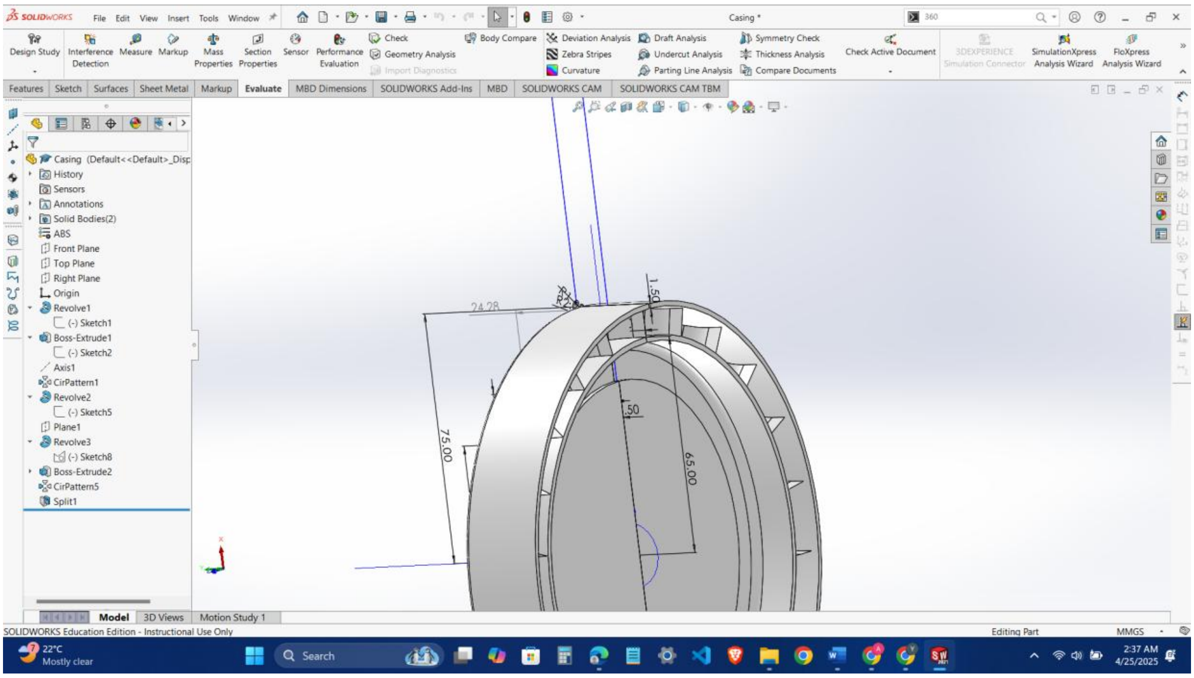


Fig15. Compressor Diffuser Dimensions

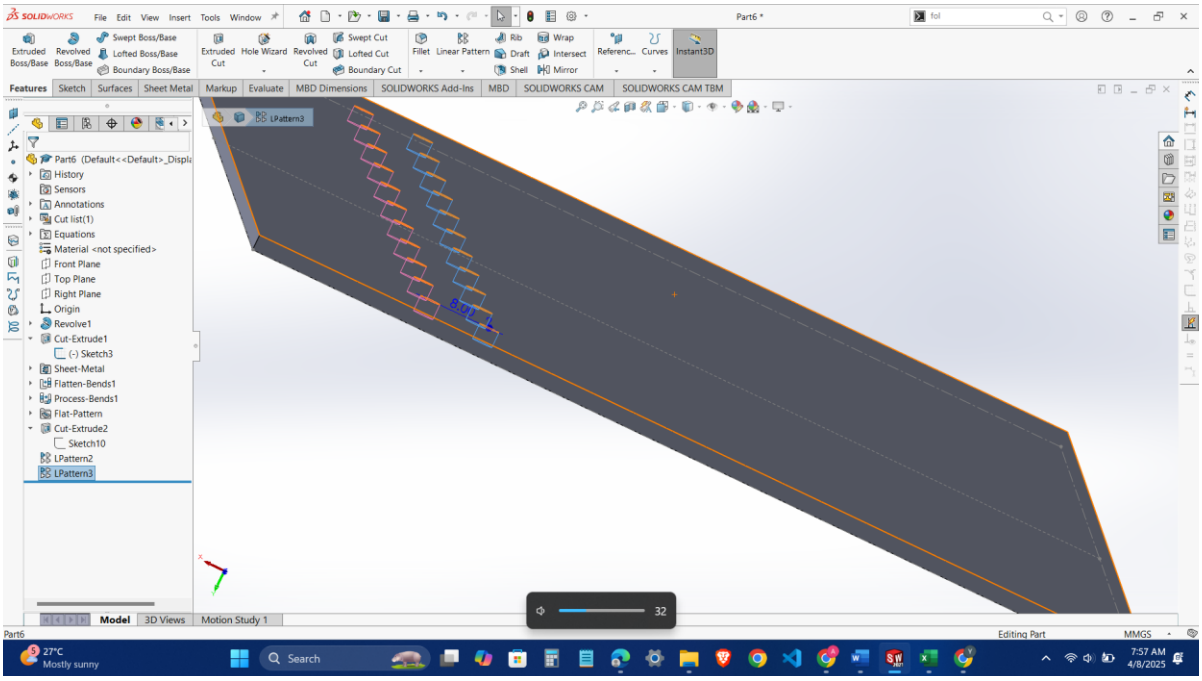
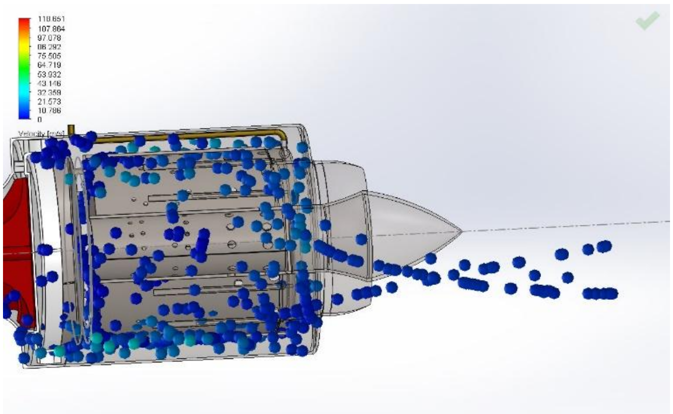


Fig16. Primary zone hole positions in unfolded outer liner



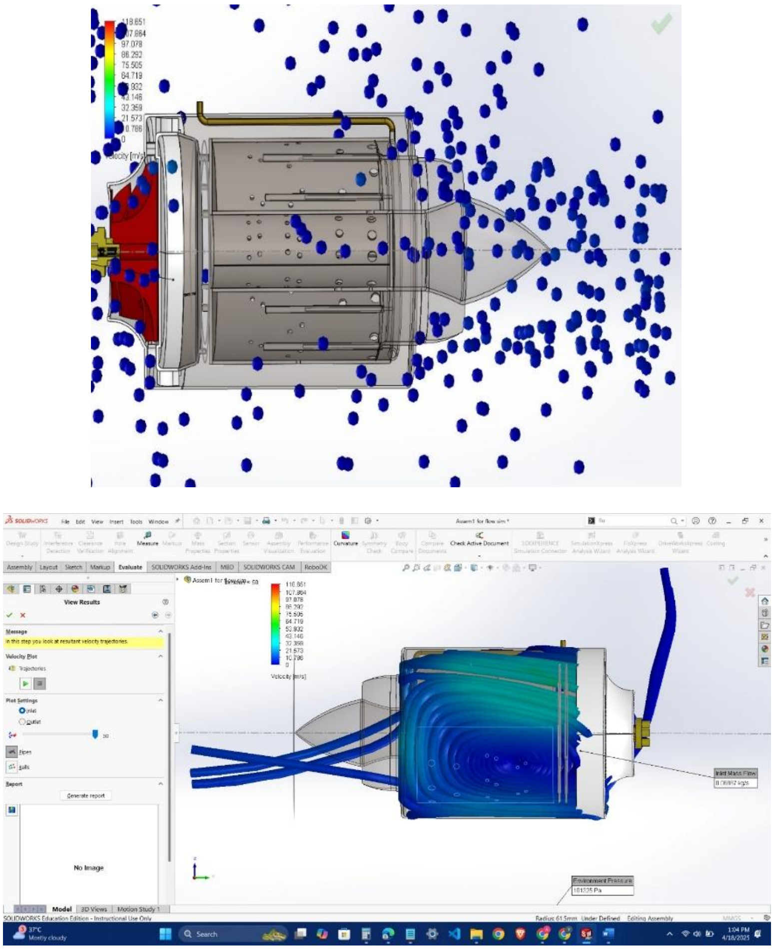


Fig17. Basic primitive flow express analysis in solid works

Only for Representation of flow ,no useful results obtained due course mesh and analysis, Primary zone hole positions in unfolded outer liner.

DESIGNED PROTOTYPE

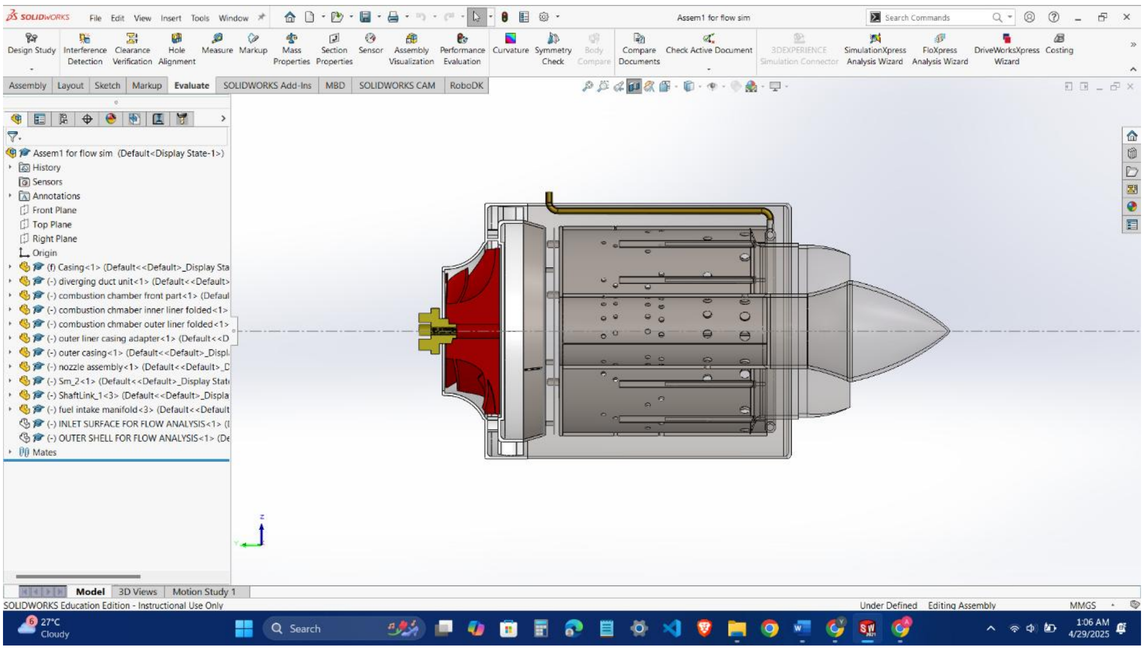


Fig18. Design prototype

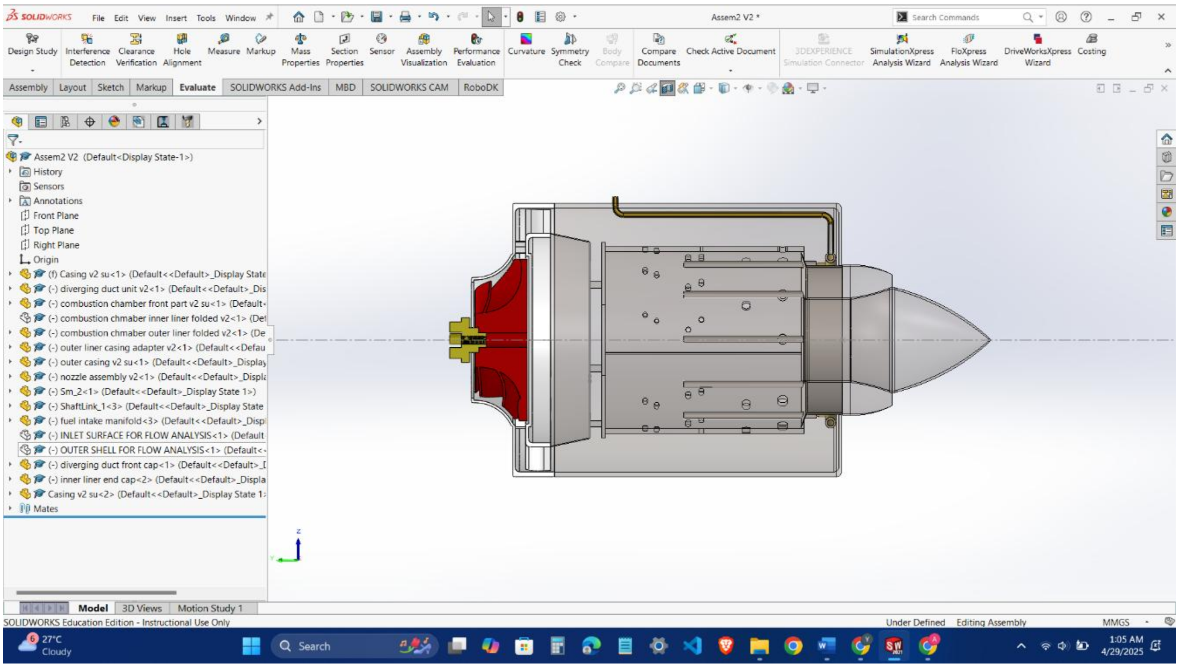


Fig19. Final prototype adjusted to available part dimensions

FINAL PROTYPE ADJUSTED TO AVAILABLE PART DIMENSIONS

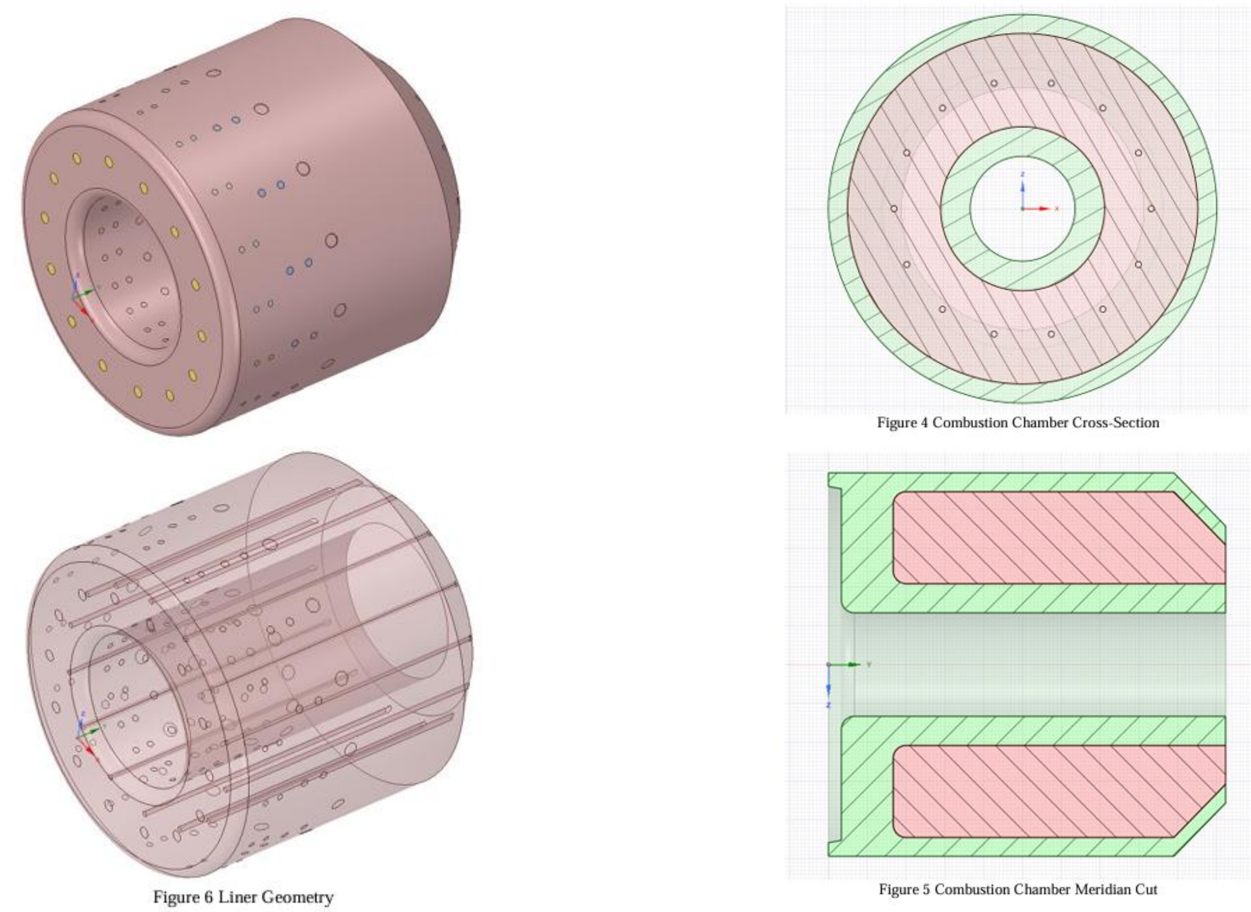


Fig20. Green is reference area and bronze is linear area

Result and observation:

1. Objective

The primary goal of this analysis is to perform a comprehensive thermodynamic cycle analysis for a 300 N thrust micro jet engine using the Brayton-Joule cycle. Key design parameters include:

- Air mass flow rate
- Compressor pressure ratio
- Cycle temperatures
- Geometric dimensions for the compressor, combustion chamber, and nozzle
- Material selection suitable for university-level manufacturing

2. Mass Flow Rate and Reference Parameters

For a mass flow rate of 0.065 kg/s and a reference velocity of 0.03 Mach, the following reference values are calculated:

- Reference Velocity: (V = 10.2 , m/s)
- Reference Dynamic Pressure: (q = 146.2 , Pa)
- Reference Area: (A = 2.26 \times 10^{-3} , m^2)

3. Pressure Loss Parameters

The combustion chamber design takes into account pressure losses from both the diffuser and liner. The following equations are utilized to derive these parameters:

1. **Combustor Pressure Loss:**

$$\frac{\Delta p_{t,3-4}}{p_{t3}}$$

2. **Combustor Pressure Drop Factor:**

$$\frac{\Delta p_{t,3-4}}{q_{\text{ref}}}$$

3. **Diffuser Pressure Loss:**

$$\Delta p_{t,\text{diff}}$$

4. **Liner Pressure Drop Factor:**

$$\frac{\Delta p_{t,L}}{q_{\text{ref}}}$$

5. **Liner Pressure Loss:**

$$\Delta p_L$$

4. Design Requirements

The design of the combustion chamber liner aims for a turbojet engine producing 200 N thrust with an air mass flow of 0.55 kg/s. Key targets include:

- Air mass distribution:
    - Primary: 20%
    - Secondary: 30%
    - Dilution: 50%
- Allowable pressure drop: 4% – 10%

5. Thermodynamic Calculations

Using the output conditions from the compressor and turbine, the following reference values are estimated:

- Reference Area
- Airspeed
- Mach Number
- Dynamic Pressure

6. Analytical and Empirical Calculations

Using Lefebvre's method, the following parameters are determined:

- Liner Dimensions: Length and diameters
- Zone-wise Liner Lengths: Primary, secondary, dilution
- Cross-sectional Areas: For each zone
- Hole Size and Number: For each zone
- Airflow Distribution: Into each zone
- Pressure Loss: Due to liner geometry

7. Theoretical Background

According to Lefebvre's analytical rules, critical factors for fuel injector placement in the primary zone include:

- Stable Flame Anchoring: Achieved through a recirculation zone using swirl or reverse flow.
- Proper Atomization and Mixing: Essential for ignition and sustained combustion.
- Recirculation Benefits: Reintroduces hot combustion products to sustain ignition.

Conclusion:

This project has effectively demonstrated a comprehensive understanding of fundamental concepts, operational principles, analytical calculations, design methodologies, numerical simulations, and the functionality of jet engines. We calculated the operational parameters necessary to achieve the desired thrust, including the diverging duct for pressure enhancement, reference area in relation to Mach number, liner area and length concerning pressure losses, hole diameter in relation to fuel-air ratio, and respective air speeds in various zones, as well as the nozzle with an optimal converging-diverging ratio and the placement of injection tubes for optimal recirculation and mixing.

Using a laser cutter, we cut unfolded conical projections of various shapes, and we intend to bend them to create more complex shapes like nozzles, diverging ducts, etc. We separated the hottest combustion chamber part from the 3D printed diffuser, used a ring-shaped fuel injector, and used sheet metal for the majority of the parts. Finally, we came up with a hybrid that is the most feasible to fabricate out of the various configurations from existing micro jet engine models. In order to test thrust, we purchased load cells.

We are halfway through our entire design plan and will be finished in three days, but all of a sudden, outside forces caused the progress to accelerate to an astounding level. We sincerely apologize for not finishing the project when we left the college; we will work on the next phase and make up the lost time the following semester. The challenges and objectives you set for us have changed our knowledge and wisdom. Thank you, Drs. Rajendra Kumar Munian and Srikant Sekhar Padhee, for your unwavering support, crucial guidance, and faith even during difficult times of progress.

Due to the expiry of the licences of Solid Works and Ansys ,outside the campus we could not access The cad views and dxf files for every part in this report are limited.

CAD files for An old version can be found at "https://github.com/BillKerman/MicroJetEngineDevelopmentEngineeringProject/tree/main/CAD%20Files"

Further work to be done:

1. Perform vibration and rotodynamic analysis of the compressor assembly.

2. Perform rigorous CFD analysis and optimise dimensions and performance of compressor and combustion chamber
3. Conduct extensive off-design performance testing and validate with experimental data.
4. Optimize the fuel injection system to further improve combustion efficiency.
5. Explore advanced cooling techniques (e.g., film or transpiration cooling) to enhance thermal management.
6. Integrate real-time sensor feedback for adaptive control of the engine during flight.4

7. Laser cutting (cutting, bending, welding 3d printing)

8. New compatible compressor

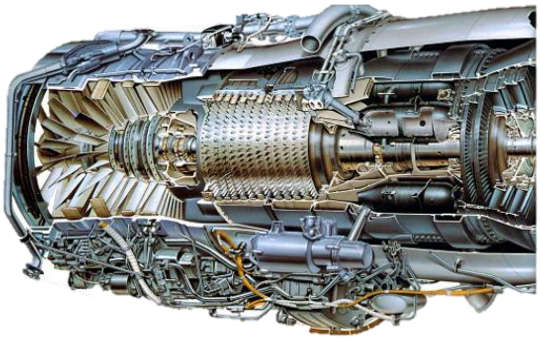
9. Spark plug and fuel injection review

10. Final assembly, Test trial

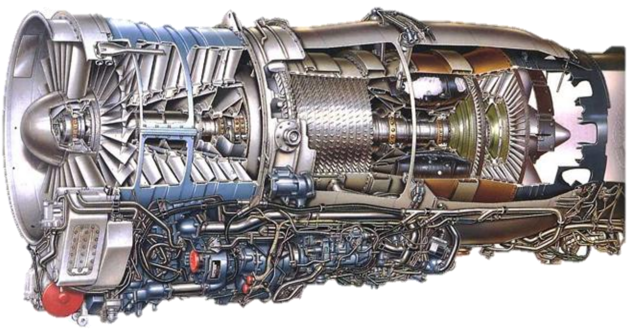
Test stand and enclosure and add experimental and theoretical results will be compared and necessary modifications and adjustments all be done until we achieve permissible error value and later we will take this project to the next stage.

References:

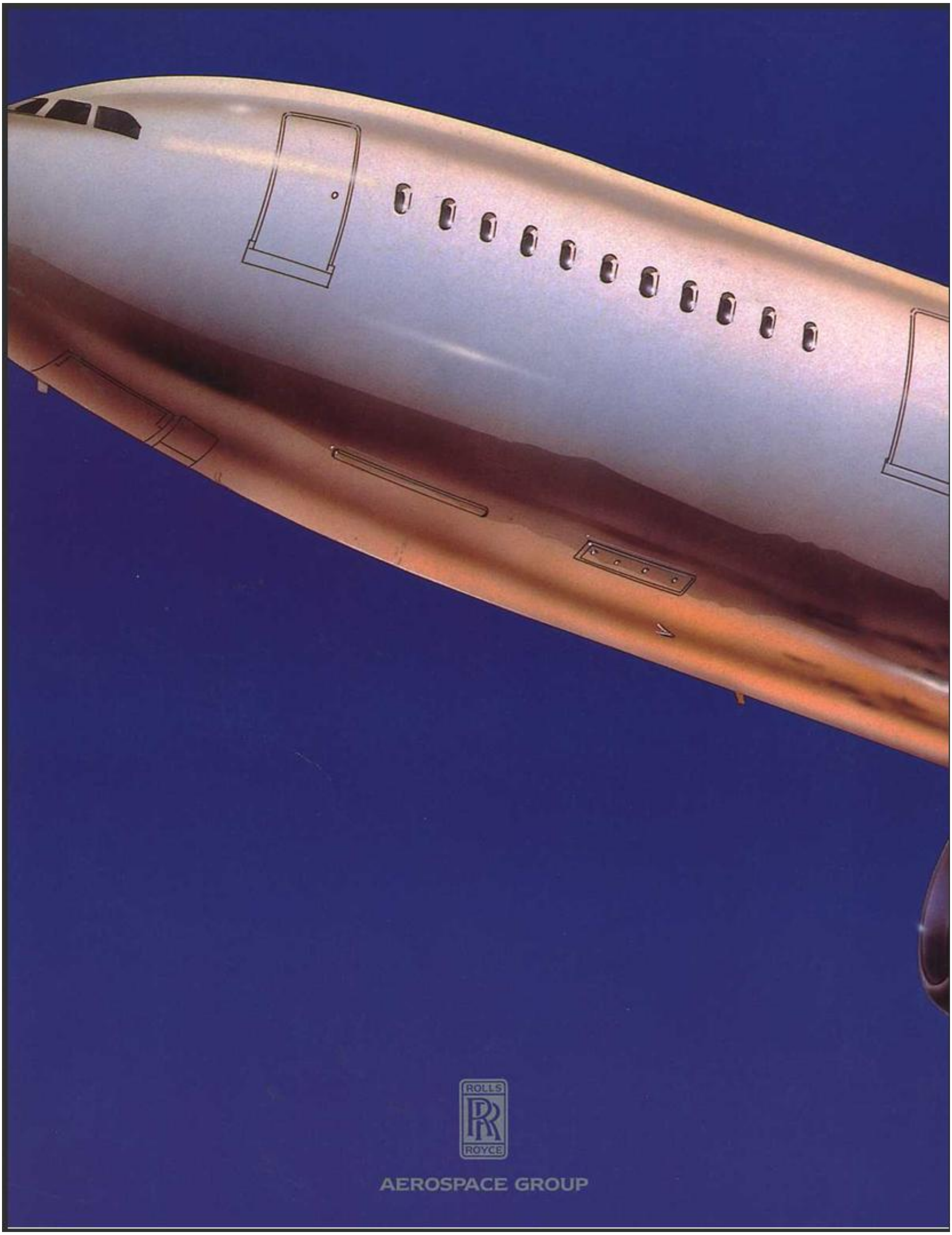
- Benini, E., & Giacometti, S. (2007). *Design, manufacturing and operation of a small turbojet engine for research purposes*. Applied Energy, 84, 1102–1116.
- Rashid S. E. Mohammed, (2019). Design and Analysis of Annular Combustion Chamber for a Micro Turbojet Engine. International Journal of Aerospace 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.
- THE JOHN ZINK HAMWORTHY COMBUSTION HANDBOOK SECOND EDITION Volume 3 APPLICATIONS.
- Design and Analysis of Annular Combustion Chamber for a Micro Turbojet Engine Cahya Amalinadhi Putra - 23619305
- Putra, C. A. (2020). *Design and Analysis of Annular Combustion Chamber for a Micro Turbojet Engine*. Bandung Institute of Technology.
- Design of Mini Turbojet Engine Combustion Chamber Liner
- With 200N Static Thrust by (Rais Ryacudu Santoso, Vicky Wuwung, Reina Fadjrin Nurul Annisa, Radi Suradi Kartanegara).
- 3rd Gas Turbine Combustion Alternative Fuels and Emissions by (Arthur H. Lefebvre and Dilip R. Ballal).
- 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 RB168 Mk202/M



Rolls-Royce RB183 Mk 555


ROLLS
RR
ROYCE
AEROSPACE GROUP