Journal of Unmanned System Technology

Combustor Design for Turbocharger Turbojet Engine

N. Fuad†, F. Hartono†, M. A. Moelyadi†

†Faculty of Mechanical and Aerospace Engineering, Institute of Technology Bandung,

Bandung, Indonesia, 40132.

***Abstract*—**This research explains several variation designs of the combustion chamber for a turbojet engine based of T70 turbocharger. The T70 turbocharger engine is widely used in vehicles, especially cars. A problem that often occurs in the combustion chamber is non-uniform flow which makes pressure loss. The other problem is no combustion occurs in the primary zone, but it occurs in the dilution zone until the outlet engine nozzle. The method performed in this research is hot flow simulation in combustion chamber using ANSYS software. The aim of this research is to make optimum combustion chamber design with minimum pressure loss, small pattern factor, and turbine inlet temperature (TIT) that does not exceed limit of the material. Flow analysis on each variation designs is performed to find the best flow patterns. A good flow pattern follows some certain conditions such as the presence of air circulation and sufficient air mass flow in each zone. The analytical parameters that used to consider the initial design target are turbine inlet temperature, pattern factor, mass flow distribution, and pressure loss. In this combustion chamber design, the temperature target should less than 1260 deg. Kelvin, the pattern factor should less than 0.2, the pressure loss is less than 8 percent, the mass flow passing through the snout, primary zone, secondary zone, dilution zone, cooling are 8, 15, 30, 45 and 5 percent, respectively. The simulation results show that the turbine temperature is 1245.1 deg. K, the pattern factor of 0.44, the pressure loss of 19.82 percent, and the mass flow passing through the snout, primary zone, secondary zone, dilution zone, and cooling are 2.82, 15.29, 27.6, 45.8 and 14.61 percent, respectively.

***Keywords*—Combustion chamber, pattern factor, hot flow, CFD**

I. INTRODUCTION

THE development of turbojet engine in Institute of Technology Bandung has been started since 2005, beginning with the manufacture of turbojet engine with small thrust about 24 N. The research has successfully built micro turbojet engines. Then, the research continued into a machine with thrust of 500 N. The development of the 500 N turbojet engine is not running perfectly due to the difficulty in manufacturing for many components, so the research diverted to a turbocharger turbojet engine. This work is a part of the space-based fuel for turbocharger turbojet engine research. In short, this work contains combustion chamber design process. It begins with analytical calculation and continued with the CFD simulation for flow analysis and validation. It is studied the correlation between the analytical calculation and CFD simulation of the combustion chamber design.

A combustion chamber is a meant to further increase compressed air temperature from the compressor. In the combustion chamber there are three zones namely primary, secondary, and dilution zones. The primary zone is a zone where fuel with air mixed causing combustion. Secondary zone is a zone for further combustion process which comes from the remnants of unburned fuel in the primary zone. The last zone is the dilution zone. Dilution zone is a zone for cooling process so that the outlet temperature of combustion chamber is not too high. It needs an optimal sizing for primary, secondary, and dilution zones so that the air flow passing through combustion chamber does not damage the combustion chamber and has a small pressure loss.

An effective and efficient combustion requires higher turbine inlet temperature with less pressure loss. Effective combustion is a combustion at a given conditions fulfilling to the design requirement. Efficient combustion is a perfect combustion which all required fuel burned out. One per cent increase in pressure loss can lead to a half percent reduction in thrust force or a quarter percent increase in specific fuel consumption. Optimum flow mixing in the combustion chamber can be yielded by creating turbulent flow. The production of turbulent flow in the combustion chamber can be generated using a reverse or swirling flow. The swirling method requires a relative longer combustion chamber than the reverse flow burner, but effective enough to produce turbulent flow and has a smaller pressure loss than the reverse flow. The selection of both models depends on the needs and objectives of each.

There is little difference in design process between the combustion chamber and other components in the machine. The design process of compressor and turbine already have clear procedures. However, the combustion chamber does not have standard design method. This due to complex flow phenomena occurred inside the combustion chamber including turbulent flow, chemical reaction combined with aerodynamics. Therefore, to design combustion chamber, trial and error is often used.

Increasing the growth of computers in both speed and memory space, the trial and error process can be simulated using Computational Fluid Dynamics (CFD) method. The cost spent for simulation using CFD for combustion chamber is much lower of than that of the experiment. However, there remains a shortfall using CFD method compared to the experiment. Modelling process with CFD has error due to uncertainties in turbulence model that needs to be corrected by experiment. Therefore, CFD analysis still requires experiment, but it needs a CFD approach in the beginning process of the combustion design.

II. BACKGROUND AND THEORY

A. *Turbocharger Turbojet Machine*

In this paper the design of combustion chamber for a turbocharger turbojet engine is accomplished. The turbocharger itself was made of only containing of turbine and compressor without combustion chamber. Therefore, to modify turbocharger into a turbojet, combustion chamber design and manufacture are necessary.

In turbochargers air is passed into compressor to get higher pressure at the outlet. It is then compressed and burned by a piston engine to obtain higher temperature. This hot air is required by a turbine in expansion process to provide energy used to rotate the compressor periodically. Unlike turbocharged engine, turbojet engine has a combustion chamber. The process is similar to the turbocharger in piston engine. However, turbocharger turbojet engine has more continuous combustion than turbocharger piston engine.

Figure 1 Turbocharger Turbojet.

B. *Combustion Chamber Components*

There are three major components in the combustion chamber namely inlet diffuser, dome / snout, and liner. Additional components includes fuel injectors, burners casing, igniter, and swirler. The layout of these components can be seen in Figures 1 and 2.

- Casing is a component of combustion chamber separating system in the combustion chamber to its outside environment. The casing acts like a pressure vessel which must be able to withstand large pressure difference between the inside of combustion chamber and the outside environment. The pressure inside the combustion chamber is greater than pressure at the outside environment. In the casing, although it is subjected high temperatures, there is a cooling process by air entering into the combustion in the dilution zone.
- Diffuser reduces air velocity entering the combustion chamber from the compressor. Another function of the diffuser is to restore the flow dynamic pressure as good as possible and to regulate air flow characteristics to be more stable and smooth when enter the liners. When designing a diffuser, pressure loss occurred in the diffuser are expected as small as possible.
- Snout is a complementary part of the dome / swirler that serves as a separator for secondary flow, intermediate airflow, cooling and dilution.
- Dome/ swirler is a part of the combustion chamber that serves as the entrance of primary air into the combustion zone (combustion zone). Dome/swirler is designed in such way so there is mixing flow occurred when primary air enters. Air turbulence is very important to enable the mixing of air with fuel, but note that the more turbulent of the mixing flow, the greater pressure loss will occur.
- Igniter is often used generally found in a form of electric spark igniter. Igniter must be placed in combustion zone where air and fuel are mixed. Igniter position should not be in the flame zone that can probably damage itself.
- Fuel injector injects fuel into the combustion zone. There are four types of fuel injector which are pressure-automizing, water blast, vaporizing, and premix / prevaporizing injector.

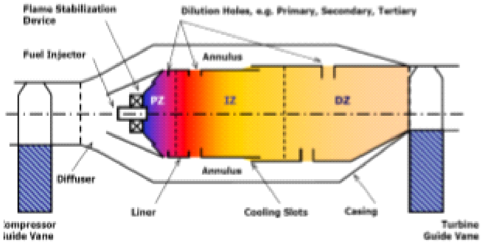

Figure 2 Components of combustion chamber.

- Liner is a part of the combustion chamber where combustion process is occurred. Liner is designed to operate at high temperatures so it requires special material. In addition, the liner remains to be cooled by air flow. In some case, a special coating liner is used around the liner wall. This coating layer is able to lower temperature from the 1800 K to 1000 K. Transpiration cooling is a new technology in coating that creates porous / void in its liner material. The flow of cold air is passed through tiny channels and voids in the material be able more cooling the liner. In this method, the air mass needed is less than in the film cooling method.

C. *Combustion Chamber Components*

Tubular combustion chamber consists of many combustion chambers (cans) which each stands on its own. Each can has a fuel injector, liner, and casing. Incoming air from the compressor is directed to go to each can evenly. This type of combustion chamber is a most widely used in the early generation of gas turbine design. Advantages and disadvantages of the can type are as follows:

Advantages:

However,

- Simple machine testing because it requires only a small fraction of mass flow machine,
- Sturdy construction,
- Easy to maintenance,
- Easy to be designed.

Disadvantages:

- High pressure loss,
- Requires connecting pipes system among the cans,
- It consists a lot of cans so the machine mass become huge.

III. METHOD

A. Visibility Analysis in 3D Urban Environments

In short, steps to make a combustion chamber is shown in Figure 3. From the flow chart can be seen that the first step is to determine the specifications of the combustion chamber such as speed, thrust, fuel flow, etc. The second step is to determine the type of combustion chamber. The combustion chamber type include annular, tubular and can-annular types. Combustion chamber type will determine the reference area. The third step is determination of fuel injector. Fuel injector will affect the shape of the combustion chamber and the combustion process occurred in the combustion chamber. After that, the sizing of the casing and other necessary parameters for determining the dimensions of the combustion chamber are carried out. After getting the casing size and the parameters of the combustion chamber, calculation of overall combustion chamber dimension including extensive liner, liner length, the length of each zone at the liner, annulus size, snout size and casing length is done. The final step is to determine the size and number of holes for each zone including primary, secondary, and dilution zones. In the making of the combustion chamber, diffuser and swirler is not considered in the design Cooling design is only estimated based a method provided in lavebre book [1].

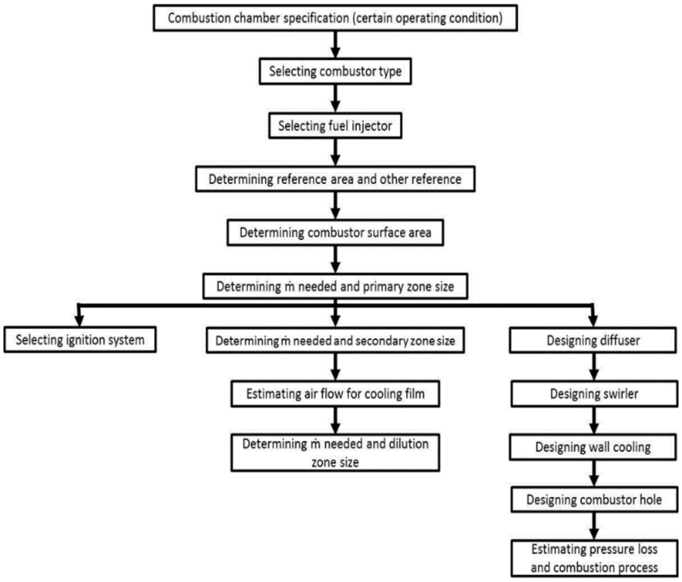

Figure 3 Combustion chamber dimension determination chart.

B. Calculating the Temperature and Input Pressure in Combustion Chamber

The The calculation Input such as combustion chamber temperature and pressure entering the combustion chamber taken from the T70 performance table as given in Table 1.

C. Dimension of Combustion chamber

The dimensions of combustion chamber is given in Table 2 and the three dimension drawing is shown at Figure 4.

Corresponding author: M. A. Moelyadi (email: moelyadi@ae.itb.ac.id, moelyadi.mochammad@gmail.com)

Parameters		**Value**	**Unit**
Air density	ρ	1.225	kg/m^3
Ambient condition temperature	T_0	288.15	K
Ambient condition pressure	P_0	1.013	Bar
Compressor Pressure Ratio	P_{t3}/P_{t2}	4.228	
Turbine Inlet Temperature	Tt_3	1260.4	K
Inlet efficiency	η_i	1	
Compressor efficiency	η_c	0.6	
Compressor coefficient	C_{pa}	1.005	
Spesific Heat Transfer	Υ_c	1.4	
Rotational velocity	RPM	105800	
Air Flow	Air_Flow	0.4931	kg/s
Fuel Air Ratio	FAR	0.0253	
Fuel Flow (40 MJ)	ff	0.0125	kg/s
Thrust	Thrust	208.367	N
Fuel Flow (Propana)	Ff	0.01086	kg/s

Table 1 Combustion Chamber Calculation Input From T70 Turbocharger Performance Table.

Combustion Chamber Dimension		
Dcasing	86.54	mm
Dliner	74.94	mm
Lliner	181.24	mm
dhpz	3.23	mm
n	35	
dhsz	6.68	mm
n	20	
dhdz	10.50	mm
n	12	

Table 2 Combustion Chamber Dimension Calculation Results.

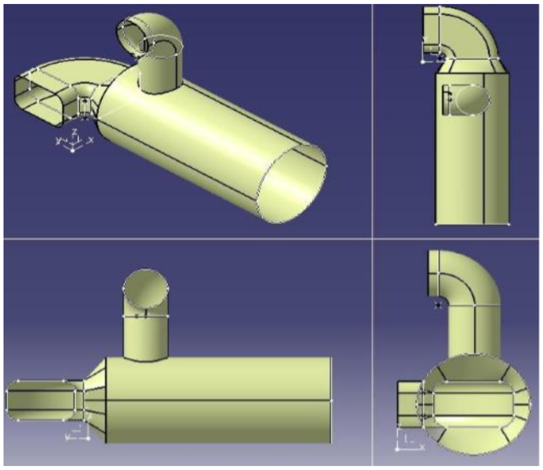

Figure 4 Three dimension drawing combustion chamber.

IV. RESULT AND ANALYSIS

A. *Result Analysis from CFD Simulation*

Figures 5 to 9 show the simulation results of the combustion chamber including total temperature distribution, total pressure distribution, propane fuel distribution, density and streamline velocity, respectively. The temperature in the combustion chamber reaches 2564 K. The combustion occurs in the primary zone as expected. However, there are some unburned fuel due to mass flow passing through the secondary zones not yet reached the target, which is only 20 percent. Therefore, the size of the hole in secondary zone should be enlarged to achieve the desired mass flow, 30 percent.

B. *Result Analysis from CFD Simulation*

Parameters of successful combustion chamber design are:

- Distribution of mass flow passing through the primary zone is 20%, secondary zone is 30%, and dilution is 50% of the total mass flow.
- Pattern Factor is less than 0.2.
- TIT is less than 1260 K.
- Pressure loss throughout the combustion chamber is less than 8%.

Combustion chamber simulation results is as shown in Table 3.

Parameter	Result	Target	Unit	Fulfilled / Not
TIT	1245.11	1260	K	✓
PF	0.44	0.2		X
Pressure loss	19.82	8		X
Mass Flow				
Snout	2.82	15 - 20	%	X
Primary	15.29		%	✓
Secondary	27.6	30	%	X
Dilution	45.8	47	%	✓
Cooling	14.61	5	%	X

Table 3 CFD Simulation Results.

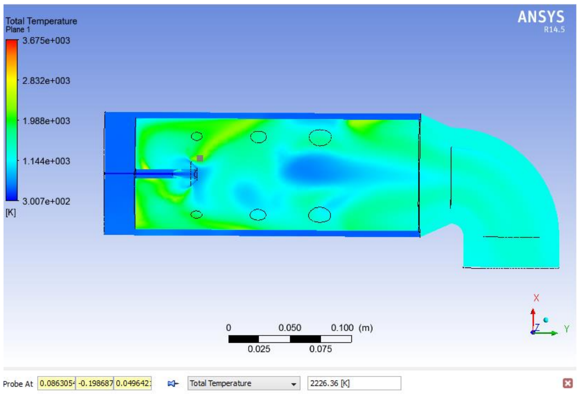

Figure 5 Total temperature distribution.

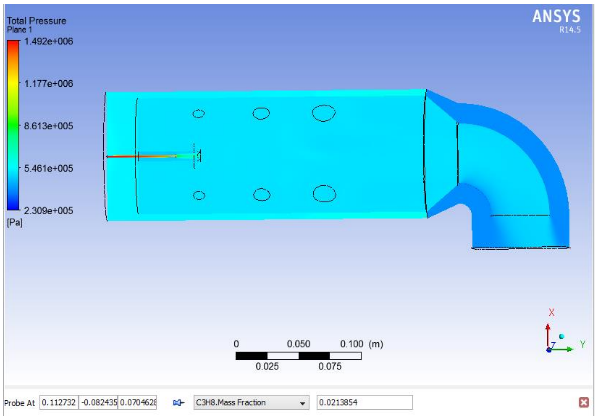

Figure 6 Total pressure distribution.

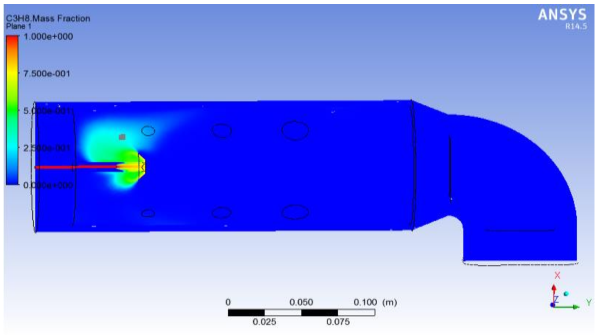

Figure 7 Propane fuel distribution.

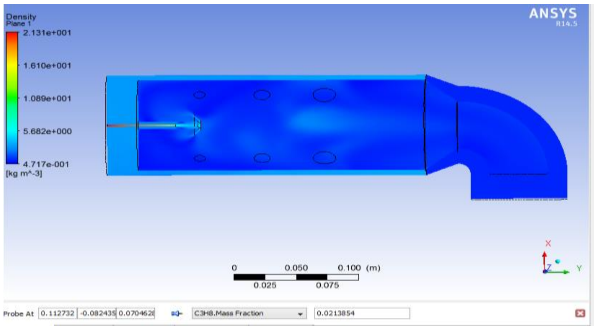

Figure 8 Density distribution.

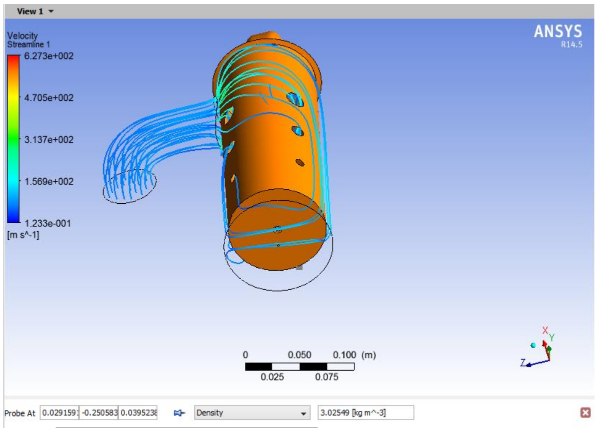

Figure 9 Streamline velocity.

V. CONCLUSION

From the simulation results, it can be concluded that:

- Propane fuel has been evenly distributed according to the original purpose, the propane fuel that passes through the upper and lower side of bowl have the same amount.
- The parameters of successful mass flow distribution in combustion chamber design have been fulfilled in primary and dilution zones, while they have not been fulfilled in snout, secondary, and cooling zones.
- Pattern factor has not met the parameter since it is difficult to design a pattern factor in a short and curvy machine model.
- Turbine inlet temperature is adequate, not exceeding the parameter.
- Pressure loss in the combustion chamber is exceeding the parameter.

REFERENCES

[1] Lavebre, Arthur H., Gas Turbine Combustion, Hemisphere publishing corp., McGraw-Hill Book Company, New York, 1976.

[2] Rahadianto, Saka. Analisis, Modifikasi, dan Simulasi Ruang Bakar Mesin Turbojet 500 N. Tugas Akhir, ITB, Indonesia, 2010.

[3] Maris, Muhammad. Desain Ruang Bakar Reverse Flow Untuk Mesin Mini Turbojet. Tesis, ITB, Indonesia, 2013.

[4] en.wikipedia.org/wiki/combustion

[5] en.wikipedia.org/wiki/Turbocharger_engine

[6] www.cfd-online.com/wiki/Turbulence_modeling.