ASPECTS REGARDING THE INTERNAL FLOW IN A TURBOJET ENGINE EQUIPED WITH AFTERBURNER COMBUSTION

AMADO STEFAN,
Military Technical Academy
81-83 George Cosbuc, 050141, Bucharest, Romania

CONSTANTIN NISTOR
Military Technical Academy
81-83 George Cosbuc, 050141, Bucharest, Romania

MIRCEA BOSCOIANU
Air Force Academy
160 Mihai Viteazul Street, 500183, Brasov, Romania

CALIN CIUFUDEAN
Stefan cel Mare University of Suceava
13 University str., 720229, Suceava, Romania

amadostefan@yahoo.com, atmnistor@yahoo.com, boscoianu.mircea@yahoo.com

Abstract: Afterburner is a long extension of the rear portion of the jet engine which mixes a large amount of the remained oxygen with the jet fuel which has entered into the turbine outlet and then burns the mixture. In this research, flow in the afterburner of aircraft engine is analyzed by numerical methods. Flow simulation inside of the afterburner considers two dimensional and axial symmetry to be turbulence and k-ε model has been used for turbulence analysis. Furthermore, Probability Density Function (PDF) method is used to model the combustion process inside the afterburner. The paper presents the results for two cases – when the afterburner system is used end not.

Key-Words: computational fluid dynamics, turbojet engine, afterburner

1 Introduction

An afterburner is used in the engines of military aircraft as a thrust-augmenting device during operations like take off, steep climb, quick acceleration and steep turns. The afterburner is located in a jet pipe downstream of turbine. The flow inside an afterburner is highly complex due to presence of struts, diffuser, fuel manifolds, flame stabilizer, screech holes and cooling holes. The flow is turbulent in nature and has high-pressure gradients and recirculation. An easiest approach is to consider the flow inside the afterburner to be axial symmetric. For 2D axial symmetric configurations, the axial and radial momentum conservation equations are given by:

where

\[ \nabla \bar{v} = \frac{\partial v_x}{\partial x} + \frac{v_y}{r} \]

and \( v_z \) is the swirl velocity.

In case of reactive flows, the conservation equations for chemical species, the local mass fraction of each species \( Y_i \) through the solution of a convection-diffusion equation for the i-th species. This conservation equation takes the following general form:

\[ \frac{\partial}{\partial t} \left( \rho Y_i \right) + \nabla \cdot \left( \rho \bar{v} Y_i \right) = - \nabla \cdot \bar{J}_i + R_i + S_i \]

where \( \bar{J}_i \) is the diffusion flux of species i, which arises due to concentration gradients, \( R_i \) is the net rate of production of species i by chemical reaction and \( S_i \) is the rate of creation by addition from the dispersed phase plus any sources. An equation of this form will be solved for
N − 1 species where N is the total number of fluid phase chemical species present in the system.

In turbulent flows, the mass diffusion in the following form:

\[ \bar{J}_i = -\left( \rho D_{i,m} + \frac{\mu_t}{Sc_t} \right) \nabla Y_i \]

where \( D_{i,m} \) is the diffusion coefficient for species \( i \) in the mixture, \( \mu_t \) is the turbulent viscosity, \( Sc_t = \rho \cdot D_t \) is the Schmidt number and \( D_t \) is the turbulent diffusivity.

2 Numerical approach

Figure 1 present the computational domain considered. Only the ring flame stabilizer and also equivalent injectors is model as shown in figure 2. The numerical determination is made with Fluent software.

![Fig. 1 Description of the fluid domain](image1)

![Fig. 2 Detailed mesh](image2)

The boundaries conditions are: mass flow rate at 66 kg/s, 900°C temperature, 10% turbulence intensity, and oxygen concentration 0.18 at the exit of the turbine is considered to and at the output of the computation domain of 283°C and 0.21 oxygen concentrations. The fluid is compressible and the \( k - \varepsilon \) turbulence model is considered. Results are presented below for different angles of the exit nozzle.

3 Flow after the turbine without afterburner system

Figures 3 to 8 present Mach number and temperature fields for three cases: 8°, 0° and -6° nozzle angle.

![Fig. 3 Mach number for 8° nozzle](image3)

![Fig. 4 Temperature for 8° nozzle](image4)

![Fig. 5 Mach number for 0° nozzle](image5)
Behind ring flame stabilizer are formed areas with higher temperatures and lower flow velocity. Mach number and temperature variations depending on angle of nozzle and are presented in Figures 9 and 10, at engine axis in the nozzle area.

There is a decrease in flow velocity on engine axis to the engine output and an increase in the vicinity of exit section with increasing angle of nozzle, from -6° to 8°. The temperature increase inside and decrease outside the exit section. Figures 11 and 12 are shown Mach number and temperature variations with nozzle radius, in the engine exit section depending on the nozzle angle.
Fig. 12 Temperature variations with nozzle radius in nozzle exit area

The increase of velocity in the engine exit section is observed at variations along the nozzle radius. At the middle radius the velocity has a maximum value for all nozzle angles at the engine exit section.

4 Flow after the turbine with afterburner system

In order to establish fuel flow through the injectors equivalent (rings) there are the following considerations. The three fuel manifolds are characterized by:

Collector I: - Has a diameter of 485 mm and 22 injectors.

Collector II: - Has a diameter of 577 mm and 45 injectors.

Collector III: - Has a diameter of 700 mm and 60 injectors.

Fuel used: C12H23 (Kerosene). The burnt fuel mass flow is 0,882 kg/s. Considering that it is distributed evenly over all the injectors, resulting flows:
- 0,152 kg/s for 485 mm diameter;
- 0,313 kg/s for 577 mm diameter;
- 0,417 kg/s for 700 mm diameter.

The fuel injection is upstream. The Probability Density Function (PDF) method is used to model the combustion process inside the afterburner. Figures 13 to 18 presents the Mach numbers and temperatures fields, for 8°, 0° and -6° nozzle angle.
Compared with the absence of afterburner system case, when the afterburner system is used the flow velocity at engine exit. There is a significant increase in the flow temperature after fuel injection. Figures 19 and 20 presents the velocity and temperature variation on engine exit axis depending on the nozzle angle.

Fig. 18 Temperature at -6°

There is a decrease of velocity inside the engine and an increase after the engine exit with increasing nozzle angle from -6° to 8°.

Fig. 19 Mach number variations with nozzle angle in nozzle exit area

Fig. 20 Temperature variation with nozzle angle in nozzle exit area

Figures 21 and 22 present the Mach number and temperature variations along the nozzle radius in the output section of the engine.

Fig. 21 Mach number variations with nozzle radius

Fig. 22 Temperature variations with nozzle radius in nozzle exit area

Significant differences appear only temperature distribution in the wall, where the temperature decreases with increasing nozzle angle.
5 Conclusion and future work

The internal flow simulation inside of the afterburner of a turbojet is based on a 2D and axial symmetric 3D computation and is based on a modified k-ε and PDF (Probability Density Function) methodology. The focus is on a comparative analysis between two cases – when the afterburner system is used end not. In Figures 23 and 24 are presented the Mach number variations along the nozzle radius at output section, with and without afterburning system; example for the nozzle angle 8° and -6°.

It is interesting to remark the efficiency at low Mach. Future work should also consider the scalability of this system of afterburning, together with finding new solution to extend the efficiency at higher Mach numbers.

References: