MATHEMATICAL MODEL AND CFD ANALYSIS OF PARTIALLY PREMIXED COMBUSTION IN A TURBOJET

Constantin ROTARU, George Cristian CONSTANTINESCU, Oliver CIUICĂ, Ionică CÎRCIU, Eduard MIHAI
”Henri Coandă” Air Force Academy, Braşov, Romania (rotaru.constantin@afahc.ro, ccg_cristi@yahoo.com, oliverciuica@yahoo.com, circiuionica@yahoo.co.uk, mishued@outlook.com)

DOI: 10.19062/1842-9238.2016.14.2.10

Abstract: *In this paper are presented some results about the study of combustion chamber geometric configurations that are found in aircraft gas turbine engines. The CFD simulations were made with jet A fuel (which is presented in the Fluent software database) for an annular flame tube with 24 injectors. The temperature profile at the turbine inlet exhibits nonuniformity due to the number of fuel injectors used in the circumferential direction, the spatial nonuniformity in dilution air cooling and mixing characteristics as well as other secondary flow patterns and instabilities that are set up in the flame tube. The results show that a spiral (helicoidal) holes distribution could improve the flame tube efficiency, turbulent vortex influence and the actual rate of heat release.*

Keywords: *partially-premixed combustion, aircraft engine, flame tube, turbine temperature profile, flame stability.*

1. INTRODUCTION

The essential elements of turbomachines are the blade rows where the whole flow is the result of blades network which impart the force and more relevantly, the moment of the flow. In axial compressors and turbines it is often possible to consider flows in rotor passages by merely adopting a moving frame of reference and for stator blades a fixed coordinate system. The pressure and shear stresses in the blades of a compressor or turbine produce a moment about the axis of rotation that could be evaluated and integrated but instead it is usual to consider the momentum for the flow entering and leaving. Because the tangential stresses at the casting and the hub are usually very small, the total work of the blade row can be inferred from measurements of the velocity components upstream and downstream. Radius changes may have important effects on axial machines where for each stages the static enthalpy might be approximately with $0.4U^2$, where U is the local blade speed. A small change in radius can produce changes in static enthalpy of the same order of magnitude as those produced by the deflection and deceleration of the flux in the blades [1].

A gas turbine engine is usually specified for operation at a design point, so that it will develop a value of the pressure ratio with a predicted value of thermodynamic efficiency at a value of the shaft rotational speed, while a design value of the mass flow rate exists in each stage. The flow rate may vary from one stage to the next because of bypass elements or extraction of the flow for cockpit pressurization or for fuel flow transfer.

All of the design requirements are associated with a particular working fluid and with the design values of total pressure and total temperature distribution in the engine.
The flow is viscous and compressible, with regions of both supersonic and subsonic local flow. Local regions of laminar, transitional and turbulent boundary layer flow may exist, accompanied by separation of flow from the blades or end walls and both the blade surfaces and the end wall boundary layers are generally three-dimensional.

2. AXIAL FLOW GASDYNAMICS

The mathematical model of the flow field in axial turbomachines is based on the concepts of streamlines, stream surfaces and stream tubes in the flow path. In order for streamlines and path lines to coincide, the flow must be steady, this being and additional restrictive assumption in gas turbine study. The combustion gases enter the turbine in a nonuniform distribution, which is random in the circumferential direction. The randomness of the temperature distribution causes the first stator stage to be designed for the maximum temperature of fluid coming out of the combustion chamber (Fig. 1).

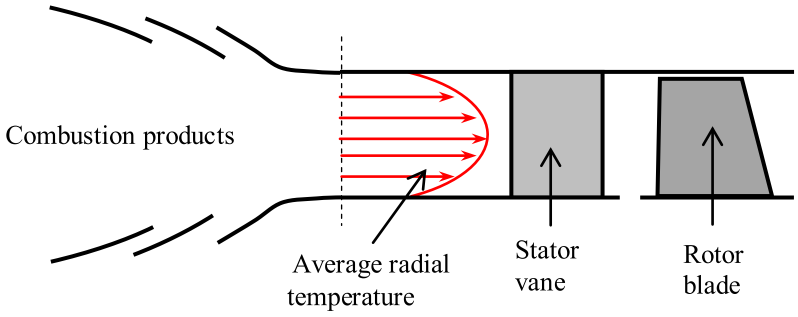


FIG. 1. Turbine blade radial temperature profile

The first rotating blade row is exposed to lower temperatures because of circumferential averaging, dilution of turbine gases with first stator vane cooling air and relative velocity effects. The second stator vane is exposed to a lower temperature because of cooling air dilution, work extraction from the turbine gases and mixing that dilutes the hot spots (Fig. 2). The first stator vane is fed by compressor discharge air that bypasses the combustor, because it requires a very high supply pressure. The first rotor blade is also fed by compressor discharge air. This amount of air is accelerated through a row of nozzles pointed in the direction of rotation [2]. The effect of this is to reduce the amount of work required to pump the cooling air and to reduce the cooling air temperature within the blade. In gas turbines, the heat transfer coefficients to the blades are very high and the conductivities of the materials are fairly low.

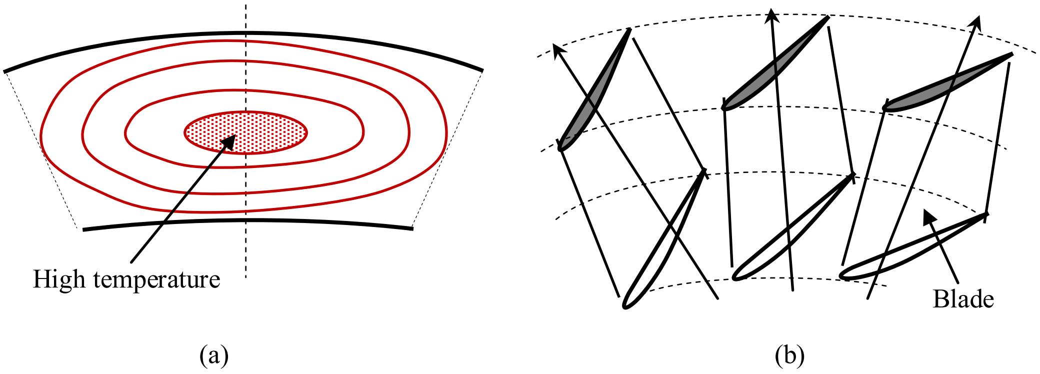
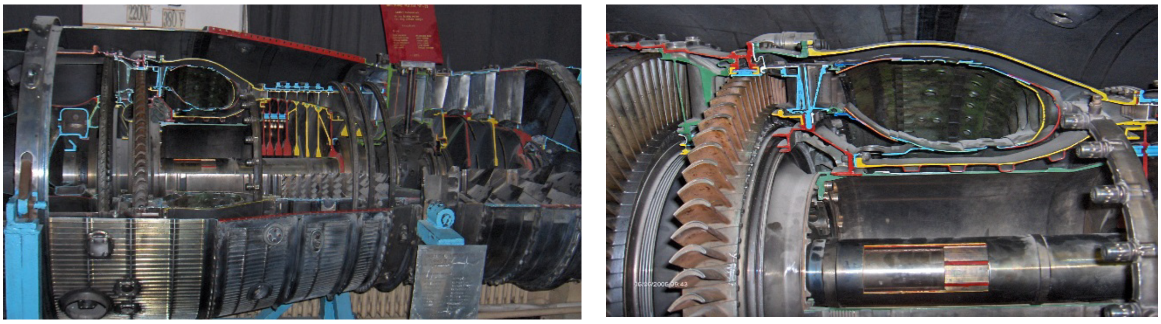


FIG. 2. The radial and circumferential temperature distribution for an annular combustion chamber (a) and surfaces in blade-to-blade flow passage (b)

This combination makes it imperative that the heat transfer coefficient distribution over a hole airfoil must be known in detail. At the stagnation point this coefficient can be correlated as that of the stagnation point of a cylinder in a cross flow that is affected by the freestream turbulence.

On the concave or pressure side of the blade airfoil, the boundary layer trips to turbulent flow and attains a heat transfer coefficient level corresponding to a low Reynolds number turbulent flow. On the convex or suction side of the blade airfoil, the boundary layer is often at first laminar, but has heat transfer coefficients higher than those corresponding to laminar flow because of the effects of freestream turbulence. The heat transfer coefficient distribution to a blade airfoil is calculated utilizing two-dimensional boundary layer theory. The end wall regions are exposed to a lower gas temperature than the corresponding airfoils. In developing the governing integral equations which describe the flow in the turbine, the local density appears under the integral sign. Excepting the case of incompressible flow with small temperature differences, it becomes necessary to determine the variation of the density across the boundary layer. Since the static pressure is constant across the boundary layer, the gas law gives the product of local density and local temperature as constant, thus the problem becomes one of specifying the local temperature [3].

In the usual steady turbulent boundary layer equations, if the laminar transport term is not neglected with respect to the turbulent transport, the governing equations are everywhere parabolic. In certain cases, the neglect of the laminar transport term can result in a hyperbolic set of equations, but this depends on the precise nature of the model of the turbulence transport. The boundary layer equations are nonlinear when they are expressed in term of velocity variables but implicit methods make a formal linearization at some points. In the axial turbojet engine the hub and casing boundary layers can be thought of as a swirling axially symmetric boundary layers. The hub and casing surfaces are often not continuous and frequently stationary surfaces lie adjacent to the rotating components. Leakage and coolant flows often occur through gaps in the surface, but all these difficulties shrink to insignificance when compared to the effect of the blade of stator rows (Fig. 3).



(a) (b)

FIG. 3. The aircraft jet engine (a) and the combustion chamber (b)

In the pitch averaging approach, the azimuthal variations are expressed as a mean value plus a small perturbation and the equations of motion averaged in the azimuthal direction. Within turbomachinery, three-dimensional flows are encountered with unfortunate regularity in the boundary layers and the freestream. An additional feature of the extended boundary layer equations concerns the absence of zones of influence which are the regions bounded by the body surfaces, the boundary layer edges and a pair of characteristic surfaces, one of the surface normal envelope passing through the external streamline and the other, the surface normal envelope passing through the surface streamline.

3. DEVELOPMENT OF THE NUMERICAL METHODOLOGY FOR CFD CALCULATION

A basic structure model for separated flow regions interacting with an inviscid outer flow region leads to expect considerable economy of computation relative to solving the full Navier-Stokes equations. If the external flow can be assumed irrotational, a single scalar potential equation may be used and the correction perturbation to this inviscid outer flow may be obtained from linearized theory (in transonic flow some perturbation potential must be solved throughout the flow). The conventional boundary layer approach leads to a parabolic system of equations but the interacted boundary layer is not well posed as an initial value condition [4].

The governing equations for interacting boundary layer can be written in the form

$$\begin{cases} (\rho u)_x + (\rho u)_y = 0 \\ \rho u u_x + \rho v u_y = -p_x - \left(\mu_T u_y\right)_y \end{cases} \tag{1}$$

where μ_T is the sum of the laminar and turbulent effective viscosity, x and y subscripts denote differentiation, u and v are the velocities, p_x is the pressure gradient and ρ is the gas density.

These equations are subject to the wall and freestream boundary conditions

$$\begin{cases} y = 0, \ u = v = 0 \\ y = \delta, \ p = p_e(x) \end{cases} \tag{2}$$

where δ is the thickness of boundary layer.

The rotating external blade boundary layer are very thin and the effects of the Coriolis and centrifugal forces on the mean flow are negligible.In partially-premixed combustion, fuel and oxidizer enter the reaction zone in distinct streams and the mixture fraction$f$ can be written in terms of the atomic mass fraction as [5]:

$$f = \frac{Z_i - Z_{i.oxidizer}}{Z_{i.fuel} - Z_{i.oxidizer}} \tag{3}$$

where Z_i is the elemental mass fraction for element i.

The mean time averaged mixture fraction equation is:

$$\frac{\partial}{\partial t}\left(\rho \bar{f}\right) + \nabla \cdot \left(\rho \bar{v} \bar{f}\right) = \nabla \cdot \left(\frac{\mu_t}{\sigma_t} \nabla \bar{f}\right) + S_m \tag{4}$$

The source term S_m is due to solely to transfer of mass into the gas phase from liquid fuel.

The mixture fractions variance, $\overline{f'^2}$ could be find from equation:

$$\frac{\partial}{\partial t}\left(\rho \overline{f'^2}\right) + \nabla \cdot \left(\rho \bar{v} \overline{f'^2}\right) = \nabla \cdot \left(\frac{\mu_t}{\sigma_t} \nabla \overline{f'^2}\right) + C_q \cdot \mu_t \left(\nabla^2 \overline{f'^2}\right) - C_d \rho \frac{\varepsilon}{k} \overline{f'^2} \tag{5}$$

where $f' = f - \bar{f}$ and μ_t is the subgrid-scale viscosity.

The $p(f)$probability density function can be thought of as the fraction of time that the fluid spends at the state f, so

$$p(f)\Delta f = \lim_{T \to \infty} \frac{i}{T} \sum_i \tau_i \tag{6}$$

where T is the time scale and τ_i is the amount of time that f spends in the Δf band.

The analytical shape of the function $p(f)$ depends on the turbulent fluctuations in f and it is expressed as a mathematical function that approximates the shapes. The temporal fluctuations describing by function $p(f)$ can be used to compute the time averaged values of variables that depend of f with the equation:

$$\overline{\varphi_i} = \int_o^1 p(f)\varphi_i(f)df \tag{7}$$

where

$$\varphi = \frac{(fuel/air)_{actual}}{(fuel/air)_{stoichiometric}} \tag{8}$$

The expression of $p(f)$ is

$$p(f) = \frac{f^{(\alpha-1)}(1-f)^{(\beta-1)}}{\int f^{(\alpha-1)}(1-f)^{(\beta-1)}df} \tag{9}$$

where

$$\alpha = \bar{f}\left[\frac{\bar{f}(1-\bar{f})}{\overline{f'^2}} - 1\right];\ \beta = (1-\bar{f})\left[\frac{\bar{f}(1-\bar{f})}{\overline{f'^2}} - 1\right] \tag{10}$$

At each point in the flow field, the $p(f)$ can be computed and used as the weighting function to determine the time-averaged mean values of species mass fraction, density and temperature.

4. NUMERICAL RESULTS

The effect of turbulence on premixed flame speed is to enhance momentum transfer between the burning front and the unburned reactants. In addition, turbulence increases the total surface area of the flame and hence increases the heat transfer between the reaction zone and the unburned gas. A recirculation zone in the burner is needed to stabilize a premixed flame. To stabilize the flame in the primary zone of a main burner, air is introduced through single or double rows of swirl vanes and in order to create a stirred reaction zone in the flame tube, part of excess air is injected through the primary air holes as radial jets.

Some important CFD results and flame tube geometrical shapes are presented in the following pictures.

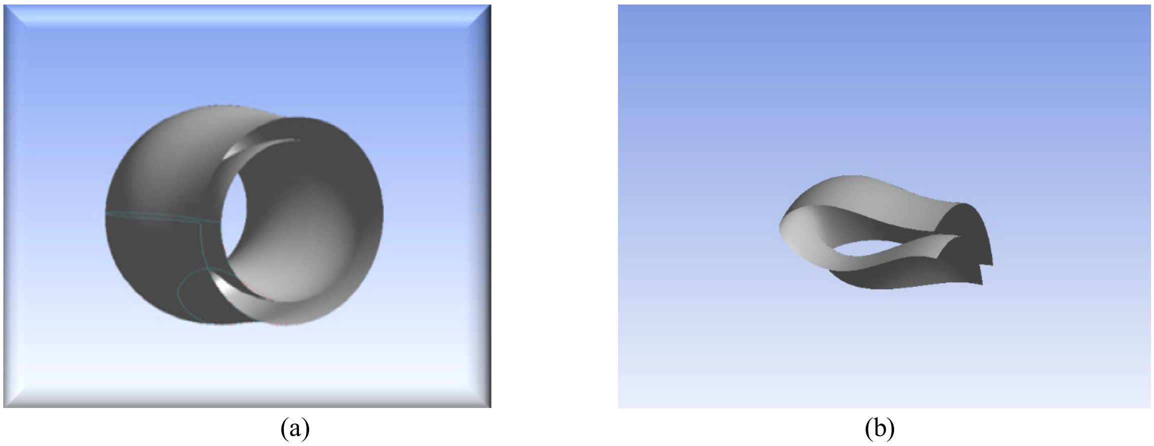


FIG. 4. The geometrical shape of the flame tube, 3D view (a) and quarter view (b)

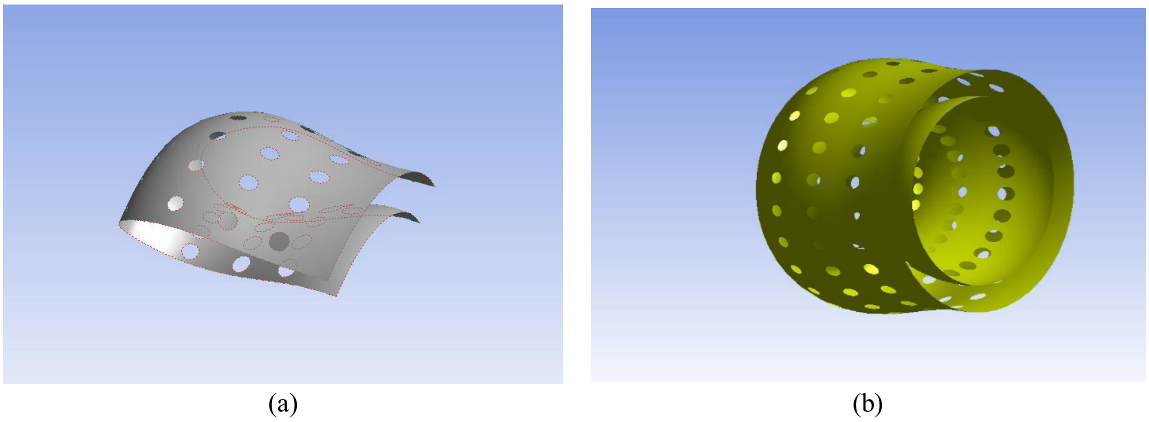


FIG. 5. The hole distributions, quarter view (a) and 3D view (b)

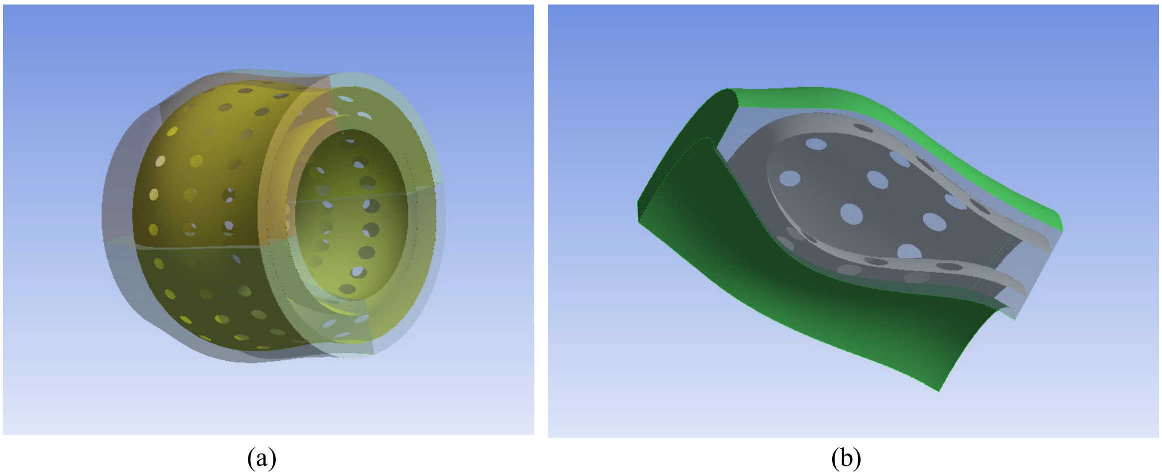
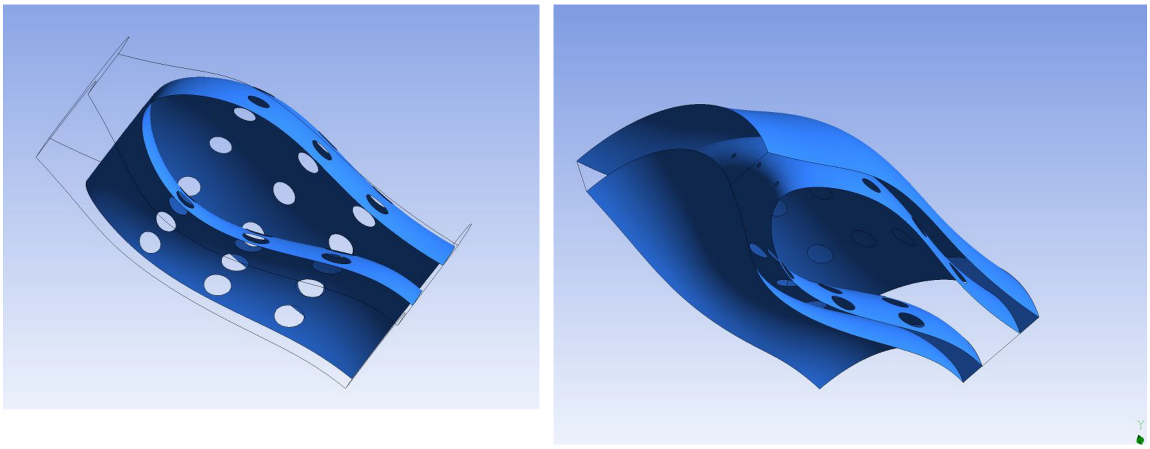
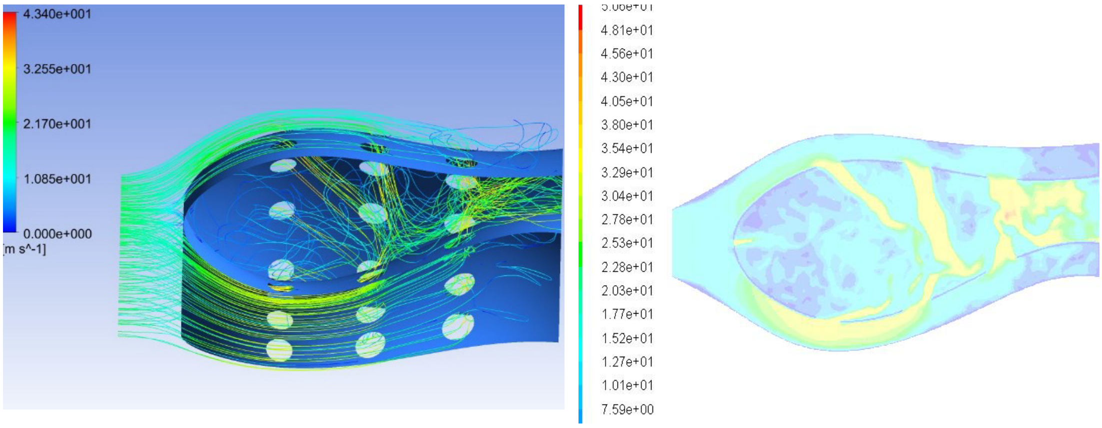


FIG. 6.Combustion chamber geometrical shape (a) and inner view (b)



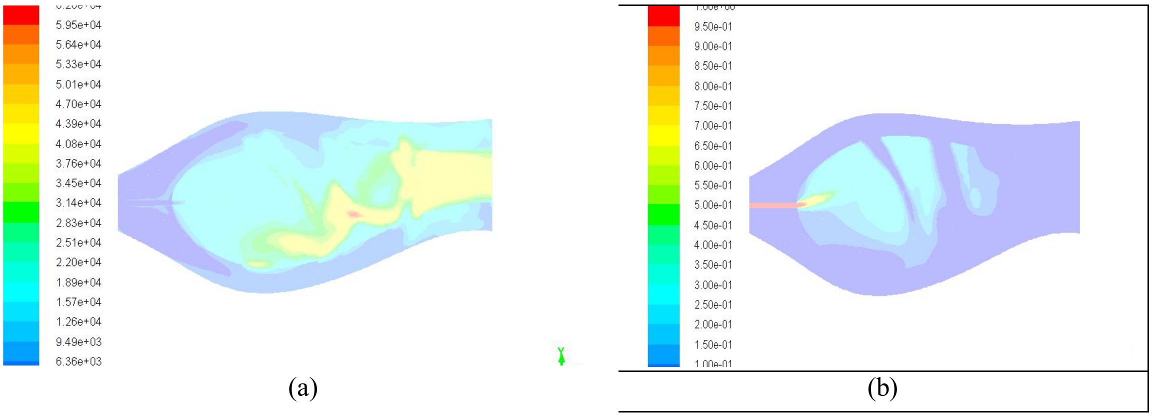
(a) (b)

FIG. 7. The inlet section (a) and the injectors holes (b)



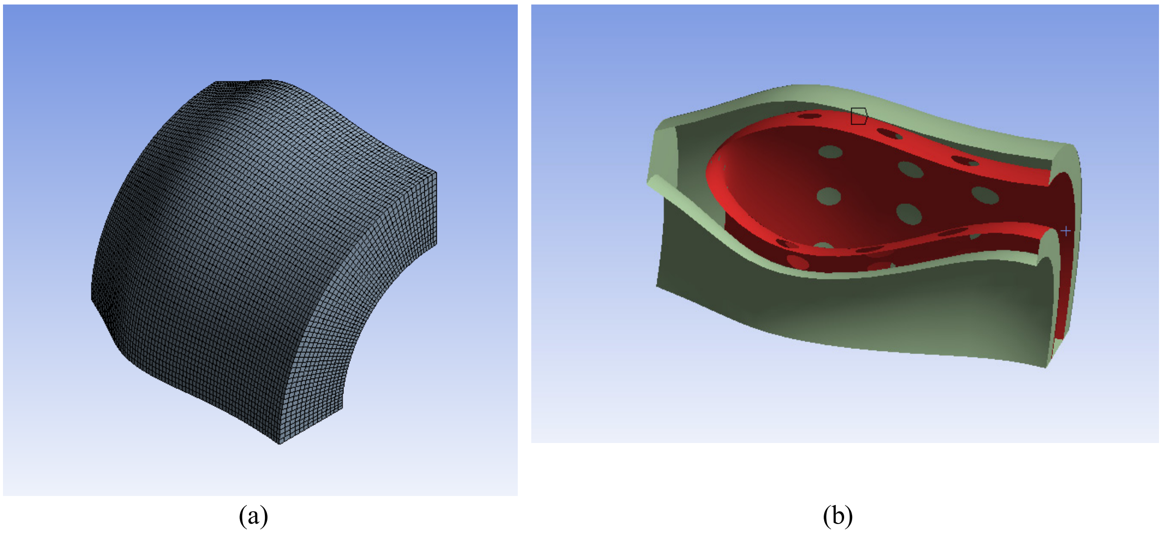
(a) (b)

FIG. 8. Stream lines (a) and velocity distribution (b)



(a) (b)

FIG. 9. Turbulent intensity (a) and Mass fraction of jet A fuel (b)



(a) (b)

FIG. 10. The flow domain mesh (a) and flame tube wall position (b)

CONCLUSIONS

The model used in this paper to describe the combustion process in an annular flame tube has the advantage of cleanly separating aerodynamic and chemical features of the process. The ignition time was calculated from a global model of the chemical reaction rate and the chemical concentrations and temperature used in the combustion chamber are taken from a mathematical model of the mixing zone that is based on time-averaged measurements. Concentration profiles for the products of combustion are similar to the temperature profiles and the vortex structures lie in the region with a strong average velocity gradient.

AKNOWLEDGMENT

This paper has been financially supported within the project entitled “Gas Turbine using in situ combustion”, contract number 286/01.07.2014 (PN-II-PT-PCCA-2013).

REFERENCES

[1] C. E. Moldoveanu, A. Giovannini, H. C. Boisson, *Turbulence receptivity of longitudinal vortex dominated flow,* Lecture Notes in Computational Science and Engineering, Vol. 69: BAIL 2008-Boundary and Interior Layer Proceedings of the International Conference on Boundary and Interior Layers-Computational and Asymptotic Methods, Limerick, Springer Verlag, Berlin, pag. 227-236, ISBN: 978-3-642-00604-3, July, 2008;

[2] A. N. Tudosie, *Supersonic Air Inlet Control System Based On The Inner Channel Minimum Cross-Section Position Control,* International Conference on Military Technologies ICMT 2009, Brno, CzechRepublic, 2009;

[3] C. Rotaru, I. Cîrciu, C. Aramă, C. G. Constantinescu, *Aspects Regarding Velocity Distribution in the Secondary Zone of a Gas Turbine Combustor,*Review of the Air Force Academy, No. 3(30), DOI: 10.19062/1842-9238.2015.13.3.5, pp. 33-37, 2015;

[4] C. Rotaru, M. Mihăilă-Andres, P. G. Matei, *An Extended Combustion Model for the Aircraft Turbojet Engine*, International Journal of Turbo & Jet Engines, Volume: 31, Issue: 3, Pages: 229-237 , 2014*;*

[5] I Cîrciu, V. Prisacariu, C. G. Constantinescu, *Gas Turbine Mixed Jet Flows Based on the Coandă Effect,* Review of the Air Force Academy, No. 3(30), DOI: 10.19062/1842-9238.2015.13.3.4, pp. 27-32, 2015.