INFLUENCE OF GEOMETRY AND TYPE OF COOLING HOLES ON THERMAL LOAD OF COMBUSTION CHAMBER

Marcin Swiatek, Roman Domański

Institute of Aviation
Krakowska Av. 110/114, 02-256 Warsaw, Poland
tel.:+48 22 577 3276, fax: +48 22 8464432
e-mail: marek.lazarczyk@yahoo.com, roman.domanski@itc.pw.edu.pl

Abstract

The objective of this thesis is to compare various methods of combustor wall cooling and their effectiveness by numerical simulations.

It was determined that the first task was to verify how much air is coming through single axial hole with 3.5% pressure drop between hot and cold part of combustion chamber. The results from this flow check serve as a base template for generating more accurate and precise models of single axial hole cooling as well as calculation of hole diameter for multihole cooling.

Second task was to generate more sophisticated single hole model with boundary layer in order to better simulate the conditions in areas near the combustion chamber walls and get more accurate results. The same method was used to create multihole model. In order to compare efficiency, all created domains in every model have the same volume, model settings, operating and boundary conditions.

*Geometry of all models described above is created using SIMENS NX4 and SIMENS NX5 program based on drawings obtained from available literature, and data acquired from the Internet. The discretization into a structural finite volume grid took place in commercial pre-processor GAMBIT® (*GAMBIT and FLUENT – commercial CFD codes from Ansys Inc.*). The airflow and heat exchange will be calculated using program FLUENT®.*

The results were shown in the thesis in terms of several comparative pictures of the temperature fields in the combustion chamber domain, and graphs representing difference in temperature fields on cooling wall of the combustion chamber.

Keywords: *aircraft engines, engine combustion chamber, combustor cooling, CFD*

1. Introduction

The objective is three-dimensional analysis of heat transfer through combustion chamber walls, determining the cooling effectiveness and verifying which type of cooling provides better protection against hot gases. Next chapters show results of analysis of axial hole cooling and multihole cooling.

All models were created using professional software for modeling Unigrapics NX5/NX6. Investigated cooling methods were based on commonly available literature. The part of combustion chamber presented in this thesis is a generic model of combustion chamber. Final models had several simplifications in geometry as well as in boundary conditions. Nugget of axial hole cooling method had omitted blends, simplified nugget geometry; hole axis was parallel to the liner wall. Cooling holes in multihole method were created perpendicular to the liner wall. First model created for analysis was a single axial hole presented on Fig. 1.

The upper blue volume represents the combustor passage through which the cooling air flows. The gray part of model contains a liner with single cooling slot with axial hole, it represents the metal. Lastly the red volume represents the interior of combustion chamber.

Preliminary analysis of axial hole was performed for several geometric variations of hole shape in order to match the mass flow rate in all methods for further comparison. Axial hole model was analyzed for three cases of half-cone angle, therefore for each case separate model was created.

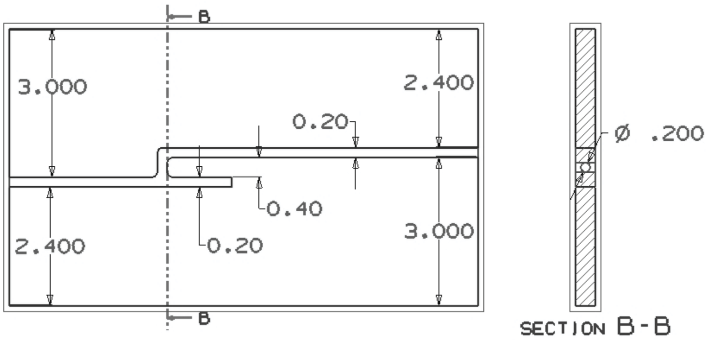

Fig. 1. Drawing of cooling slot with axial hole (All dimensions are in inches)

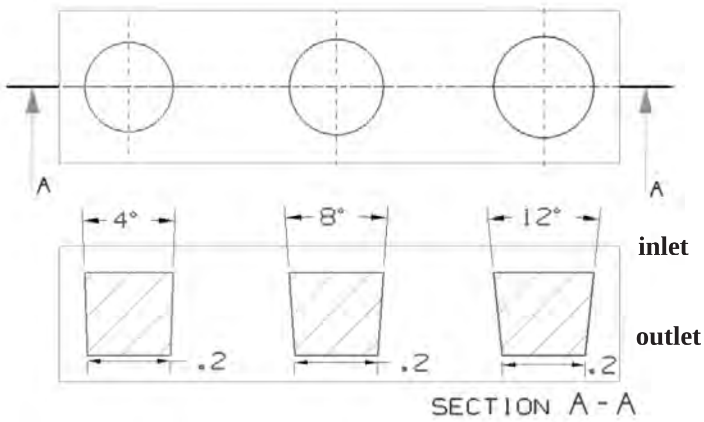

Fig. 2. Types of axial holes geometry (All dimensions are in inches)

Next model was a straight liner with twenty smaller holes (zero half-cone angle). The sketch with multihole holes is presented on Fig. 3.

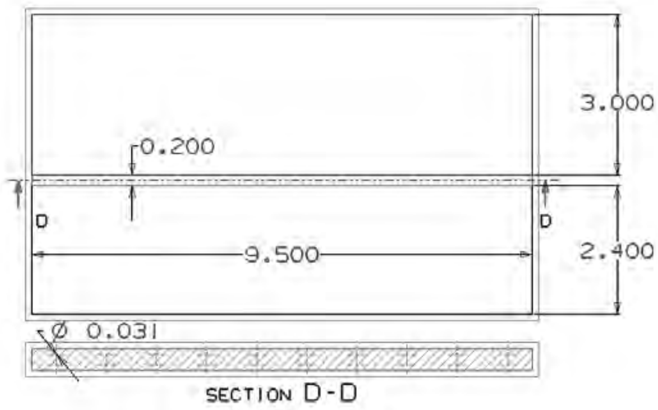

Fig. 3. Sketch of liner with multihole cooling holes (All dimensions are in inches)

Setting boundary conditions is a final stage of discretization process in GAMBIT. Each volume in the model needs specification of a continuum type (Fig. 4.). Therefore every volume in blue and red areas was allocated to the FLUID type, while volumes in gray area were allocated as SOLID type.

Boundary types of the faces of inlet and outlet to the cooler and hotter parts set (Fig. 4.) to PRESSURE_INLET type for the inlet faces and PRESSURE_OUTLET type for outlet faces. The lower wall of hot domain was set to SYMMETRY type and upper face of cold domain was set to WALL type. Finally the meshes were exported to the solver, which was Fluent program.

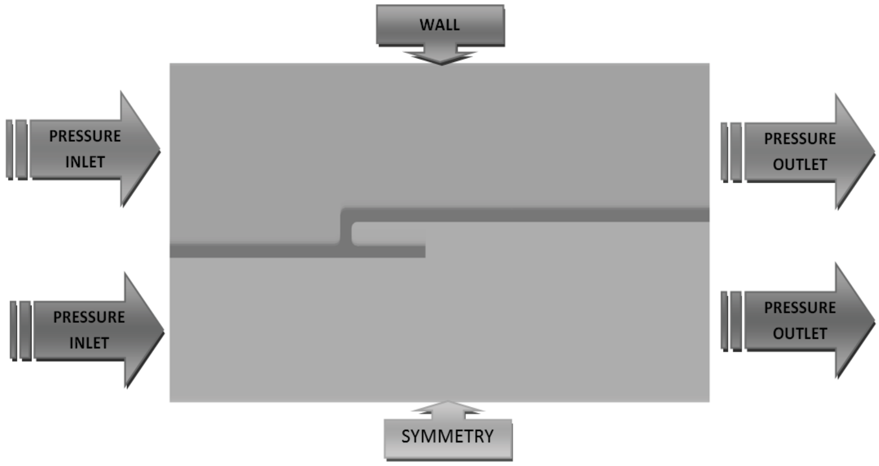

Fig. 4. Boundary conditions

All numerical analysis were performed using professional program FLUENT 6.3.26 installed on the computer equipped in 2 GB of RAM memory and Intel Core2Duo 1.87GHz processor. Computing power of the computer mentioned above allowed performing analysis on models which mesh size did not exceed 2mln controlled volumes (Fig. 5.).

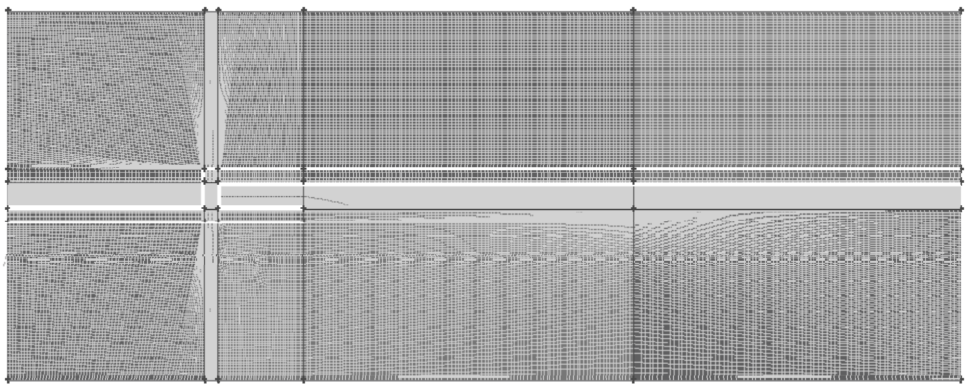

Fig. 5. Axial hole cooling system mesh

The processor power was able to perform approximately 1000 iteration per 24 hours. Every flow check analysis took 2000-3000 iterations and final models due to complexity and required accuracy were analyzed up to 10000 iterations.

First step after successful mesh upload is a grid scaling procedure and creating periodic boundaries on sidewalls of the model. Grid check procedure is next very important procedure, which always should be done in order to determine mesh integrity and avoid any errors later on. If the grid check fails the uploaded mesh have unacceptable errors, which will result in model crashing.

Solver settings determine type of analysis. All the settings were chosen to produce as accurate results as it is possible taking under consideration available computing power. Next important set up is choosing the viscous model. Since Y-plus level is below 5 than the near wall treatment must be set to Enhanced Wall Treatment.

Discretization in solution controls are set to First Order Upwind (better convergence) for first 2000 iterations, afterwards all parameters are changed to Second Order Upwind (more accurate results).

Setting material data. The physical parameters of air are collected in Tab. 1.

The liner walls are made of nickel whose properties are taken from the Fluent material database.

Tab. 1. Air material properties

Air		
Density	ideal gas	
Cp	piecewise-linear	
	Temperature [K]	Cp Value [J/kgK]
point 1	300	1007
point 2	500	1030
point 3	700	1087
point 4	900	1120
point 5	1200	1175
point 6	1300	1190
point 7	1400	1208
point 8	1500	1230
point 9	2000	1337
point 10	2500	1665
point 11	3000	2726
Thermal Conductivity	0.0242	W/mK
Viscosity	sutherland	
Molecular Weight	28.966	kg/kgmol

Tab. 2. Nickel material properties

Nickel		
Density	8900	kg/m3
Cp	460.6	J/kgK
Thermal Conductivity	91.74	W/mK
Molecular Weight	28.966	kg/kgmol

Finaly boundary conditions must be setup in the code. Previously created material needed to be related to the appropriate domains. The operating pressure in set to 144,924 psi (999216 Pascal's) Pressure inlets and pressure outlets in every domain have also been set to parameters listed in Tab. 3. In the domain representing combustion chamber temperature is assumed to be 2000K, and to the domain with cooler air has temperature assigned at 500K (1).

Tab. 3. Parameters used In analysis. Pressure defined in this table is overpressure above operating pressure

	Cold domain	
Temperature	500	K
Pressure inlet	4.0889	psi
Pressure outlet	3.6231	psi
	Hot domain	
Temperature	2000	K
Pressure inlet	0.1137	psi
Pressure outlet	0	psi

Parameters listed in Tab. 3. have been set in every analysis in the same manner.

2. Preliminary models – flowcheck

Preliminary model created in order to include a vena contracta effect in flow through the model holes. All flow check models are simplified. The liner walls are adiabatic and the boundary layer is not modelled, what allows an analysis of all models in the same amount of CPU time, as well as boundary layer was not modelled. Furthermore the solution converges after approximately 1000 iterations.

Single axial hole cooling was the first model which was analyzed. For better and more accurate results several cases were taken under consideration. Axial hole with 2, 4 and 6 half-cone angle degree are presented in this chapter. For every half-cone angle discharge coefficient was calculated in order to select the closest match for comparison with single radial hole cooling.

Flowcheck analysis showed that with increase of the half-cone angle the pressure is dropping. Increase of taper also results in decrease of turbulent kinetic energy. The discharge coefficient value also varies from 0.897 for 2 degrees half-cone angle to 0.963 for 6 degree half-cone angle (Tab. 4.) (1). Based on this results in further analysis the 2 degrees half-cone angle is used in order to acquire smaller difference in mass flow rate.

Tab. 4. Axial hole discharge coefficient

Axial hole			
Half-cone angle	2	4	6
Cd	0.897	0.941	0.963

Preliminary single hole diameter for multihole model was calculated from area of single axial hole divided by number of holes (20). Purpose of analyzing this model was to determine the difference in overall mass flow between single axial hole and several smaller holes. Due to simplification in geometry the no taper were applied in multihole model. For the same mass flow rate the maximal velocity in case of multiholes is 20% lower than the velocity achieved in case of axial hole.

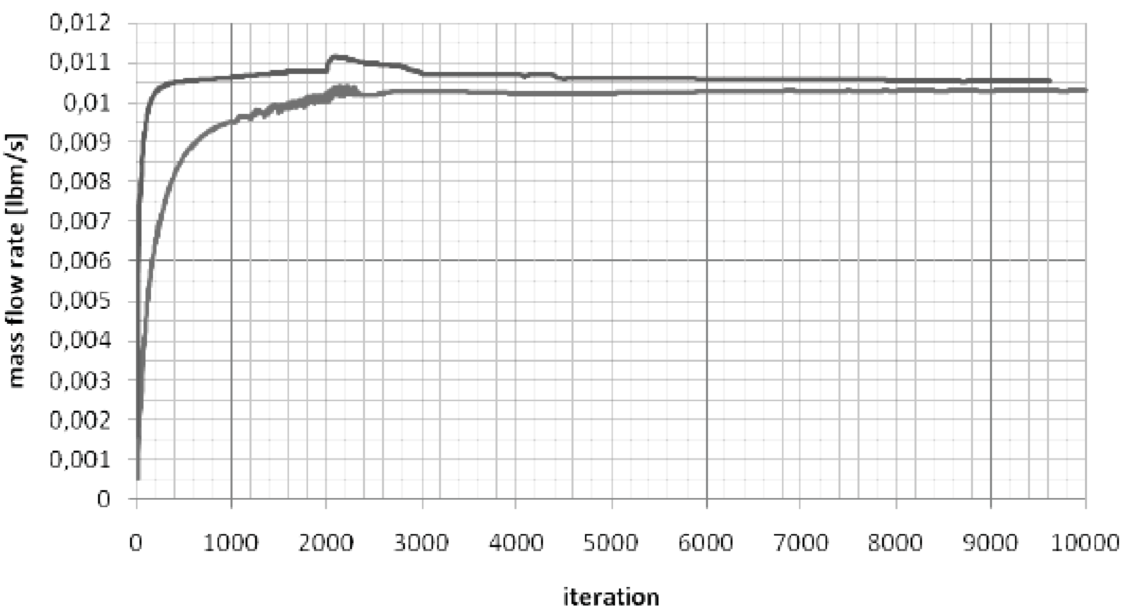

Fig. 6. Comparison of mass flow rate for four different cooling systems

3. Final model – single axial hole

Final model has 5% (1) smaller diameter of axial hole in order to better match mass flow in other analyzed cases, include the analysis of the metal wall, and the hot and cold domains as well as liner have been extended in order to eliminate problems with cooling film creation and reversed flow.

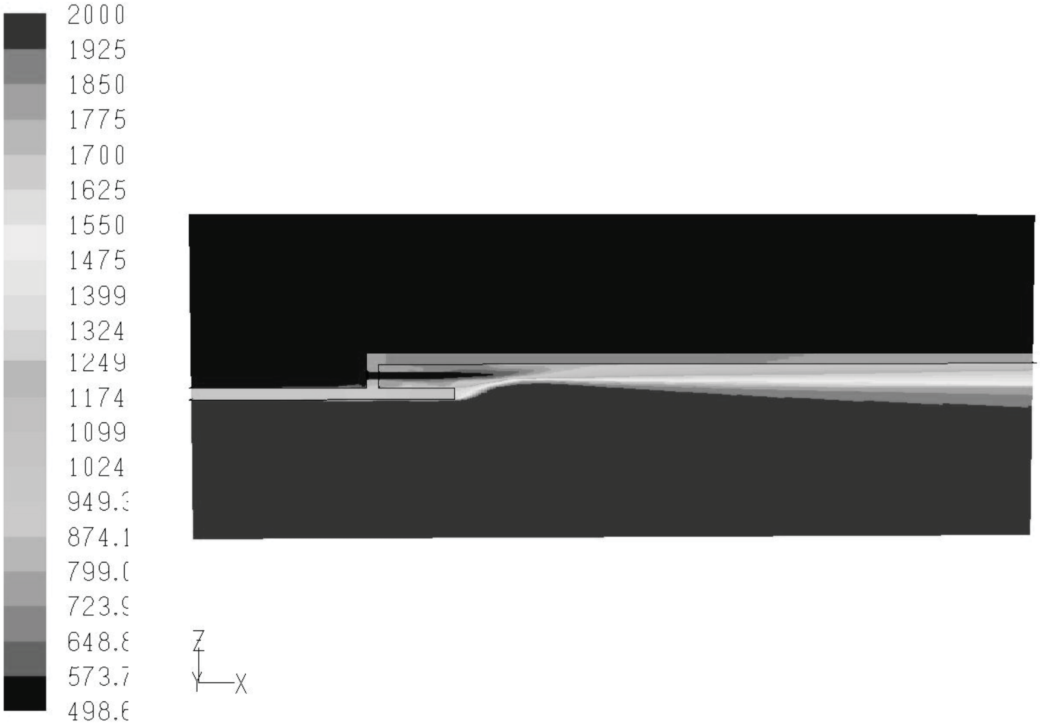

Fig. 7. Final model axial hole cross section temperature [max 2000K]

Fig. 8. Final model Axial hole cross section temperature [between 500K and 1500K]

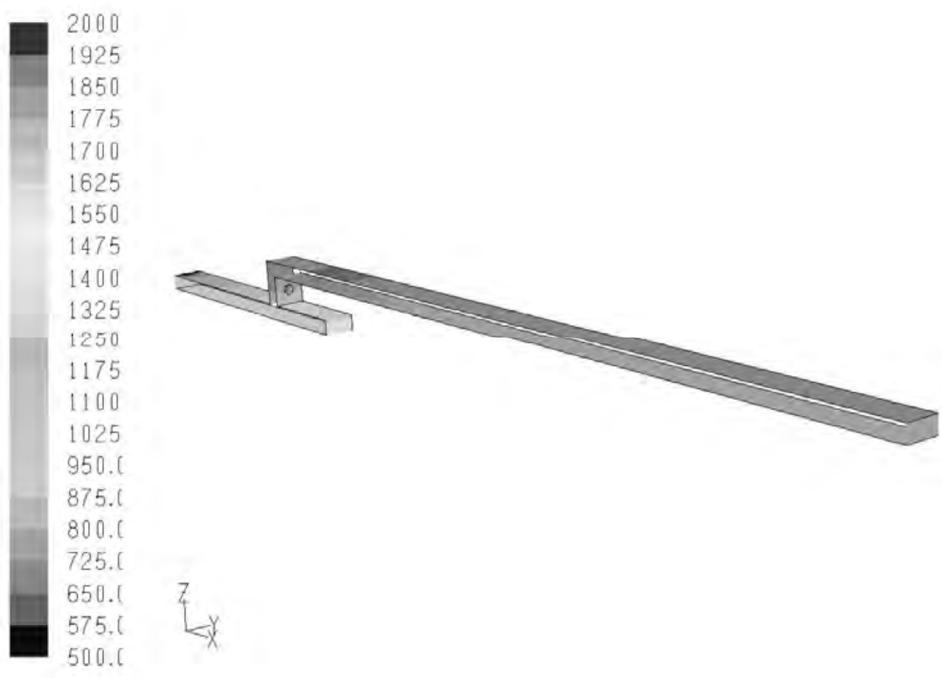

Fig. 9. Final model axial hole liner temperature [max 2000K]

4. Final model – multiholes

Final multiholes model with adjusted diameter of cooling holes (difference in total mass flow between axial hole cooling and multiple holes is 2.16% and is acceptable for comparison). This model has boundary layer and like previous model – extended domain and liner by 60% (total model length increased from 9.5in to 15.2in).

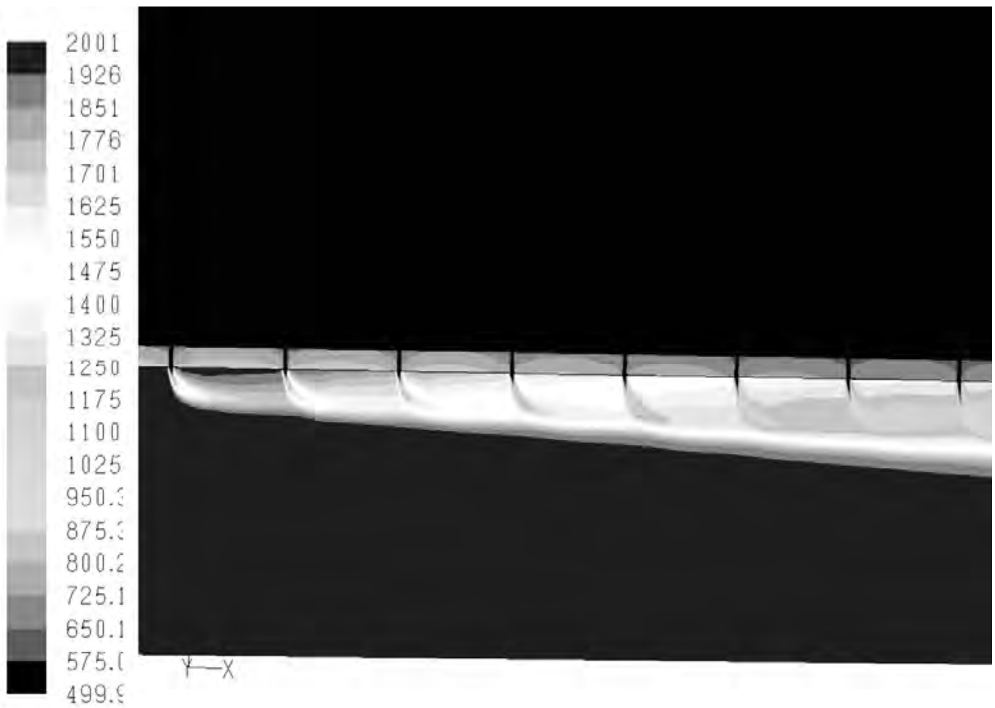

Fig. 10. Final model multiholes cross-section temperature [up to 2000K]

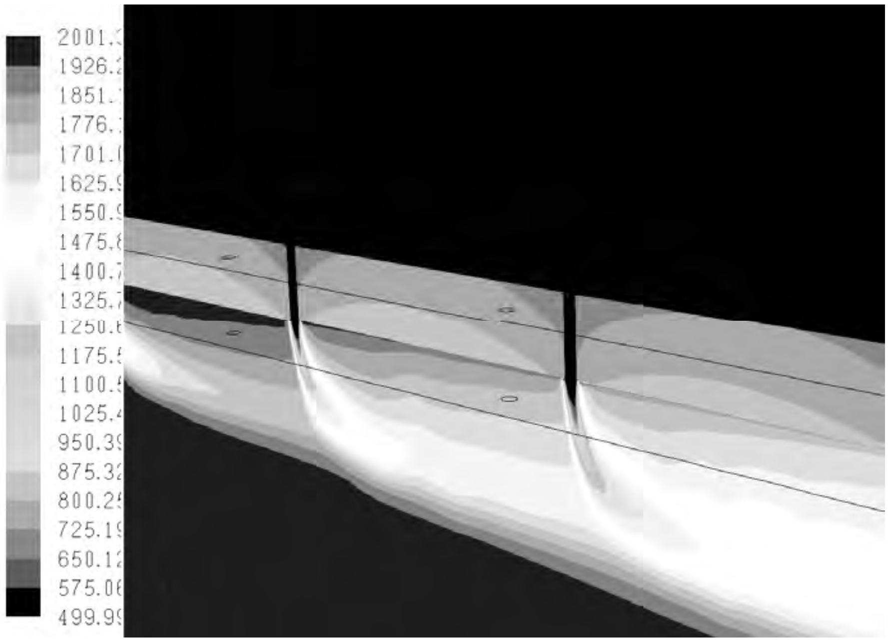

Fig. 11. Final model multiholes cross section temperature [max range, close view]

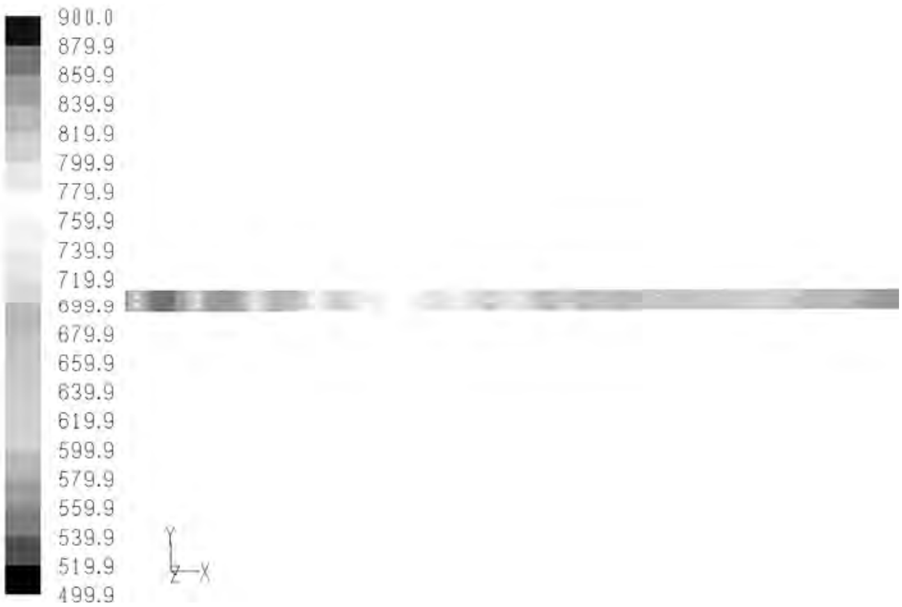

Fig. 12. Temperature along the liner wall

5. Summary

The amount of cooling air, liner material (nickel), gases on the hot and the cold side properties and boundary conditions were the same so back to back comparison of temperature profiles along the combustor liner lip and the panel was possible (Fig. 13.).

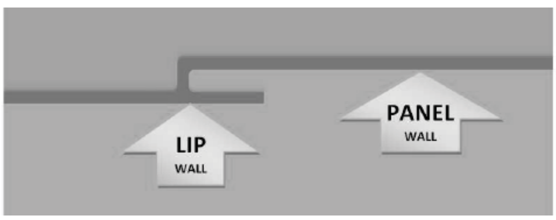

Fig. 13. Heat loaded combustor areas

Comparing temperature profiles allowed to determine the best method of cooling down the most heat loaded parts of combustion chamber – the liner walls (Fig. 15 and 17).

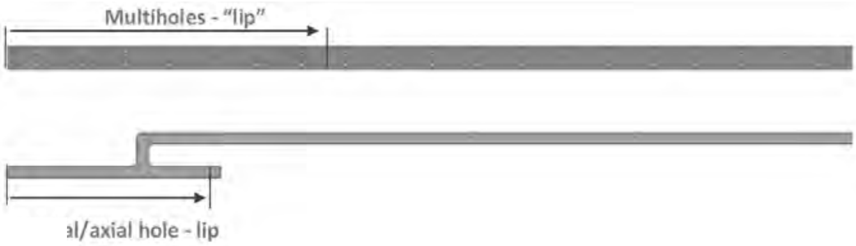

Fig. 14. Areas from which temperatures were read - liner lip (from the hot side) - all cases

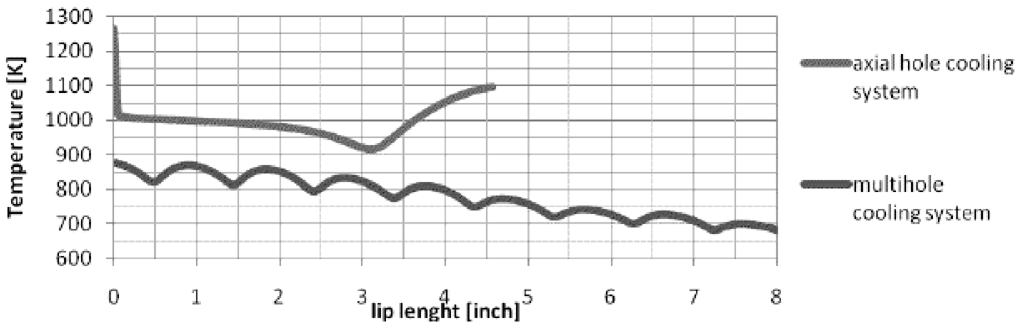

Fig. 15. Temperature plots liner lip (from the hot side) - all cases

Figure 15 presents temperature along cooled lip. Axial hole cooling system provides little protection from heat at lip end due to model simplification. In order to prevent burnthorughts as well as metal creep additional cooling should be installed. Other but also important issue is high temperature gradient on panel on short distance, which can lead to high stresses and in addition with high temperature can lead to the nugget deformation. The method that provides the lowest temperature of the lip is the multihole cooling system that has temperature at level of 690K (1). The mean temperature in this case is a much smaller than in case of axial hole cooling system. Although the temperature for multiholes is the lowest, the distribution is not even and between rows temperature gradients reaches approximately 50K.

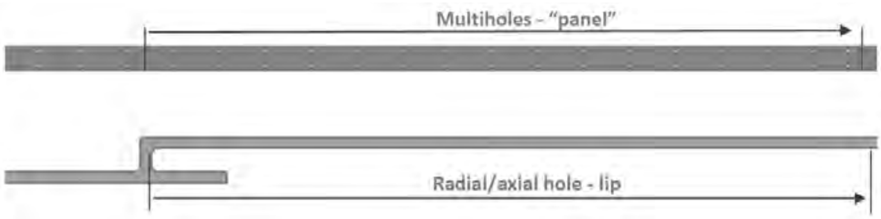

Figure 16. Areas from which temperatures were read - liner panel (from the hot side)

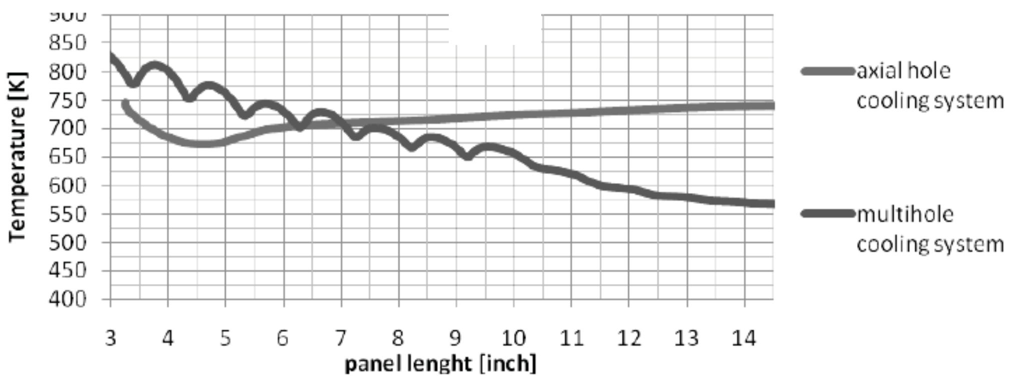

Fig. 17. Temperature plots on liner panel (from the hot side) all cases

Analysis of multihole cooling system shows the most promising results, which have significant advantage over the other cooling system (Fig. 17.). Using almost the same amount of cooling air, minimal temperature for downstream panel is (575K) (1).

It should be noted that multihole cooling system can be sensitive to holes plugging by dirt. Also the effectiveness of the latter can be increased by optimizing the number and spacing between holes. This modification can decrease temperature gradients between rows and improve cooling effectiveness cooling film. The efficiency of axial hole cooling system is significantly decreased by many geometrical simplifications, therefore results presented are only valid for this generic geometry.

References

[1] Łazarczyk, M., *Influence of geometry and type of cooling holes on thermal load of combustion chamber*, Proceedings of the Warsaw University of Technology, Faculty of Power and Aeronautical Engineering, Warsaw 2009.

[2] Łapucha, R., *Komory spalania silników turbinowo-odrzutowych,* Proceedings of Institute of Aviation, Warsaw 2005

[3] Lefebvre, A., *Gas turbine combustion*, Proceedings of Taylor & Francis, Philadelphia, PA 19106, 1988.