Transient Performance Simulation of Propulsion System for CRW Type UAV Using SIMULINK®

Changduk Kong*, Jongha Park*
***Dept. of Aerospace Engineering, Chosun University**

Keywords: *Performance Simulation, Propulsion System, CRW Type UAV*

Abstract

A Propulsion System of the CRW(Canard Rotor Wing) type UAV(Unmanned Aerial Vehicle) was composed of the turbojet engine to generate the propulsive exhaust gas, and the duct system including straight and curved ducts, tip-jet nozzles, a master valve and a variable main nozzle for three flight modes such as lift/landing mode, low speed transition flight mode and high speed forward flight mode. In this study, in order to operate safely the propulsion system, the dynamic propulsion performance behavior of the system was modeled and simulated using the SIMULINK®, which is the user-friendly GUI type dynamic analysis tool provided by MATLAB®. In the transient performance model, the inter-component volume method was used. The performance analysis using the proposed models was performed at various flight conditions, valve angle positions and fuel flow schedules, and analysis results could set the safe flight mode transition region to satisfy the inlet temperature overshoot limitation as well as the compressor surge margin. Performance analysis results using the SIMULINK® were compared with them using the commercial program GSP.

1 INTRODUCTION

Recently, the KARI (Korea Aerospace Research Institute) has been developing the smart UAV with new concept since 2002. This project is to develop the civil-use multi-purpose UAV, which has some new technologies such for CRW concept with VTOL(Vertical Take-Off/Landing) and high speed forward flight, high reliability, small, light and intelligent flight capability.

The CRW concept, which uses a hot cycle, is suitable for the smart UAV to fulfill requirements of the VTOL UAV mission. This concept can operate the tip-jet nozzle driven by hot gas of the main jet engine to rotate the rotary wing in vertical flight mode and use the turbojet engine for high speed forward flight[1]. The propulsion system for this concept flight vehicle can be operated by dividing into three flight modes such as the hovering mode for take-off/landing, the high speed forward flight mode with fixed wing and the transition flight mode between the above two flight modes. In order to establish safe operation region of the propulsion system, performance simulation should be needed.

The performance simulation can provide important data not only to confirm performance characteristics in much wider flight envelope, which experimental tests are not able to carry out, but also to design the engine controller or the integrated flight control system.

In this study, the propulsion system for the smart UAV was modeled to predict dynamic performance behaviors using the SIMULINK®. The SIMULINK® is a software package for modeling, simulating and analyzing the dynamic system. The SIMULINK® provides a GUI platform for building models as block diagrams for modeling, and this analysis tool includes linear and trimming, which can be accessed from the MATLAB application toolboxes.

There are some research examples using the SIMULINK®. For instance, a dynamic non-linear model for a single shaft industrial gas turbine was developed by Bettocchi et al(1996)[2]. This model consisted of modular structure representing engine components, and it was carried out in simplification configuration. Crosa et al (1998)[3] described a 65 MW heavy-duty gas turbine plant model using the SIMULINK®. In this model, a compressor with variable inlet guide vanes was represented in serial arrangement, separated by dynamic blocks with tabular characteristic data set using 2D-look-up table blocks. Kim (1999)[4] developed an accurate and reliable dynamic engine model using the SIMULINK® , and he used the dynamic model to simulate normal operating conditions from idle to maximum and severe conditions such as the fuel cut-off condition in flight and the starting condition in wind milling flight. Moreover, Kong (2003)[5] proposed a performance simulation model of the PT6A-62 turboprop engine using the SIMULINK® to predict transient and steady state behaviors.

In this study, in order to operate safely the propulsion system, dynamic behavior of the system was modeled and simulated using the SIMULINK®. Performance analyses using the developed models were performed with several kinds of fuel flow schedules, and their results could establish safe flight mode transition regions to keep inlet temperature overshoot limitation. In order to verify the proposed performance program using the commercial performance program GSP.

2 PROPULSION SYSTEM MODEL

The studying propulsion system is mainly composed of an engine and duct systems. A turbojet engine is used as a major power plant, and the duct system is divided into the straight duct for the main nozzle, the controllable main nozzle, the curved duct with tip-jet nozzles and the master valve for controlling tip-jet nozzle thrust. The hot pressure jet cycle was selected to rotate rotary wings with tip-jet nozzles driven by hot gas from the main turbojet engine [1].

In the take-off/landing mode, the vehicle can hover using the rotary wing with tip-jet nozzles driven by hot gas ejected through the curved and straight ducts from the engine, but the main nozzle should be closed to prevent the forward flight. However, if the flight velocity reaches to the proper forward velocity more than the stall speed, then the vehicle can fly using the only main jet nozzle thrust but the master value should be closed. The main jet nozzle uses the variable convergent type, and the tip-jet nozzle uses the fixed convergent type. The Fig. 1 shows a schematic layout of the Smart UAV propulsion system.

Design point performance was set at sea level standard atmospheric condition and 100% engine rotational speed. Therefore the design point reference performance data of engine major components excluding ducts and the master value were determined.

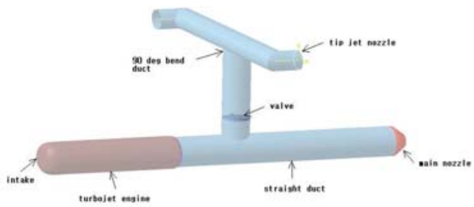
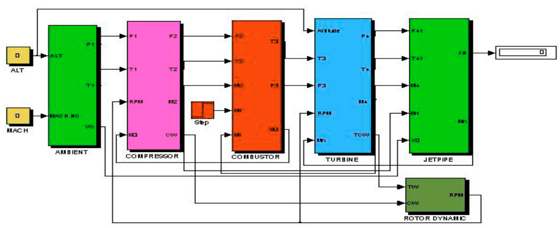


Fig.1. Simplified layout of smart UAV propulsion system

3 SIMULINK MODEL

The overall model was composed of modular blocks representing individual components such as intake subsystem, compressor subsystem, combustor subsystem, compressor turbine subsystem, inline duct subsystem, curved duct subsystem, main nozzle subsystem, tip nozzle subsystem, and rotor dynamics subsystem. The overall SIMULINK® model is shown in Fig. 2. Component subsystems are shown as Fig. 3, 4 and 5, respectively.



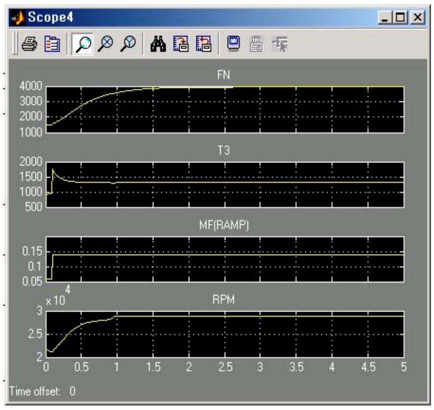


Fig.2. SIMULINK overall model and display window for calculation results

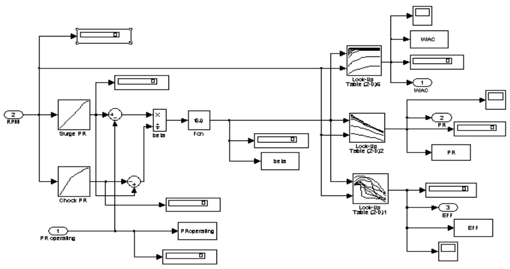


Fig. 3. Compressor map search subsystem

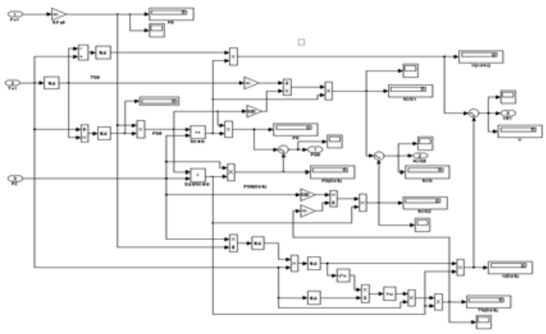


Fig.4. Subsystem for searching the choked main nozzle

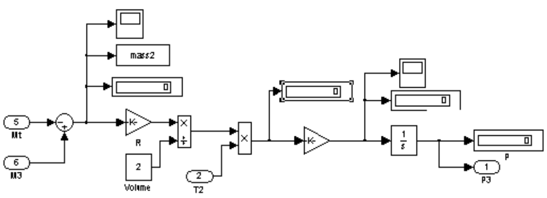


Fig. 5 Inter-Component Volume subsystem

3.1 Inter-Component Volume (ICV) Method

In the transient performance model using the SIMULINK®, the ICV (Inter-Component Volume) model was used. In this method, both mismatches of work and mass flow rate between components are assumed during transient engine operation. The mass flow rate mismatch between components must be used to calculate the rate of change of pressure at each station of the study engine, by taking values of inter-component volumes and applying the perfect gas law. Also, the work mismatch is utilized to estimate the variation of rotational speed by integrating the difference work between compressor and turbine during transient operation [5].

If the initial steady state operation at the given fuel flow and nozzle area condition is defined by

$$X_0 = (P_2, P_4, N) \tag{1}$$

and the fuel flow, W_f, is changed, then the new values of turbine torque, compressor torque, compressor mass flow, W_c, turbine mass flow, W_t and nozzle mass flow, W_n, at a certain time may be found. The accumulation of mass flow rate in the combustion chamber, $\dot{m}_1$, can be given by

$$\dot{m}_1 = W_c + W_f - W_t \tag{2}$$

Similarly, the accumulation of mass flow rate in the main nozzle or tip-jet nozzles, $\dot{m}_2$, can be given by

$$\dot{m}_2 = W_t - W_n \tag{3}$$

If the perfect gas law is applied to the flow of the inter-component volume, V, and then,

$$P = \frac{R}{V} \cdot (m \cdot T) \tag{4}$$

The time derivative of pressure is,

$$\dot{P} = \frac{R}{V} \cdot \frac{d}{dt} \cdot (m \cdot T) = \frac{R}{V}(\dot{m} \cdot T + m \cdot \dot{T}) \tag{5}$$

Where the second term can be neglected because it is relatively too small compared to the first term as well as for simplicity. Thus, the time derivative of pressure in the combustion chamber can be expressed by the following equation,

$$\dot{P}_2 = \frac{R \cdot T_2}{V_1} \cdot (\dot{m}_1) \tag{6}$$

By the same manner, the time derivative of pressure in the duct system with the main nozzle or the tip-jet nozzles can be derived by the following equation,

$$\dot{P}_4 = \frac{R \cdot T_4}{V_2} \cdot (\dot{m}_2) \quad (7)$$

Therefore, if relationships derived as the above are used, $\dot{p}_2$ can be calculated from values of W_c, W_t, W_f and inter-component volumes of compressor and combustor. Similarly, $\dot{p}_4$ may be found from values of W_t, W_n, and inter-component volumes of turbine and duct including nozzle. The rotational speed increase in the rotor acceleration can be found from the difference torque between compressor and turbine, and the rotor inertia by the following equation,

$$\frac{dN}{dt} = \left(\frac{30}{\pi}\right)^2 \cdot \frac{J}{N \cdot I} \cdot [TW - CW] \quad (8)$$

where, I is the polar moment of inertia of the rotor spool, J is Joulean mechanical equivalent of heat, N is the shaft rotational speed, and TW and CW are works of turbine and compressor, respectively.

Thus, from the given value of the engine state vector at any time t,

$$X_0 = (P_2, P_4, N) \quad (9)$$

and the externally applied conditions of fuel flow and nozzle area, it is possible to calculate the values of $\dot{p}_2$, $\dot{p}_4$ and $\dot{N}$. The values of $\dot{p}_2$, $\dot{p}_4$ and $\dot{N}$ can be repeatedly used to predict the value of X at time $t + \Delta t$. Therefore, if the engine state vector at any time t may be given, its changed value at time $t + \Delta t$ may be predicted. Therefore the transient simulation can be performed by applying repeatedly at each time step until the target time[6]. In this study, the 4th order Runge-Kutta method is applied for the integrator[7].

3.2 Duct System Modeling

Because the proposed propulsion system has ducts between the gas generator and the tip-jet nozzle as well as the main nozzle, the duct loss should be estimated to simulate the precise propulsion system model depending on its length and shape. In order to predict the friction coefficients in the duct, the Reynolds number R_n is defined as the following equation,

$$R_n = \frac{V}{\sqrt{T}} \times \frac{D \cdot \rho}{\mu} \quad (10)$$

where μ is dynamic viscosity coefficient, ρ is density, V is gas velocity, and D means exit diameter of the duct.

Therefore internal duct friction coefficients have the following value at each R_n range.[4]

$$f = \frac{0.0791}{R_n^{0.25}} \qquad (3000 < R_n \leq 10{,}000) \quad (11)$$

$$f = \frac{0.0460}{R_n^{0.2}} \qquad (10{,}000 < R_n \leq 200{,}000)$$

$$f = 0.0014 + \frac{0.0125}{R_n^{0.32}} \qquad (200{,}000 < R_n \leq 3{,}000{,}000)$$

A calculation procedure for the loss of the straight duct is shown with the following step using the Fanno line theory.[4]

Firstly, the characteristic length, $\frac{fL_{\max}}{D}$, of the duct has the relationship with the flow mach number as follows,

$$\frac{fL_{\max}}{D} = \frac{1 - \gamma M^2}{\gamma M^2} + \frac{\gamma + 1}{2\gamma} \ln \frac{(\gamma + 1) M^2}{2\left(1 + \frac{\gamma - 1}{2} M^2\right)} \quad (12)$$

where, L_{max} is the maximum length at choking condition due to friction, and γ is the specific heat ratio.

The characteristic length at the duct exit can be calculated from the difference between the characteristic values at the duct entry and the given geometry.

$$\left(\frac{fL}{D}\right)_{M_{exit}} = \left(\frac{fL}{D}\right)_{M_{ent.}} - \left(\frac{fL}{D}\right)_{M_{geo}} \quad (13)$$

where subscripts M_{exit}, M_{entry}, and M_{geo} mean Mach number at exit, entry, and given geometry, respectively.

Therefore the flight Mach number at the duct exit can be calculated from the equation (12) and (13). The pressure as to flight Mach number can be expressed as the following equation,

$$\frac{P}{P_s^*} = \frac{1}{M} \left[\frac{\gamma - 1}{2 + (\gamma - 1) M^2} \right]^{1/2} \quad (14)$$

where P means pressure, the subscript * means choke condition (M = 1) and the subscript s means static.

Therefore, the static pressure at the duct exit is,

$$\left(P_s\right)_{exit} = \left(P_s\right)_{ent.}\left[\left(\frac{P_s}{P_s^*}\right)_{exit} \times \left(\frac{P_s^*}{P_s}\right)_{ent.}\right] \qquad (15)$$

And the total pressure at the duct exit is,

$$\left(P_t\right)_{exit} = \left(P_s\right)_{exit}\left[1+\frac{\gamma-1}{2}M_{exit}^2\right]^{\frac{\gamma}{\gamma-1}} \qquad (16)$$

In case of 90° bend duct, the loss can be calculated as follows, [4]

$$\Delta P_t = K_{90} \times q_{ent.} \qquad (17)$$

where $q_{ent.}$ is the dynamic pressure at the duct entry, and K_{90} is the correction coefficient of the 90° bend duct as follows,

$$\begin{aligned} K_{90} = \; & -0.0167373(r/D)^9 + 0.4402006(r/D)^8 \\ & -3.680925(r/D)^7 + 15.33455(r/D)^6 \\ & -36.91412(r/D)^5 + 54.3209(r/D)^4 \\ & -49.29369(r/D)^3 + 26.62241(r/D)^2 \\ & -7.464826(r/D) + 0.9646493 \end{aligned} \qquad (18)$$

where r means the bend radius of the curved duct. Subsystems for straight and curved ducts using the SIMULINK® are shown as the Fig. 6.

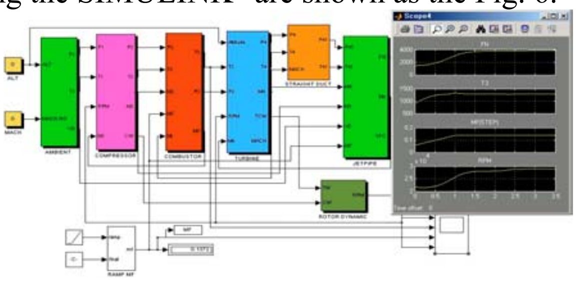


Fig.6. SIMMLINK modeling with straight duct and curved duct

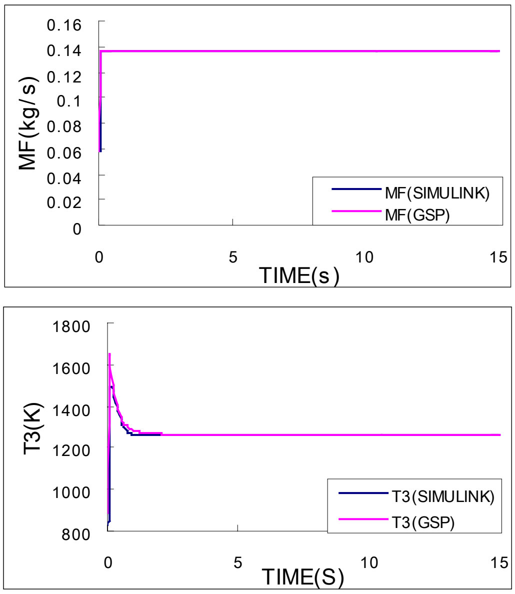
4 Transient Performance Analysis

Transient performance analysis was performed on 3 cases such as the main turbojet case for verification of the proposed simulation model, the propulsion system case including the curved duct with tip-jet nozzles for the take-off/landing flight mode and the propulsion system case including the straight duct with the main nozzle for the cruising flight mode.

In order to determine initial input values for analysis, steady-state analysis results by using the previously developed program were used. For evaluation of the validity, the analysis results by the SIMULINK® model were compared with them by the commercial program for evaluation of the validity.

4.1 Transient behaviors of main turbojet engine

Transient analysis results of the turbojet engine without duct systems are shown in the Fig. 7 and 8. Fuel flow schedules to accelerate from idle of 75% rpm to maximum rotational speed of 100% rpm at sea level static standard atmospheric condition were divided into the step increase case and the ramp increase case to avoid the overshoot of turbine inlet temperature. As shown in Fig.7, analysis results using the proposed SIMULINK® model are well agreed with them using the commercial program GSP. It means the developed SIMULINK® model is acceptable for transient performance analysis of the study turbojet engine. In comparison of the transient behavior results between the step fuel flow increase schedule and the ramp fuel flow increase schedule, it was found that the turbine inlet temperature overshoot was greatly severe in the step fuel flow increase operation.



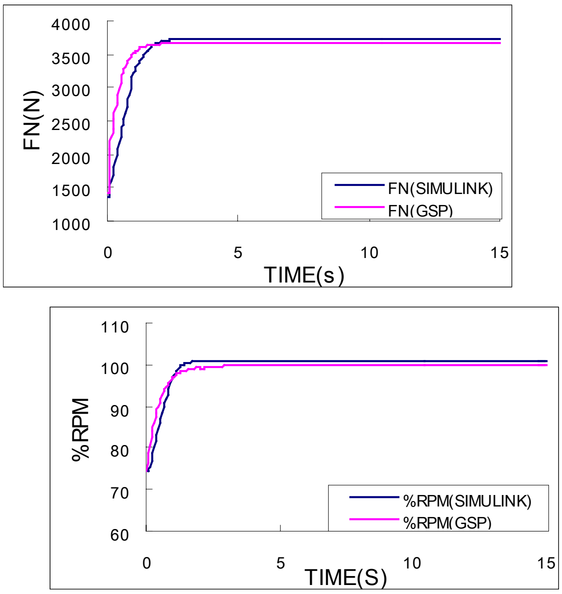
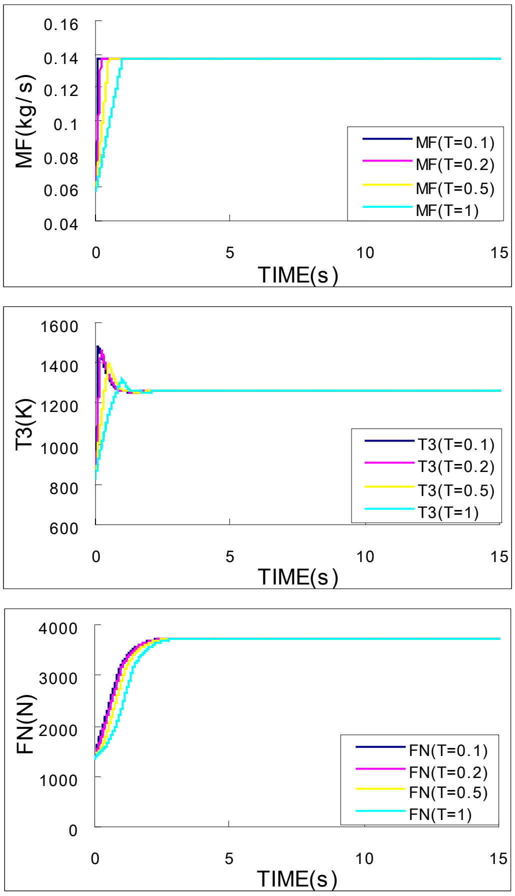


Fig.7. Transient analysis results of turbojet engine with step fuel increase case.



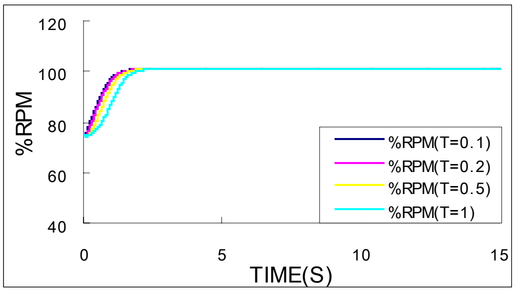
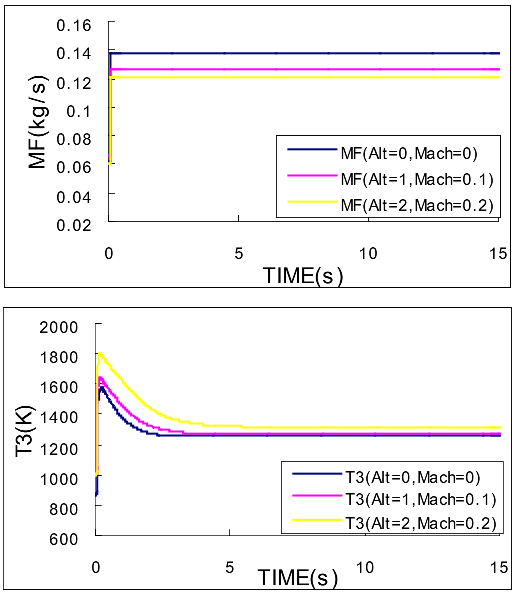


Fig. 8 Transient analysis results of turbojet engine with ramp fuel increase case

4.2 Transient behaviors of propulsion system with straight duct system (Fixed wing flight mode)

Transient analysis results of the propulsion with straight duct systems, which is fixed wing flight mode, are shown in the Fig.9. There are two cases of the step fuel flow increase schedule from 75% rpm to 95% rpm for the sea level static flight condition and the forward altitude flight condition with the altitude of 1km with the flight Mach number 0.1, respectively. As shown in Fig.10, the main nozzle net thrust of the forward altitude flight case is decreased less than it of the sea level static flight condition, and the turbine inlet temperature overshoot for all two cases are also severe as the main turbojet engine case.



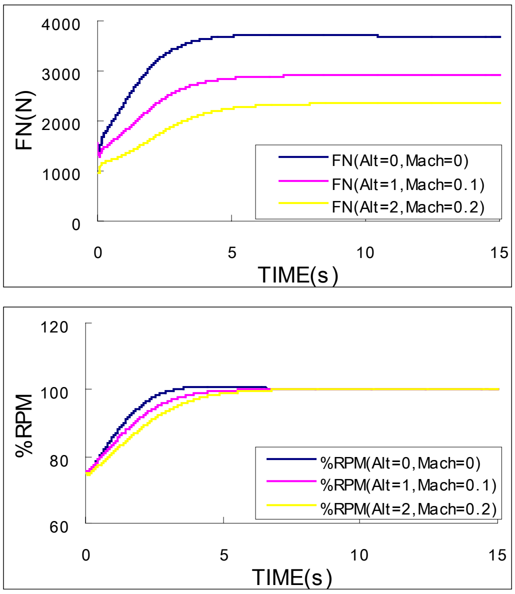
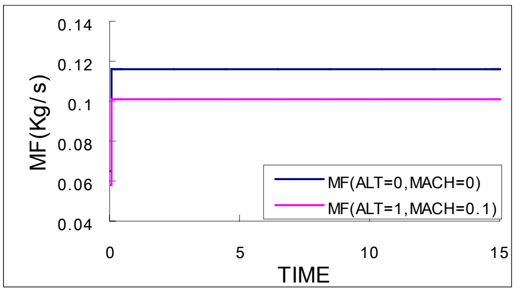


Fig.9. Transient analysis results of propulsion system with straight duct subsystem. (Cruising flight mode)

4.3 Transient behaviors of propulsion with curved duct system (Rotary wing flight mode)

Transient analysis results of the turbojet engine with the curved duct systems, which is rotary wing flight mode, are shown in the Fig. 10. There are also two cases of the step fuel flow increase schedule from 75% rpm to 95% rpm for the same flight conditions as the previous fixed wing cruising flight mode. As shown in Fig.10, the net thrust at the tip-jet nozzle of the rotary wing take/off landing flight mode case is decreased less than it of the sea level static flight condition, and turbine inlet temperature overshoots for all two cases of this flight mode are much severer than them of the forward flight mode case.



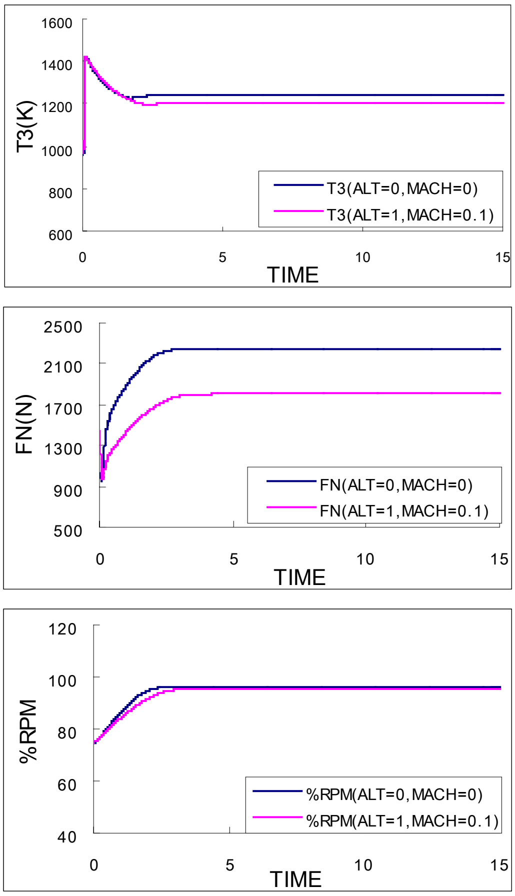


Fig. 10. Transient analysis results of propulsion system with curved duct subsystem. (Take-off/landing flight mode)

Conclusion

A new concept propulsion system, which can operate the flight vehicle in the vertical flight mode for take-off/landing using the rotary wing driven by the tip-jet nozzle as well as in the flight speed forward flight mode with the fixed wing using the main thrust nozzle, was modeled, and the transient performance analysis of this propulsion system was performed. The SIMULINK® program was used for transient performance modeling and analysis. Transient simulation performance utilized the ICV(Inter–component volume) method and the Euler integration method. Transient performance analysis was performed on 3 cases such as the main turbojet case for verification of the proposed simulation model, the propulsion system case including the curved duct with tip-jet nozzles for the take-off/landing flight mode and the propulsion system case including the straight duct with the main nozzle for the cruising flight mode. Fuel flow schedules to accelerate from idle to maximum rotational speed were divided into the step increase of the most severe case and ramp increase cases to avoid the overshoot of turbine inlet temperature, and variations of thrust, rotational speed and the turbine inlet temperature were investigated in some transient analysis cases.

In comparison of analysis results of the turbojet engine mode using the proposed SIMULINK® model and the commercial program GSP, it was confirmed that the validity of the proposed performance simulation program was verified because two program results were very close.

According to transient analysis of the propulsion system for the fixed wing flight mode, the main nozzle net thrust is decreased less than it of the sea level static flight condition, and the turbine inlet temperature overshoot for all two cases are also severe as the main turbojet engine case. In case of transient analysis of the propulsion system for the rotary wing flight mode, the net thrust of the tip-jet nozzles is case decreased much less than it of the sea level static flight condition, and the turbine inlet temperature overshoot for the rotary wing flight mode is severer than that of the forward flight mode.

In the near future, transient performance behaviors of the CRW propulsion system by valve operation will be additionally investigated.

References

[1] Kong, C., Kang, M., et al., 2003, "*A Study on Steady-state Performance Simulation of Smart UAV Propulsion System*", Journal of the Korean Society of Propulsion Engineers, Vol. 7, No. 3, pp. 38~44

[2] Bettocchi, R., Spina, P.R. et al., 1996, "*Dynamic Modeling of Single-Shaft Industrial Gas Turbine*", ASME 96-GT-332

[3] Crosa, G., and F. Pittaluga et al., 1998, "*Heavy-Duty Gas Turbine Plant Aero-thermodynamic Simulation Using SIMULINK*", Transaction of the ASME

[4] Kim S., 1999,"*Gas Turbine-Dynamic Simulation Using SIMULINK*", MSc Thesis, SME, Cranfield University

[5] Kong, C. and Rho, H., 2003, "*Steady-state and Transient Performance Simulation of Turboprop Engine using SIMULINK Model*", ASME Turbo Expo 2003 Conference, ASME 2003-GT-38181

[6] Fawke, A. J., 1970, "*Digital Computer Simulation of Gas Turbine Dynamic Behavior*", Ph.D. Thesis, University of Bristol, UK

[7] Math Works, 1997,"*SIMULINK: Dynamic System Simulation for MATLAB, Ver.2*", The Math Works, Inc., USA