CFD 기법에 의해 예측된 흡입구 및 배기구 손실을 고려한 터보축 엔진의 장착성능에 관한연구

공창덕* · George Omollo Owino**

Installed Performance Analysis of a Turboshaft Engine Considering Inlet and Exhaust Losses Estimated by Cfd Technique

Changduk Kong* · George Omollo Owino**

ABSTRACT

The purpose of this study is to analyze the installed performance of the PW206C turbo shaft engine used in the development of the smart UAV(Unmanned Ariel Vehicle) by KARI(Korean Aerospace Research Institute). It mainly aims to investigate performance behavior at installed conditions using both inlet and exhaust losses generated by CFD analysis of the ducts.

The ways employed to be able to analyze the performance extensively were mainly carried out by performing design point analysis of the engine where the performance simulation results from the commercial program 'GASTURB 9' used for simulation were used as inlet boundary condition for the ducts in CFD program

The use of CFD tool involve modeling of the ducts to conform with the stipulated shape and sizes as defined by KARI with a grid density that allows reasonable flow characteristics applicable to aircraft components.

Respective values of Shaft horse power obtained by varying flight Mach number, Gas generator RPM and Altitude considering several losses inclusive of those estimated by use of CFD tool were then plotted at three conditions with the ECS-OFF, ECS-MAX and at un-installed condition

Reasonable results were obtained as a result of using computational fluid dynamics that can hence be justified as an alternative tool for use in future flow analysis of engine and components.

Key Words: Installed performance, Inlet and Exhaust duct losses, Turbo Shaft Engine, CFD simulation.

1. Introduction

Since the model of study involves advanced flight dynamics it is essential that the intake and exhaust system including the plenum chamber be designed with maximum accuracy to facilitate good performance characteristics, hence the purpose of this study being the analysis of inlet and exhaust losses experienced

---

* 조선대학교 항공우주공학과
** 조선대학교 항공우주공학과 대학원
연락저자, E-mail: cdgong@chosun.ac.kr

in the ducts in question. Computational fluid dynamics program CFD is employed to analyze the duct total pressure loss that are later used together with other losses to analyse installed performance of PW 206C engine.

2. Engine Description

It is important that we understand the engine of study as its performance parameters are used as inlet boundary conditions for the ducts

The Figure 1 shows the schematic layout of this engine used, and Table1shows two working parameters at maximum take-off sea level static condition and the 10000ft static condition data from the manufacturer to the public domain. Table 2 shows operating range of a PW 206C engine as used on the smart UAV. [1][2]

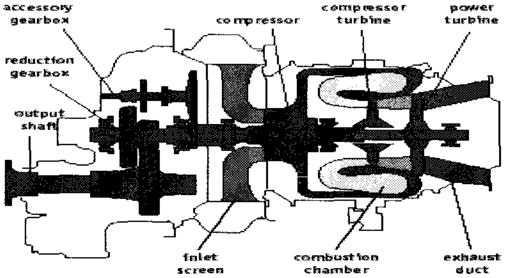


Fig. 1 Engine schematic diagram of PW206C turbo shaft engine

Table 1 performance parameters at two reference points as given in the EEPP program [3]

Atmospheric condition	static sea level standard	10000ft Mach No 0 static
Mass flow rate(lbm/s)	4.418	3.307
Fuel flow rate (lbm/s	0.087	0.0724
Compressor pressure ratio	7.912	9.82
Turbine inlet temp	2258	2388
Shaft horse power(hp)	560.8	496.0
SFC (lbm/hp hr)	0.556	0.5260
Gas gen spool speed	58900	56221

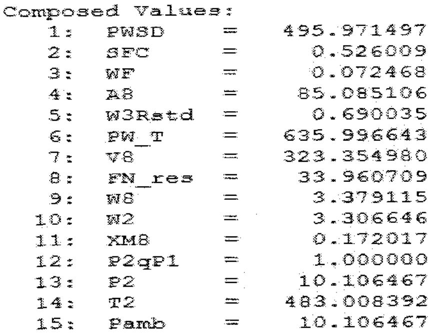
Table 2 Operating range of propulsion system

Gas Generator RPM	65% ~ 100%
Altitude (ft)	0 ~ 15000
Flight Mach No.	0 ~ 0.4

3. CFD simulation of inlet and exhaust ducts

The engine performance data like values of T_1, T_2 , P_1, P_2 Mass flow rate , and far-field ambient conditions at the given operating altitude are used in CFD program as inlet boundary condition for the ducts

It is therefore mandatory that design point simulation must be done to obtain this preliminary inlet boundary conditions.[4]



```
Composed Values:
  1:  PWSD      =   495.971497
  2:  SFC       =     0.526009
  3:  WF        =     0.072468
  4:  A8        =    85.085106
  5:  W3Rstd    =     0.690035
  6:  PW_T      =   635.996643
  7:  V8        =   323.354980
  8:  FN_res    =    33.960709
  9:  W8        =     3.379115
 10:  W2        =     3.306646
 11:  XM8       =     0.172017
 12:  P2qP1     =     1.000000
 13:  P2        =    10.106467
 14:  T2        =   483.008392
 15:  Pamb      =    10.106467
```

Fig. 2 Design point simulation at 10000ft Mach 0

Having obtained all inlet boundary conditions inlet and exhaust duct modeling begins in CFD. Structured volume grids of 61034 and 20000 grid elements for intake and exhaust ducts were used respectively.

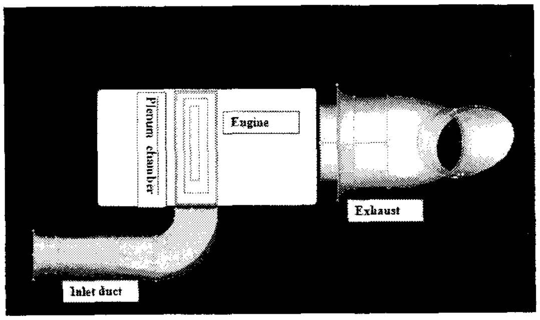


Fig. 3 CFD schematic arrangement of engine, and duct

Inlet and outlets are defined and respective inlet boundary condition same values as design point simulation are entered ready for CFD analysis .

Table 3 boundary conditions from design point data

Intake	Temp k	Pressurepa	Density kg/m^3	Sonic vel.m/s	Viscositym^2/s
Ambient	268.67	10.10	0.909	328.58	1.86*10^-5
Station 2	483.00	10.10	0.909	-	-

Table 4 Exhaust boundary conditions

Exhaust	Temperature k	Pressure Pa
inlet	887	183.800
outlet	626.8	-

CFD output is then analysed for reasonable results if not then the nature of grid smoothness is corrected using wall smoothing factor Reynolds Navier strokes (RANS) together with wall functions standard $k-\epsilon$ model and Kato launder $k-\epsilon$ model. 800 iteration were performed that resulted to throat velocity of 141m/s and exit 26.39m/s indicating that engine to component mass flow rate has been matched.

$$\dot{m} = \rho UA \tag{1}$$

$$\dot{m} = 0.909 * 26.39 * 0.148 = 3.307 \tag{2}$$

Table 5 CFD simulation results of intake duct based on design point data at 10000ft

Intake Duct	inlet	outlet
Density Kg/m^3	0.957	0.9916
Mach No	0.435	0.316
Pressure (Pa)	70119.68	68506.263
Temperature (K)	268.0	270.28
Velocity U (m/s)	141	26.39

Table 6 CFD simulation results of the exhaust duct

Exhaust Duct	Inlet	Outlet
Temperature (K)	868.86	626.8
Mass flow rate kg/s	3.555	Both exit 3.981
Pressure (Pa)	71271.1	70202.33

Percentage total pressure loss through the intake duct is then calculated using as

$$\frac{P_{01} - P_a}{P_{01}} * 100 \tag{3}$$

$$\frac{70119.68 - 68506.263}{70119.68} * 100 = 2.300\% \tag{4}$$

Exhaust duct using equation 3 above
Loss= 1.499%

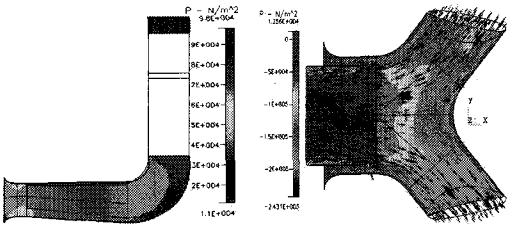


Fig. 4 pressure distribution in the intake and exhaust duct

4. Installed Performance

Table 6 indicates two installed conditions ECS-off and ECS-Max, and the losses considered.

Fig. 5, 6 and 7 shows comparison results of shaft power between ECS-off and ECS-max installed loss conditions a According to comparison, the shaft power with ECS-Max is less than the shaft power with ECS-off due to power extraction by ECS, inlet temp rise, bleed air loss and mechanical losses.[5]

Table 7 Total loss parameters considered

Loss parameter	ECS OFF	ECS MAX	UNINSTALLED CONDITION
inlet efficiency	1	0.9922	1
inlet temp rise	5R	5R	0
Bleed air loss	0	5%	0
Power extraction	5HR	7HP	0
Exhaust loss	1.499%	1.499%	0
Intake loss	2.3%	2.3%	0

Table 8 variations considered

Gas gen RPM %	60	70	80	90	100
Altitude ft	0	5000	10000	15000	20000
Mach no	0	0.1	0.2	0.3	0.4

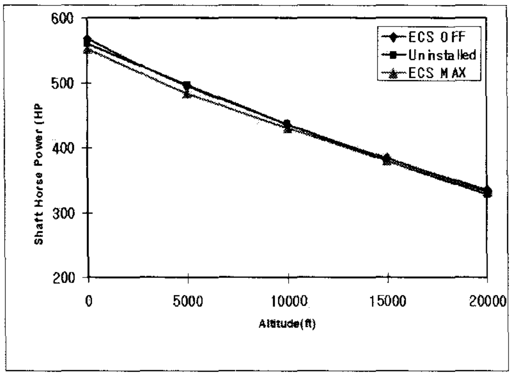


Fig. 5 variations of altitude

Variation of altitude was done at sea level static condition with Mach number 0.4 Gas generator RPM 100%

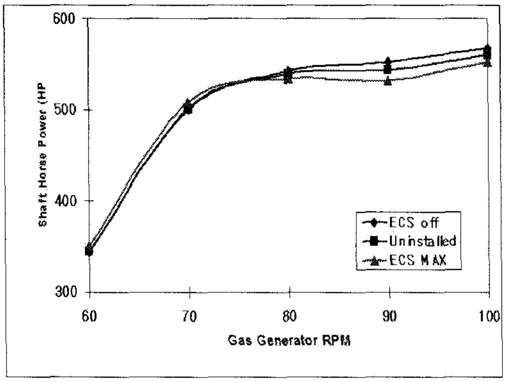


Fig. 6 variations of Gas generator speed

Analysis was performed at Sea level static mach number 0.4

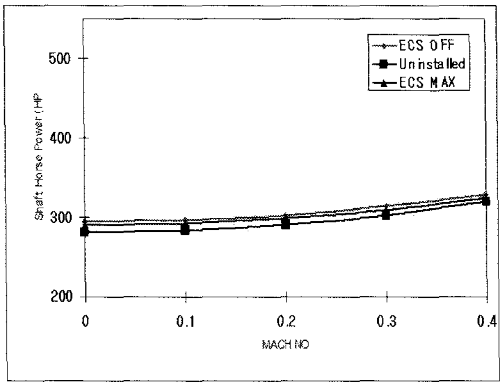


Fig. 7 variations of Mach number

Evaluation done at 10000ft with Gas generator RPM at 60%

5. Conclusion

Through this study, the design and off design performance simulation of the PW206C turbo shaft engine, used in the development of the smart UAV (Unmanned Ariel Vehicle) byKARI(Korean Aerospace Research Institute) were performed and analysis results used for CFD simulation.

According to analysis results, it was found that the duct loss estimation by CFD were reasonable with an added advantage of being able to view both flow and pressure distribution within the ducts.

Secondly, shaft power with ECS-max was mostly less than the shaft power with ECS-off due to power extraction by ECS, inlet temp rise, bleed air loss and due to mechanical inefficiencies. However in case of high altitude, the ECS power extraction is not influenced greatly

REFERENCE

1. Kong, C. and Ki, J.,2003,"A New Scaling Method for Component Maps of Gas Turbine Using System Identification",J.of Engineering for Gas Turbines and Power, Vol. 125, Number.
2. C.D. Kong et al., 2006, Component Map Generation of a Gas Turbine Using Genetic Algorithms, J. of Engineering for Gas Turbines and Power, vol.128 no.128.
3. P&WC,"EEPP(Estimated Engine Performance Program)
4. GASTURB 9,2001 ,"Operation Manual"
5. George"analysis of installed and un-installed performance of turbo shaft engine"(24Th KSPE)