Design and Analysis of Annular Combustion Chamber for a Micro Turbojet Engine

Cahya Amalinadhi Putra
Bandung Institute of Technology

Design and Analysis of Annular Combustion Chamber for a Micro Turbojet Engine

Cahya Amalinadhi Putra - 23619305

AE 5014 – Aerodinamika Propulsi – 30 April 2020

Abstract

This report deals with the preliminary design of a combustion chamber as a further study of the initial design of a micro turbojet engine that has been finished on parametric cycle analysis (PCA) step. The calculation includes reference values evaluation, combustion chamber, and its component dimension, air mass flow distribution, and sizing of the orifices in the combustion chamber. This preliminary design later is compared with numerical simulation using computational fluid dynamics (CFD) software. On-design and off-design analysis are presented in this report.

Keywords – annular combustion chamber, micro turbojet engine, CFD

1. 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. Micro turbojet engines usually use an annular combustion engine because of having several advantages such as ease of ignition, minimum cross-section area, minimum length and weight, minimum pressure drop (around 5%), and need less cooling air.

This report shows the result of the preliminary design of an annular combustion chamber for a micro turbojet engine of 1,200 N. The design process depends on reliable database reference and empirical correlation from the experiment. The approach process will not present the most accurate result, hence the preliminary design will further be analysed on design point and off-design by CFD.

2. Preliminary Design

2.1. 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).

2.2. Design Specification

The initial design parameters from PCA becomes the input of preliminary design. We will focus on station at compressor exit (station 3) and turbine inlet (station 4).

Table 1 Initial Design Parameters

Symbol	Quantity	Value	
m_a	Air mass flow rate	2.47	Kg/s
T_{t3}	Comp. exit total temp.	475	K
p_{t3}	Comp. exit total press.	385,035	Pa
m_f	Fuel mass flow rate	0.0552	Kg/s
T_{t4}	Turb. inlet total temp.	1,175	K
p_{t4}	Turb. inlet total press.	346,532	Pa

2.3. 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.

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.

2.5. Diffuser sizing

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.

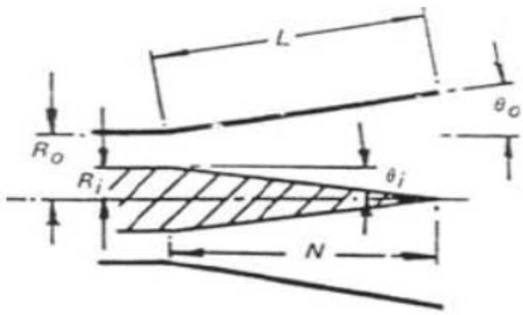

Figure 1 Equiangular Annular Dump Diffuser

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° and 12°. To find the aspect ratio, we can use the below equation.

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

To start the design process, we use the geometry of the compressor exit (inner diameter and outer diameter as R_i and R_o). 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.

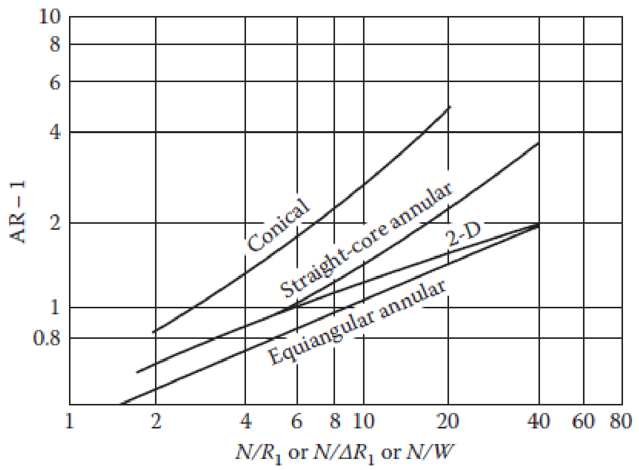

Figure 2 Lines of First Stall

Table 2 Diffuser Sizing and Calculation

Symbol	Quantity	Value	
Input			
D_{out}	Outer exit comp. dia.	0.2963	m
D_{in}	Inner exit comp. dia.	0.2752	m
θ_o	Outer divergence angle	0	deg
θ_i	Inner divergence angle	10	deg
L	Wall length	0.01	m
Output			
AR	Aspect Ratio	3.60	-
N	Axial length	0.01	m

Table 2 shows the sizing and calculation of the diffuser used in this micro turbojet combustion chamber. The outer divergence angle is set to 0° to prevent the enlargement of engine diameter. 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.

Table 3 Additional Variables in Diffuser

Symbol	Quantity	Value	
ρ_{t3}	Exit comp. total density	2.81	Kg/m^3
$U_{dif\text{-}inle}$	Diffuser inlet velocity	98.8	m/s
$U_{dif\text{-}outle}$	Diffuser outlet velocity	25.8	m/s

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 p_{t4} and p_{t3} or π_B), air mass flow, and geometry of compressor exit and turbine inlet become the input of calculation.

Reference values

The reference area (A_{ref}) 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.

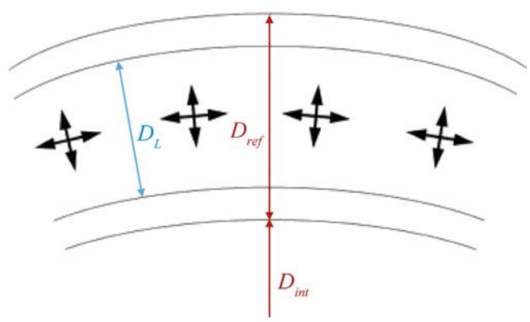

Figure 3 Geometry of Reference Area

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

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.

$$U_{ref} = \frac{\dot{m}_a}{\rho_{t3} A_{ref}} \quad (3)$$

$$q_{ref} = \frac{1}{2}\rho_{t3} U_{ref}^2 \quad (4)$$

Table 4 Reference Values Calculation

Symbol	Quantity	Value	
Input			
D_{ref}	Ref. diameter	0.2963	m
D_{int}	Int. diameter	0.0800	m
Output			
A_{ref}	Ref. area	0.0639	m^2
U_{ref}	Ref. velocity	13.75	m/s
a_{ref}	Ref. speed of sound	426.78	m/s
M_{ref}	Ref. mach number	0.03	-
q_{ref}	Ref. dynamic pressure	266	Pa

<u>Pressure loss parameters calculation</u>

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.

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

The combustor pressure loss ($\Delta p_{3-4}/p_{t3}$) is obtained by solving eq. (5). This value later is used to find the pressure loss on the diffuser and the liner as explained in eq. (6).

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

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).

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

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

After calculating throughout the equation, we will obtain the pressure loss parameters as stated in the table below.

Table 5 Pressure Loss Parameters Calculation

Symbol	Quantity	Value	
Input			
π_B	Burner press. ratio	0.90	-
Output			
$\Delta p_{t,3-4}/p_{t3}$	Combustor press. loss	0.10	-
$\Delta p_{t,3-4}/q_{ref}$	Combustor press. drop factor	144.7	-
$U_{in,diff}$	Avg. velocity in diffuser	92.83	m/s
λ	Dump diffuser constant	0.45	-
$\Delta p_{t,diff}$	Total press. loss in diffuser	3,356	Pa
$\Delta p_{t,diff}/q_{ref}$	Diffuser press. drop factor	12.6	-
$\Delta p_{t,L}/q_{ref}$	Liner press. drop factor	132.1	-
$\Delta p_{t,L}$	Total press. loss in liner	35,147	Pa

<u>2.7. Liner sizing</u>

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

$$A_L = k A_{ref} \quad (9)$$

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

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

Where

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

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

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

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

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

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 in the table below.

Table 6 Liner Sizing Calculation

Symbol	Quantity	Value	
Input			
k	Liner to reference area ratio	0.70	-
$D_{L,in}$	Inner liner dia.	0.1200	m
$D_{L,out}$	Outer liner dia.	0.2672	m
$T_{t,max}$	Maximum outlet local temp.	1,210	K
Output			
D_L	Liner dia. or flame tube height	0.0736	m
PF	Pattern factor	0.05	-
L_L	Liner length	0.2170	m
L_{PZ}	Primary zone length	0.0552	m
L_{SZ}	Secondary zone length	0.0368	m
L_{DZ}	Dilution zone length	0.1251	m

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.

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

Below is the table of airflow distribution calculation.

Table 7 Airflow Distribution Calculation

Symbol	Quantity	Value	
Input			
m_a	Air mass flow	2.4737	Kg/s
m_f	Fuel mass flow	0.0552	Kg/s
α_{PZ}	Primary zone AFR	0.9	-
α_{SZ}	Secondary zone AFR	1.7	-
k_{PZ}	Ratio of airflow that enter axially to total in primary zone	0.3	-
Output			
$m_{PZ,ax}$	Primary zone airflow – axial	0.2201	Kg/s
$m_{PZ, ann}$	Primary zone airflow – annular	0.5135	Kg/s
m_{SZ}	Secondary zone airflow	1.3857	Kg/s
$m_{SZ,jet}$	Secondary zone airflow – jet	0.6521	Kg/s
m_{DZ}	Dilution zone airflow	2.4737	Kg/s
$m_{DZ,jet}$	Dilution zone airflow – jet	1.0880	Kg/s

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 d_j. Then to calculate the size of the holes, we can use the following equation.

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

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

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

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

Table 8 Liner Holes Calculation

Symbol	Quantity	Value	
Primary Zone			
$d_{j,ax}$	Holes diameter, axial	9.00	mm
n_{ax}	A half no. of holes, axial	14	-
$d_{j,ann}$	Holes diameter, annular	4.74	mm
$n_{ann,1}$	A half no. of holes, axial, row-1	17	-
$n_{ann,2}$	A half no. of holes, axial, row-2	17	-
Secondary Zone			
$d_{j,SZ}$	Holes diameter	6.15	mm
$n_{SZ,1}$	A half no. of holes, row-1	13	-
$n_{SZ,2}$	A half no. of holes, row-2	13	-
Dilution Zone			
$d_{j,DZ}$	Holes diameter	11.00	mm
n_{DZ}	A half no. of holes	14	-

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 Y_{max} is a maximum penetration depth (a ratio to liner diameter). Once the Y_{max} has decided, then the effective diameter can be obtained. After that, we have to find the real hole's diameter by using eq. (21). C_D is the discharge coefficient.

We use typical value of $C_D = 0.6$. After solving eq. (18) to eq. (21), we will get the liner holes size as listed in table 8.

3. Simulation Setup

3.1. CAD Model

The first step is to create a model of our combustion chamber. The model is created with the help of ANSYS SpaceClaim. Below is the model of the combustion chamber.

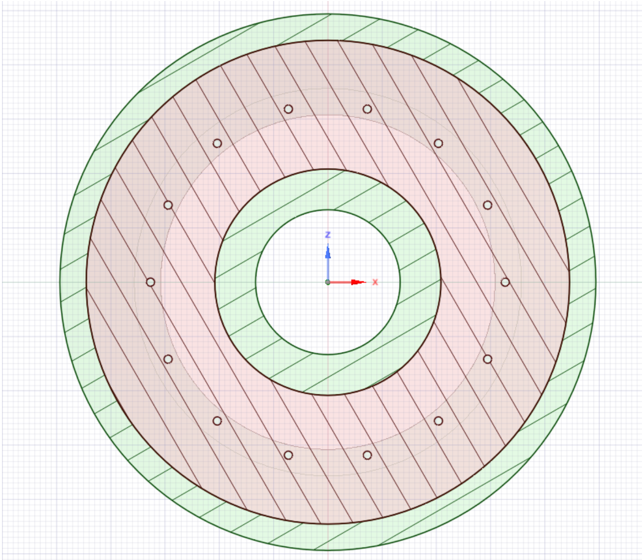

Figure 4 Combustion Chamber Cross-Section

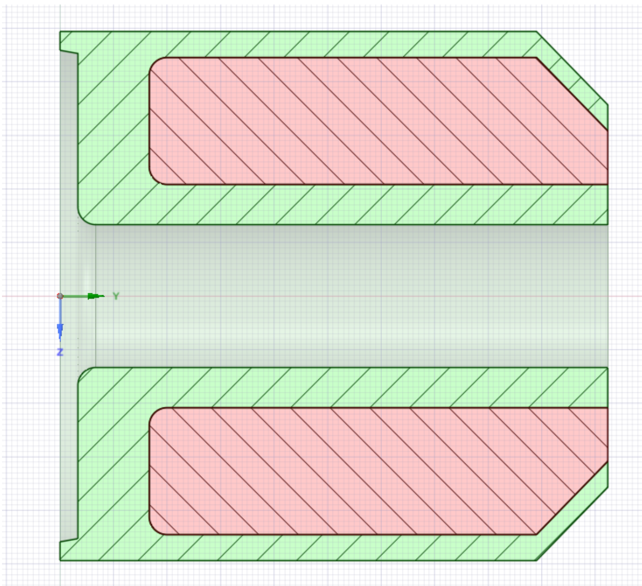

Figure 5 Combustion Chamber Meridian Cut

From figure 4 and figure 5, the liner is represented with the red region while the rest is represented with green color.

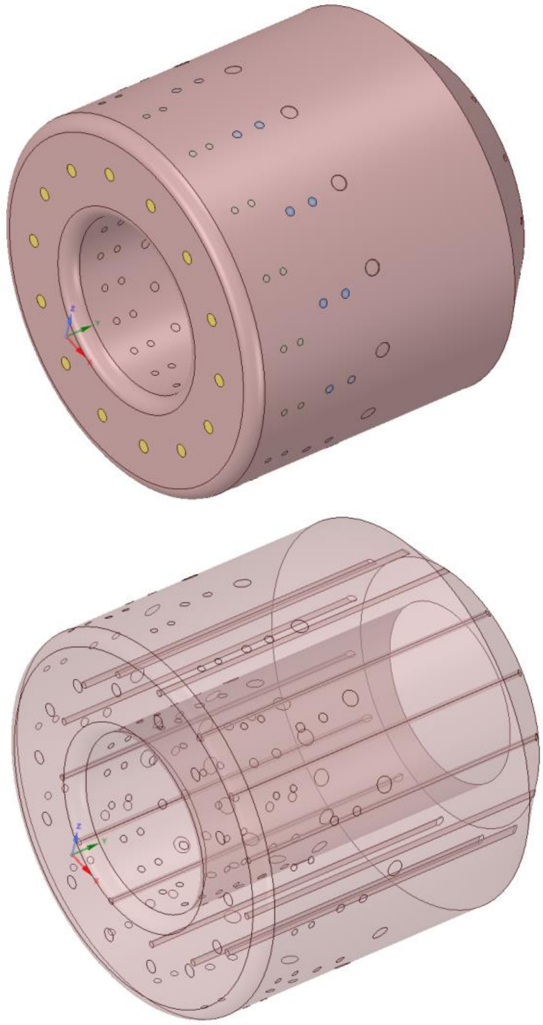

Figure 6 Liner Geometry

Figure 6 illustrates the liner geometry. In the liner, there are several air holes to administrate air flows into the liner. In Figure 6, above, the yellow holes are the primary zone axial-holes while the green holes around the liner circumference are the primary zone-annulus holes. The blue ones are the secondary zone holes and the rest are the dilution zone holes. As we know from the preliminary design section, we only know the region of each zone without knowing the exact position of holes in each zone. This is a kind of freedom given to the designer to choose the right location thus could maximize the combustion chamber performance. The holes not only are distributed in the outer part of the liner but also the inner part of the liner. The designer must aware and make sure that the holes in the inner part of the liner do not intersect with each other.

Further, there are some pipes inside the liner. This pipe is for transferring fuel from outside to liner. To make the drawing process easier, we make the pipe as simple as possible which is drawn out from downstream to upstream. The fuel is directed to flow oppositely with the airflow so that it can induce the mixing process and lower both of their velocity. Also, the fuel is flowed directly to the primary zone thus start the ignition process.

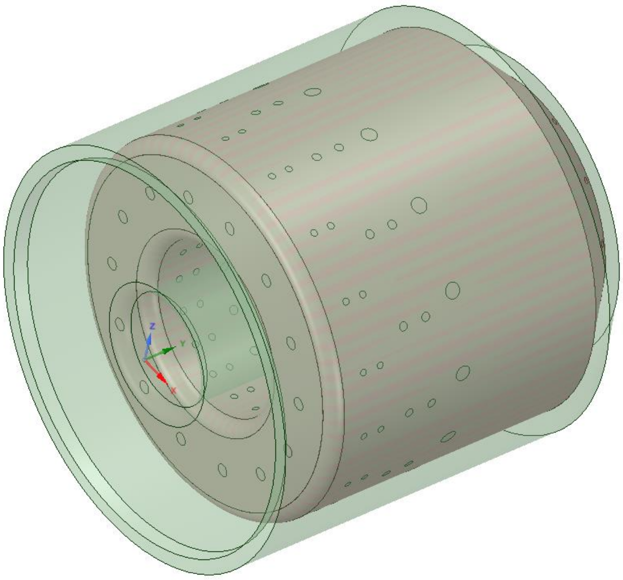

Figure 7 Full Combustion Chamber Geometry

3.2. Mesh

The mesh is created by using ANSYS Meshing. The mesh is unstructured with quad dominant.

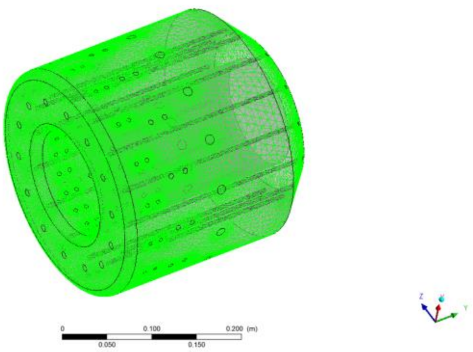

Figure 8 Liner Mesh

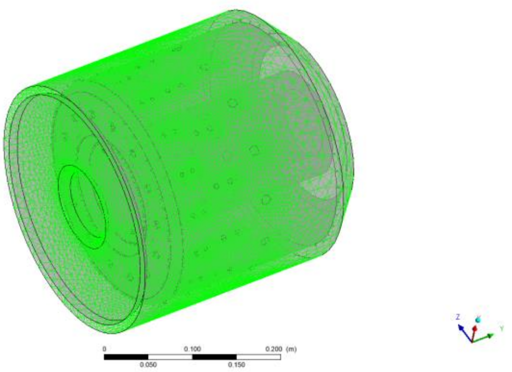

Figure 9 Casing Mesh

Figure 8 and figure 9 illustrate the mesh of liner and casing of the combustion chamber. Due to the limitation of computation performance, we could not generate mesh larger than 2.5 million elements. In this simulation, we will use mesh with 2.1 million elements.

3.3 CFD Setup

The setup is solved with ANSYS CFX. Further, below are the details about simulation assumption, boundary condition, initial condition, and convergence criterion.

Simulation assumption

Below are some assumptions used in this simulation.

1. Steady-state analysis type.
2. Reacting mixture simulation.
3. Solve the mass, momentum, and energy equation.
4. Use the k-epsilon turbulence model.
5. Consider the thermal radiation effect.
6. The reaction is JetA air single step and NO formation.
7. Use a thin wall model assumption.
8. Not performing a grid independence check.
9. Uniform airflow at the inflow boundary.

Boundary Condition

There are 3 kinds of boundary conditions, inlet (for air and fuel), outlet, and wall, respectively.

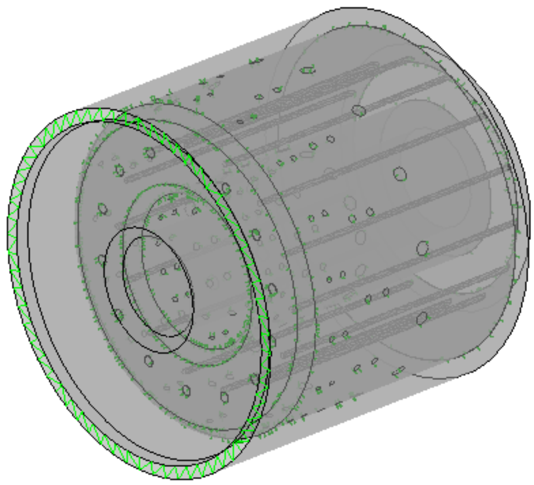

Figure 10 Air Inlet Boundary (Green Color)

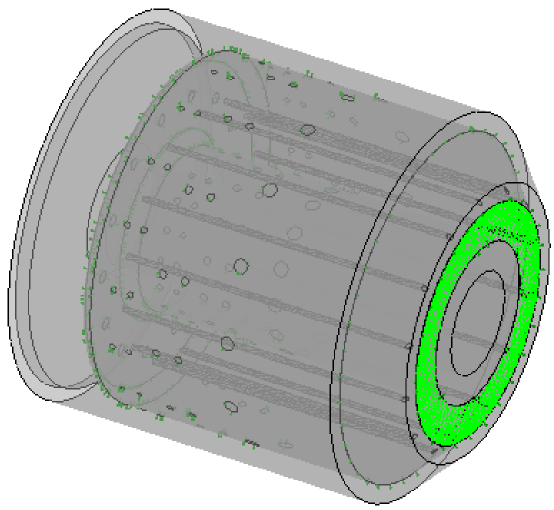

Figure 11 Outlet Boundary (Green Color)

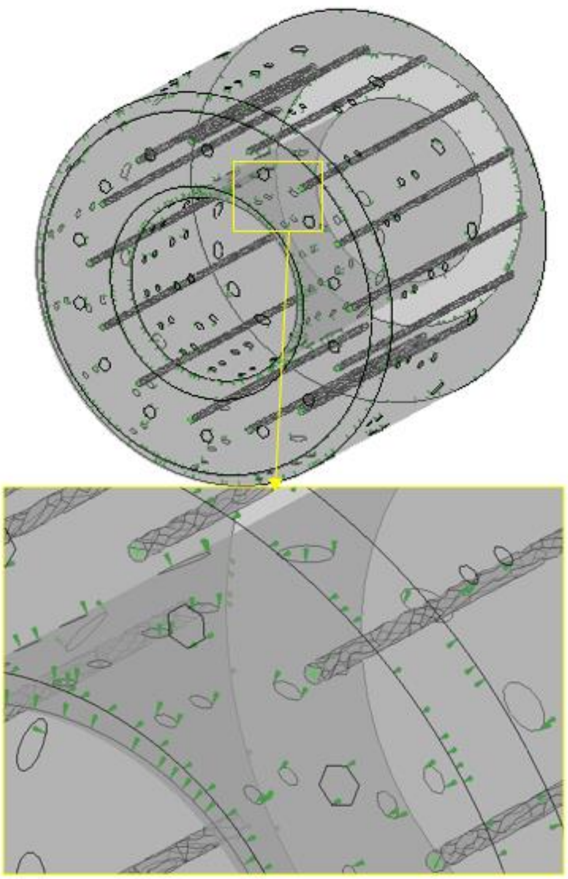

Figure 12 Fuel Inlet Boundary (Green Color)

The rest is defined as a wall. Further, below is the detailed condition in each boundary.

Table 9 Boundary Condition Values

Symbol	Quantity	Value	
Air Inlet			
m_{dot}	Mass flow rate	2.4737	Kg/s
T_t	Total temperature	475.56	K
Fuel Inlet			
m_{dot}	Mass flow rate	0.0522	Kg/s
T	Static temperature	300.00	K
Outlet			
p	Average static pressure	3.3	atm
Wall			
Type	No slip wall		

Initial condition

The initial condition is an important step to perform the main simulation. Usually, the initial condition is obtained by performing cold flow analysis (flow only without any combustion process). However, due to the limitation of time, we do not perform the cold flow analysis, instead, we set velocity as the initial condition. We set the whole domain to have 5 m/s velocity from upstream to downstream.

Convergence criterion

For the continuity and momentum equation, root means square (RMS) error under 1.0 x 10^{-3} is sufficient. However, if we add the energy equation, usually we need the RMSE to converge below 1.0 x 10^{-5}. This is the main convergence criterion. However, if the simulation does not converge on the specific RMSE value, we could deduce that the simulation already converges if our interested variables RMSE are already levelled.

4. Results and Discussion

4.1. On-design

The on-design analysis boundary conditions have already explained in the previous section. During the solving process, the RMSE did not less than 1.0 x 10^{-5} thus we stop the simulation after making sure that our important variables are already levelled. Table 10 listed the comparison between analytic and CFD simulation.

Table 10 On-Design Comparison

Symbol	Quantity	Analytic	CFD	Error
Mass flow		**Kg/s**	**Kg/s**	**%**
m_{PZ}	Primary zn.	0.7336	0.7461	1.70
$m_{PZ,ax}$	Primary zn. – axial	0.2201	0.2820	28.14
$m_{PZ,am}$	Primary zn. – annular	0.5135	0.4641	9.64
m_{SZ}	Secondary zn.	0.6521	0.6074	6.85
m_{DZ}	Dilution zn.	1.0880	1.0716	1.51
Temperature		**K**	**K**	**%**
T_{t3}	Burner inlet	476	476	0.00
$T_{t4,ave}$	Average burner outlet	1,175	1,112	5.39
$T_{t4,max}$	Max. burner outlet	-	1,415	-
T_{max}	Max. flame temp.	-	2,559	-
Pressure		**Pa**	**Pa**	**%**
p_{t3}	Burner inlet	385,035	415,633	7.95
p_{t4}	Burner outlet	346,532	349,931	0.98
Mach		-	-	**%**
$M_{4,ave}$	Average outlet	0.25	0.26	4.84
$M_{4,max}$	Maximum outlet	-	0.28	-
Efficiency		-	-	**%**
η_B	Burner efficiency	0.85	0.78	8.39

Almost all the variables are having error under 10% except for the primary zone – axial airflow (error around 28.14%). However, if we add all the primary zone airflow, we get an error of 1.70%. This means our analytical prediction of mass flow through the primary zones holes is not accurate. Furthermore, the error of burner efficiency is around 8.39%. This phenomenon can happen due to numerical error (mesh, turbulence model, chemical reaction, and effect of radiation).

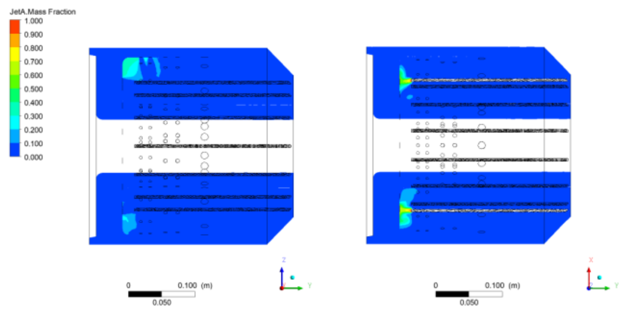

Figure 13 JetA Contour On-Design

The fuel comes out from the fuel line that flows oppositely from the downstream. From figure 13, the contour of fuel already dispersed near the primary zones area which is a good sign because the fuel will be mixed automatically. At the dilution zone, there is no indication of fuel mass fraction which also a good sign because all of the fuel has already been burned.

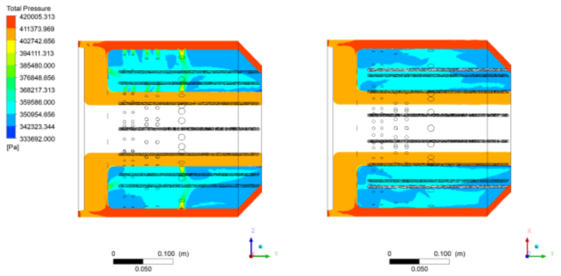

Figure 14 Total Pressure Contour On-Design

Figure 14 illustrates the total pressure contour inside the combustion chamber. It clearly shows a high-pressure drop occurs inside the liner. It creates suction to the outer of liner thus the air is sucked inside. The high error between annulus and axial primary zone mass flow might arise from this pressure contour. It shows that the axial and inner part of the combustion chamber is more suction than the outer part of the combustion chamber.

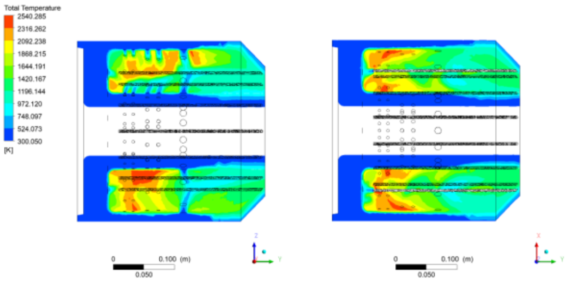

Figure 15 Total Temperature Contour On-Design

Further, the burning process is located in the primary and secondary zone. It can be seen clearly by the total temperature contour that is illustrated in figure 15. The maximum temperature at the burning process reaches 2,500 K. As a consequence, we need to choose a material that holds the high thermal stresses. After passing the dilution zone, the total temperature of mixed flow decreases and reaches around 1,111 K. This indicates the dilution zone sizing is functioned properly.

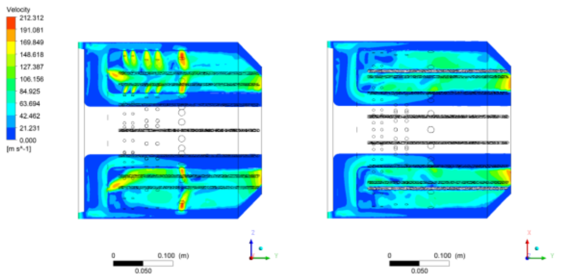

Figure 16 Velocity Contour On-Design

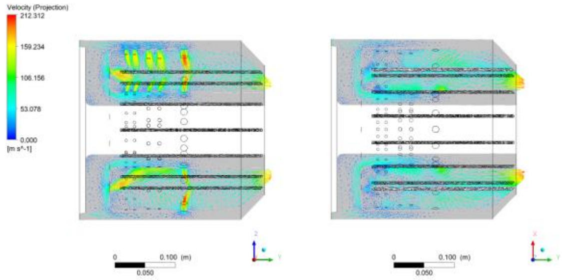

Figure 17 Velocity Vector On-Design

Based on figure 16 and 17, the whole admission holes flow reach almost half-length of liner diameter. For the primary and secondary zone, this configuration is helping the mixing and burning process because those processes need high turbulence to make it more effective. However, it seems that the primary zone axial airflow enters the liner with high enough velocity that can turn the fuel flow back to the fuel line. Further, along with the downstream, there is no indication of high swirling flow.

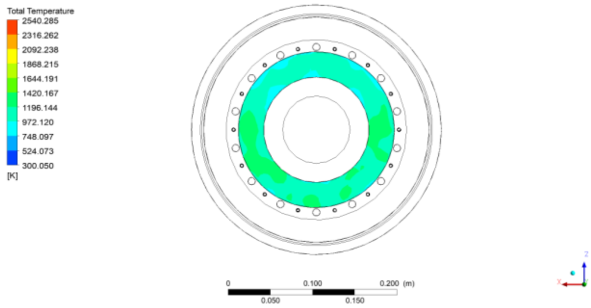

Figure 18 Outlet Total Temperature Contour On-Design

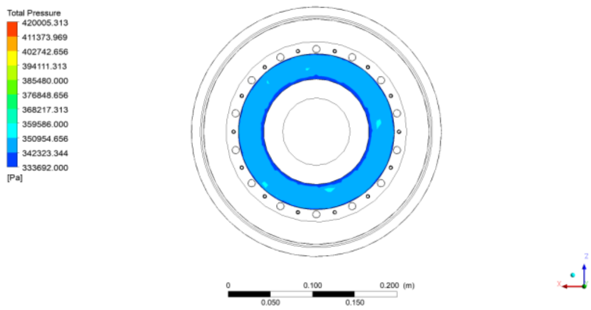

Figure 19 Outlet Total Pressure Contour On-Design

From Figures 18 and 19, total temperature and pressure are evenly distributed. This is a good sign for the input of the next component. Moreover, the burner efficiency

4.2. Off-design

The off-design analysis will calculate the case of 1) mass flow drops 10% and 2) based on CFD analysis of the prior component. Again, both of the cases did not reach convergence RMSE less than 1.0 x 10^{-5} hence we stop the iteration after our important variables have levelled. Table 11 listed the comparison between analytic and CFD simulation.

Case-1: Air mass flow drop 10%

Table 11 Off-Design Case 1 Comparison

Symbol	Quantity	Analytic	CFD	Error
Mass flow		**Kg/s**	**Kg/s**	**%**
m_{PZ}	Primary zn.	0.7336	0.6740	8.13
$m_{PZ,ax}$	Primary zn. – axial	0.2201	0.2550	15.86
$m_{PZ,am}$	Primary zn. – annular	0.5135	0.4190	18.41
m_{SZ}	Secondary zn.	0.6521	0.5840	15.97
m_{DZ}	Dilution zn.	1.0880	0.9700	10.85
Temperature		**K**	**K**	**%**
T_{t3}	Burner inlet	476	476	0.00
$T_{t4,ave}$	Average burner outlet	1,175	1,227	4.42
$T_{t4,max}$	Max. burner outlet	-	1,503	-
T_{max}	Max. flame temp.	-	2,541	-
Pressure		**Pa**	**Pa**	**%**
p_{t3}	Burner inlet	385,035	406,314	5.53
p_{t4}	Burner outlet	346,532	342,084	4.42
Mach		-	-	**%**
$M_{4,ave}$	Average outlet	0.25	0.26	4.84
$M_{4,max}$	Maximum outlet	-	0.28	-
Efficiency		-	-	**%**
η_B	Burner efficiency	0.85	0.91	6.89

Based on table 11, we see that the whole of airflow has decreased. This can affects combustion performance. The table also indicates an increasing temperature in the burner outlet that means a decrement performance in dilution zones.

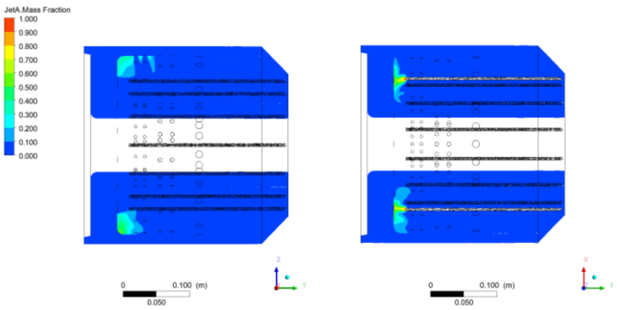

Figure 20 JetA Contour Off-Design Case-1

From figure 20, the fuel distributed according to the plan which is only in primary and secondary zones.

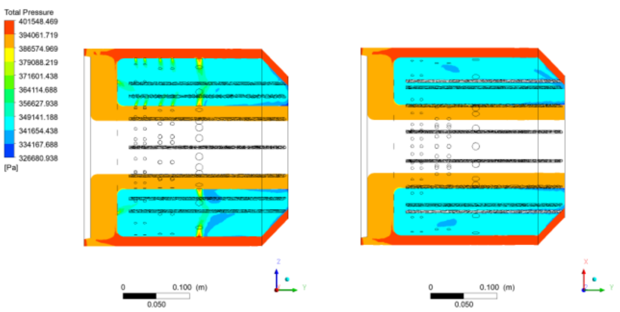

Figure 21 Total Pressure Contour Off-Design Case-1

The total pressure drop of off-design case-1 is bigger than the on-design. Similar to on-design analysis, in figure 21, the inner pressure is more suction than the outer region.

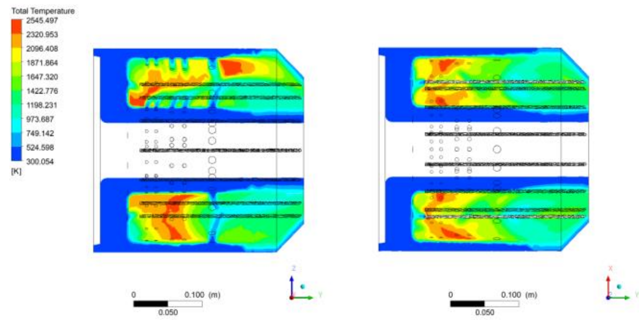

Figure 22 Total Temperature Contour Off-Design Case-1

The combustion process mainly occurs in the secondary zone. This can happen because of an insufficient amount of air in the primary zone. As a consequence, there is high temperature occurs on the secondary zone and near the dilution zone. Due to a lack of air mass flow on the dilution zone, the cooling down process is ineffective and create a bigger outlet temperature.

Due to the decrement value of air mass flow, the jet penetration depth has to decrease. Further, in Figures 23 and 24, the outlet velocity decreases. This can affect the targeted thrust.

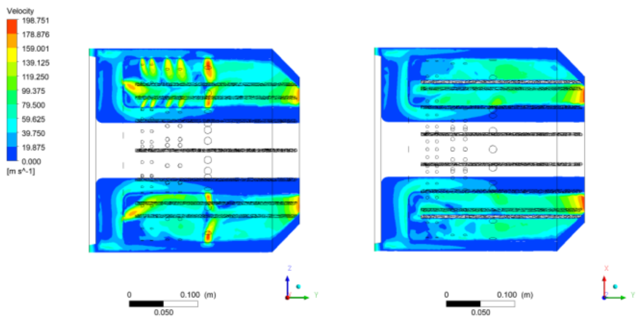

Figure 23 Velocity Contour Off-Design Case-1

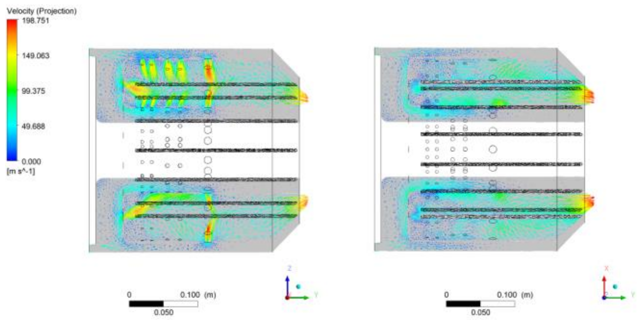

Figure 24 Velocity Vector Off-Design Case-1

Furthermore, similar to the on-design analysis, the total temperature and total pressure at the outlet are distributed evenly as shown in figure 25 and figure 26. This is a good sign for the turbine input.

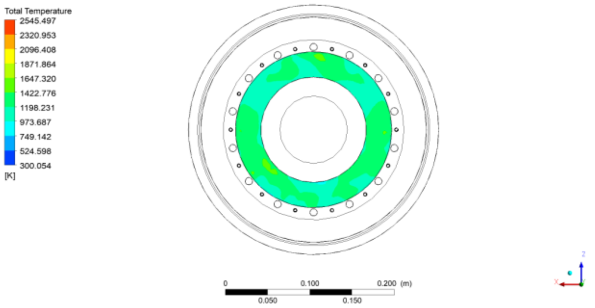

Figure 25 Outlet Total Temperature Contour Off-Design Case-1

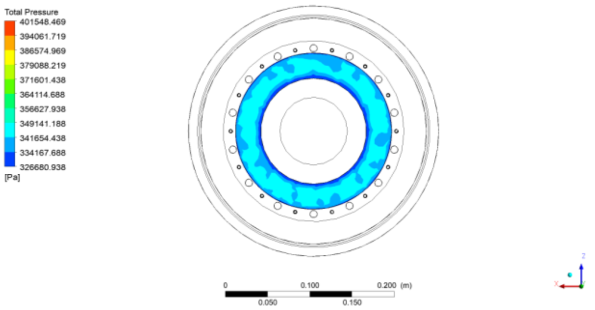

Figure 26 Outlet Total Pressure Contour Off-Design Case-1

In conclusion, when the air mass flow is dropped around 10%, we would not have sufficient velocity to get the interested thrust and also has a bigger total temperature at the burner outlet which is bad for turbine structures.

Case 2: Lower inlet total pressure

This is not a real off-design case, however, due to the independent CFD analysis per component, there must be some numerical errors that arise from the previous component of the combustion chamber. Hence, we will call these results as an off-design.

Table 12 Off-Design Case 2 Comparison

Symbol	Quantity	Analytic	CFD	Error
Mass flow		**Kg/s**	**Kg/s**	**%**
m_{PZ}	Primary zn.	0.7336	0.6740	1.69
$m_{PZ,ax}$	Primary zn. – axial	0.2201	0.2550	9.65
$m_{PZ,am}$	Primary zn. – annular	0.5135	0.4190	28.13
m_{SZ}	Secondary zn.	0.6521	0.5840	6.76
m_{DZ}	Dilution zn.	1.0880	0.9700	1.38
Temperature		**K**	**K**	**%**
T_{t3}	Burner inlet	476	476	0.00
$T_{t4,ave}$	Average burner outlet	1,175	1,150	2.17
$T_{t4,max}$	Max. burner outlet	-	1,397	-
T_{max}	Max. flame temp.	-	2,551	-
Pressure		**Pa**	**Pa**	**%**
p_{t3}	Burner inlet	385,035	406,314	6.12
p_{t4}	Burner outlet	346,532	344,013	0.73
Mach		-	-	**%**
$M_{4,ave}$	Average outlet	0.25	0.28	12.00
$M_{4,max}$	Maximum outlet	-	0.29	-
Efficiency		-	-	**%**
η_B	Burner efficiency	0.85	0.82	3.38

From centrifugal compressor CFD analysis, there is a decrement in total pressure at the compressor outlet however, the total temperature at the compressor outlet is relatively the same. Table 12 is listed in the comparison between analytic and CFD.

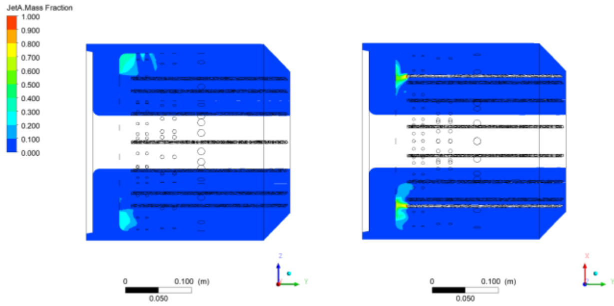

Figure 27 JetA Contour Off-Design Case-2

Based on figure 27, the fuel distribution is similar to on-design or off-design case 1. There is no indication of fuel mass fraction on the dilution zone, hence the combustion process only occurs in the primary and secondary zone.

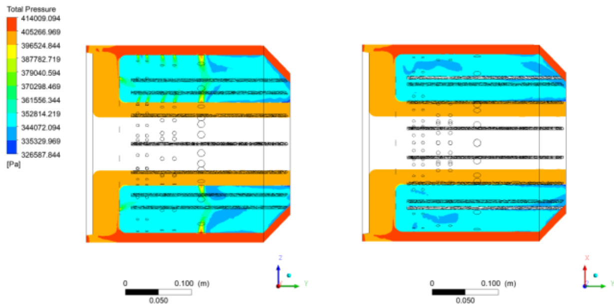

Figure 28 Total Pressure Contour Off-Design Case-2

In case-2, the compressor compression ratio does not reach the PCA analysis (around 3.73 of 4.00). From figure 28, it shows a larger pressure drop region after dilution zone if we compare with off-design case 1, but smaller region than on-design. This makes a lower pressure drop throughout the liner.

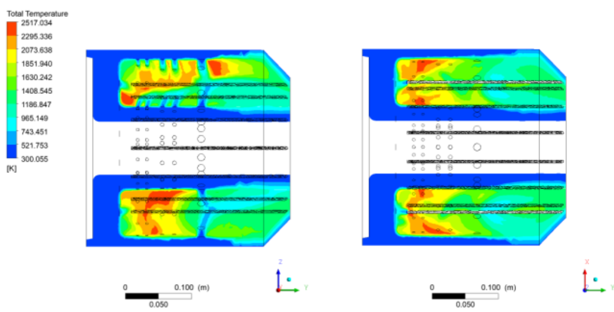

Figure 29 Total Temperature Contour Off-Design Case-2

Figure 29 illustrates the total temperature contour throughout the combustion chamber. The high-temperature value means burning regions. This off-design case is similar to off-design case 1. There is still some high-temperature gas after the dilution zones. This happens due to insufficient jet velocity that comes in the dilution zone because of smaller burner inlet pressure (in the other word, lower the suction region). As a consequence, the outlet temperature rises.

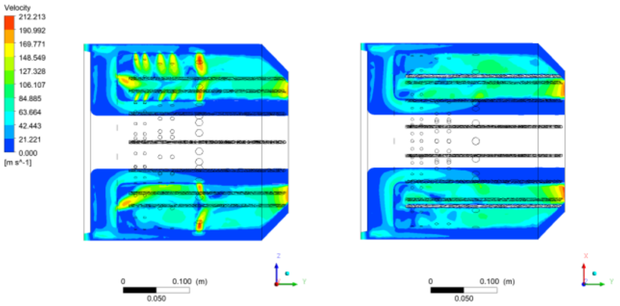

Figure 30 Velocity Contour Off-Design Case-2

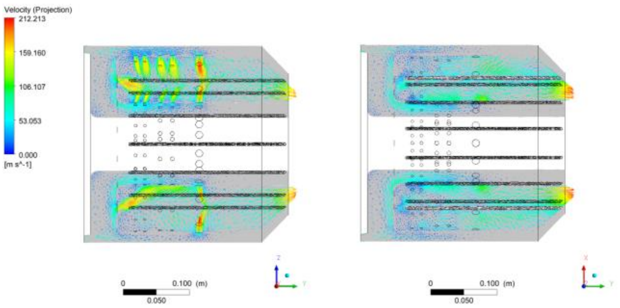

Figure 31 Velocity Vector Off-Design Case-2

The velocity contour and vectors in figure 30 and figure 31 are similar to the on-design velocity contour and vectors. However, the average outlet velocity of off-design case-2 is bigger than the on-design. This is happening because of the gas still has energy from its temperature. This indicates a higher expected thrust. In contrast, based on PCA, the lower the compressor compression ratio, the lower the thrust that is generated. These results lead to confusion and need to analyse further.

Further, similar to on-design and off-design case-1, the outlet total pressure and temperature are distributed evenly. This is a good sign for the next component analysis.

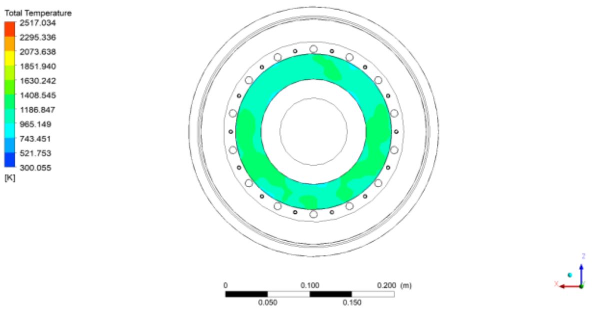

Figure 32 Outlet Total Temperature Contour Off-Design Case-2

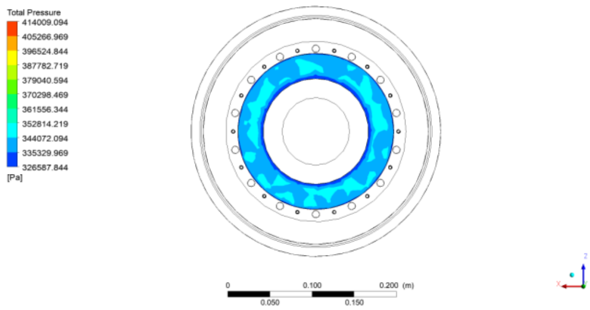

Figure 33 Outlet Total Pressure Contour Off-Design Case-2

5. Conclusion

Based on the obtained results, it can be concluded that:

1. The **preliminary design calculations** have been achieved based on empirical correlations of data derived from previous design experiences which is suitable for the preliminary phase.
2. The **on-design CFD analysis** has an acceptable error value which is lower than 10%. However, the airflow distribution in the primary zone, especially between the annulus and axial flow is higher than 10%.
3. The **off-design CFD analysis case: airflow drop 10%** resulted in the increasing value of outlet temperature and burning efficiency with a decrement of thrust due to lower outlet velocity.
4. The **off-design CFD analysis case: lower inlet total pressure** resulted in a higher burning efficiency and outlet total temperature. However, the outlet velocity increases too while what we expected is a decrease in outlet velocity.
5. This **CFD simulation is not perfect**. There are many sources of error such as coarse mesh, RMSE is still high, limitation to control combustion process, and not performing grid convergence test. Hence, better CFD simulation is needed to be re-performed

References

[1] Arthur H. Lefebvre, B. R. (1983). Gas Turbine Combustion Chamber. McGraw-Hill Book Company.

[2] Rashid S. E. Mohammed, (2019). Design and Analysis of Annular Combustion Chamber for a Micro Turbojet Engine. International Journal of Aerospace and Mechanical Engineering.

[3] C. Priyant Mark, A. Selwyn, (2016). Design and Analysis of Annular Combustion Chamber of a Low Bypass Turbofan Engine in a Jet Trainer Aircraft.

[4] Firman Hartono (2020). Aerodynamic Design of Combustion Chamber. ITB