COMBUSTOR LINER COOLING METHODS – NUMERICAL SIMULATION

Marek Lazarczyk, 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 to evaluate advantages and disadvantages of each applied cooling methods.

It was determined that the first task was to verify how much air is coming through singe radial hole with 2.5% pressure drop between hot and cold part of combustion chamber. Flowcheck was calculated also to see how geometry of cooling hole affects hole effective area.

Second task was to generate 3d model and mesh of both calculated types of cooling. Each model mesh was covered with boundary layer in order to better simulate conditions near the combustion chamber walls and obtain accurate results. In order to run back-to-back analysis, all created models have the same number of mesh elements, same materials used, same fluent settings, same operating and boundary conditions.

Geometry of all models described above was created using Unigraphics NX4 program based on drawings obtained from available literature, and data acquired from the Internet. The discretization was done in commercial pre-processor GAMBIT®. The airflow and conjugated heat transfer analysis was calculated in program FLUENT®.

The goal of this thesis was to obtain temperature fields and distribution in the combustion chamber domain (lip and panel wall) and to evaluate if applied cooling is sufficient to cool down heat loaded part of the combustor chamber.

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

1. Introduction

The purpose this paper was to understand methods used for cooling heat loaded parts used in combustion chambers of modern aircraft turbo jet engines, getting to know the possibilities and restrictions of the professional numerical programs such as Fluent, and critical evaluation of used models and analysis results. The aim of this work is also to calculate the effect of applied cooling on combustor wall temperatures, evaluate what are the advantages and disadvantages of various cooling methods using conjugated heat transfer approach. During my work I had to prepare several CFD combined with heat transfer Fluent analyses to calculate temperature and temperature gradients on combustor liner wall and evaluate cooling film distribution. Below you can see the models of cooling used in my work. These types of cooling are commonly used in modern aircraft engines.

- Cooling slot with radial cooling hole (see Fig. 1.),
- Cooling slot with impingement cooling (see Fig. 2.).

Fig. 1. Cooling slot with radial cooling hole

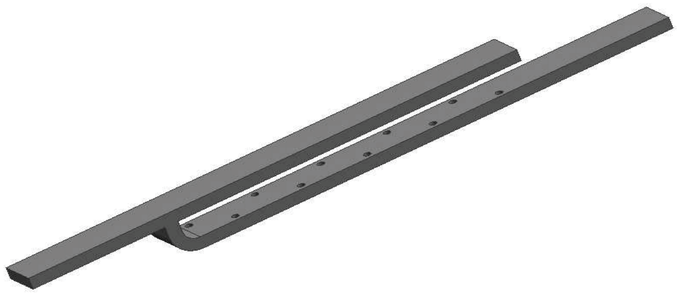

Fig. 2. Cooling slot with impingement cooling

2. Model Preparation – preliminary task

3D model of the liner with the cooling hole was prepared in program Unigrpahics®. To simplify the task (no need to model a heat transfer), only the volume of the air around the liner was prepared. Three different geometries of the cooling holes were prepared (half cone angle- 2, 4 and 6 deg).

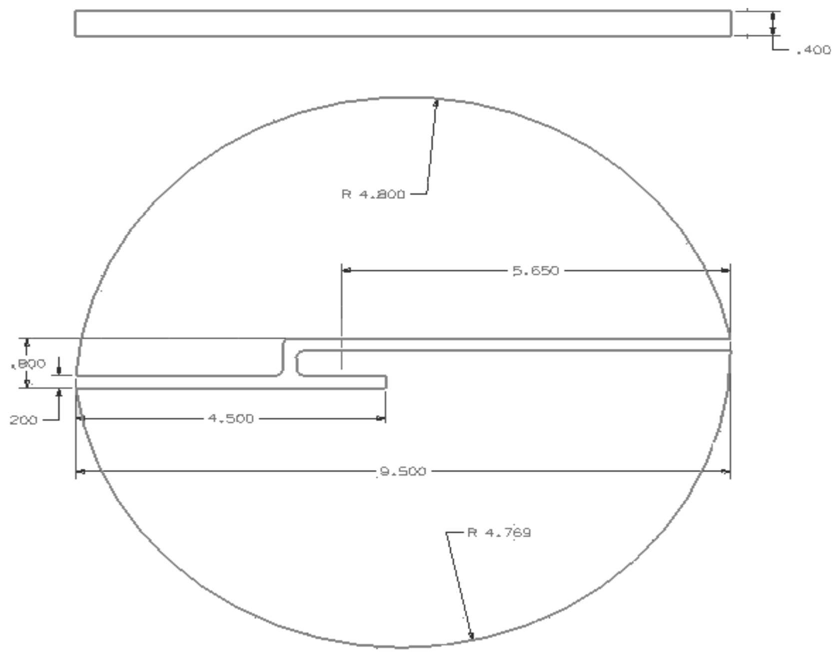

Fig. 3. Flowcheck model geometry details

Model mesh for the preliminary task was prepared in Gambit program. The mesh consisted hex elements. This model was only to calculate the discharge coefficient; heat transfer analysis was not modelled so it consists of rather big elements with no boundary layer applied. To prepare a proper mesh, model was split into the 8 volumes (see Fig. 4.).

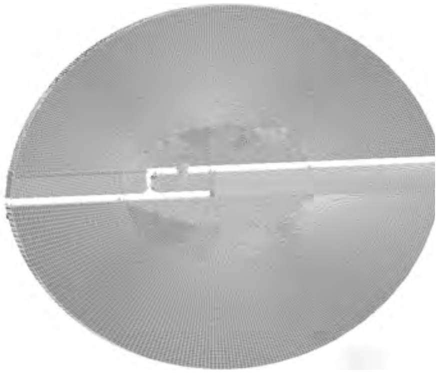

Fig. 4. Flowcheck model mesh details

Mesh consists of 803916 elements, all the elements are good quality, and almost 70% of the elements skewness is under 0.1. Max value of skew is 0.9 while the 0.1 skew are elements with the best quality and 1- the worst quality. Only 20 elements have skewness higher than 0.9 while 0.97 is the max skewness of the element accepted by the Fluent program.

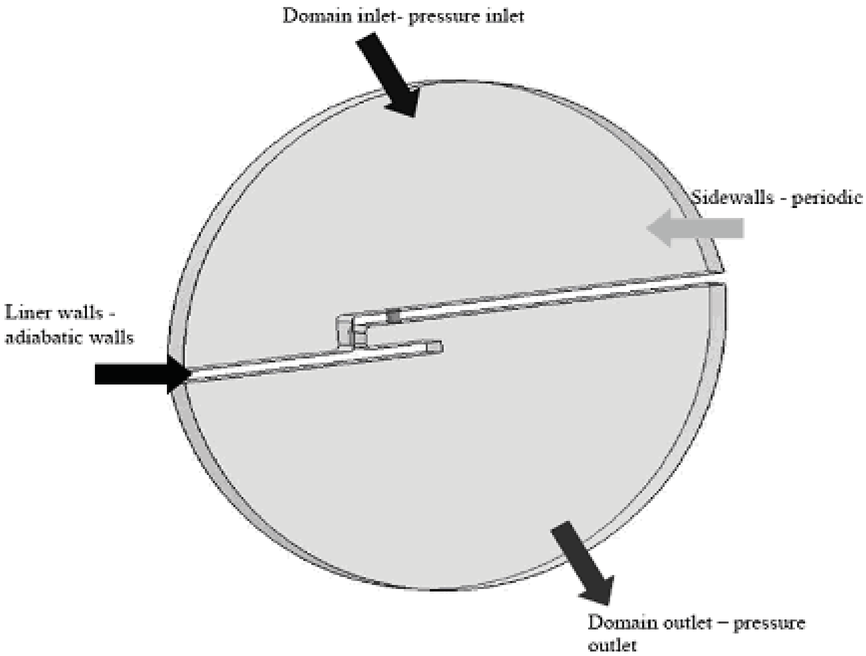

Fig. 5. Flowcheck model boundary conditions

To reflect flow check conditions, the boundary conditions were set in Fluent program as:

- Domain inlet - pressure inlet,
- Domain outlet - pressure outlet,
- Liner walls - adiabatic walls,
- Sidewalls - periodic.

All boundary conditions are summarized in Fig. 5.

Operating pressure was set as a 14.698psi - the average day atmospheric pressure (1 atm). The gauge pressure equal to 0.51443 psi was set on the domain inlet while the gauge pressure of 0 psi was set on the domain outlet. Such difference between inlet and outlet pressure is equal to 3.5% pressure drop. Temperatures of 297 K were set both on the domain inlet and outlet. Gas used in this analysis was a standard Air taken from the Fluent library database. Solver settings were set to determine the type of Analysis - 3D Steady state analysis in this case. Turbulence model was set as standard K-epsilon. Both on the domain inlet and outlet the turbulence properties, after consultation with experienced engineers, were set as follows: Turbulent intensity -5% Turbulent viscosity ratio- 250 To reflect the laser drilling process of the holes, in the further analysis the 6 deg half cone angle was used. CD parameter for this taper is equal to 0.845 for the 0.1 [in] radius hole. This value of CD parameter is reasonable and reflects the real CD obtained from the tests. The main conclusion is that the discharge coefficient parameter is highly affected by the hole taper, the CD of the hole increases as the taper increase.

3. Model Preparation

The geometry of the combustor liner used in analysis was prepared in Unigraphics®. The model simulates a combustor liner with cooling nugget with a radial hole or impingement cooling (Fig. 6.). Model was made based on extracted sketches and can be considered as a flat. Model is simplified, all blends are omitted to simplify next step – meshing. To have an opportunity to model a combined heat transfer analysis (conjugated heat transfer), both models were placed in the passage, which simulates the flow of the cold (cold passage - blue colour) and hot (hot passage- red colour) gasses on the both sides of the liner.

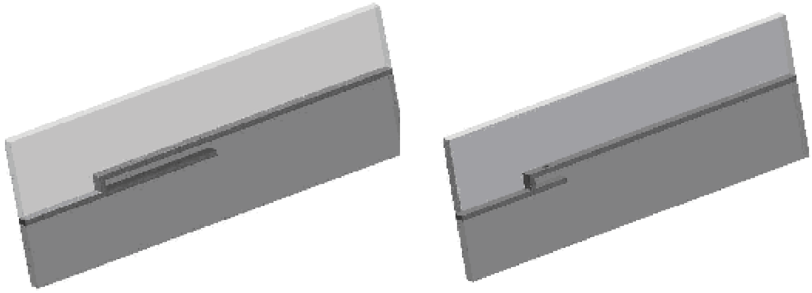

Fig. 6. Analyzed geometry models

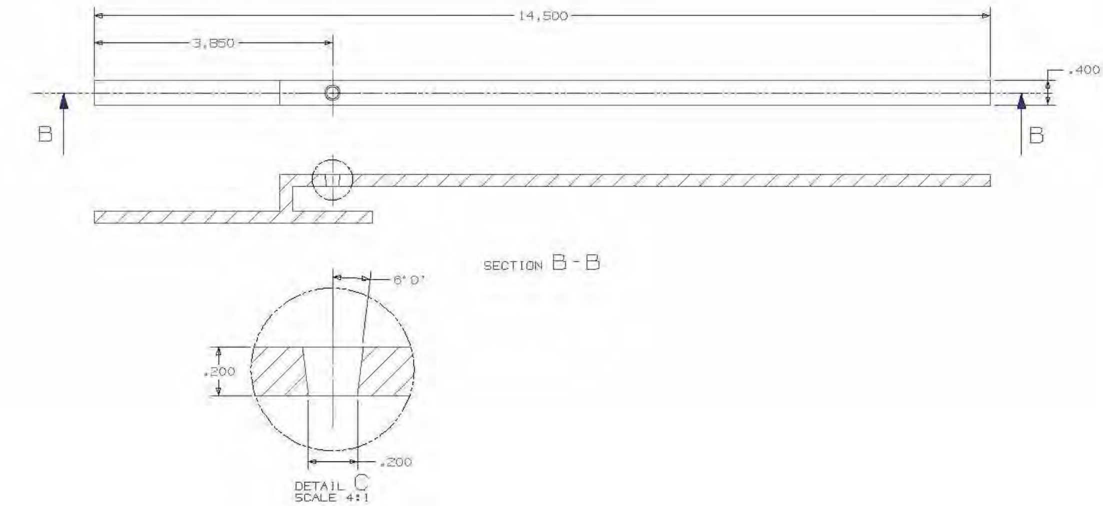

Fig. 7. Analyzed geometry - radial hole details

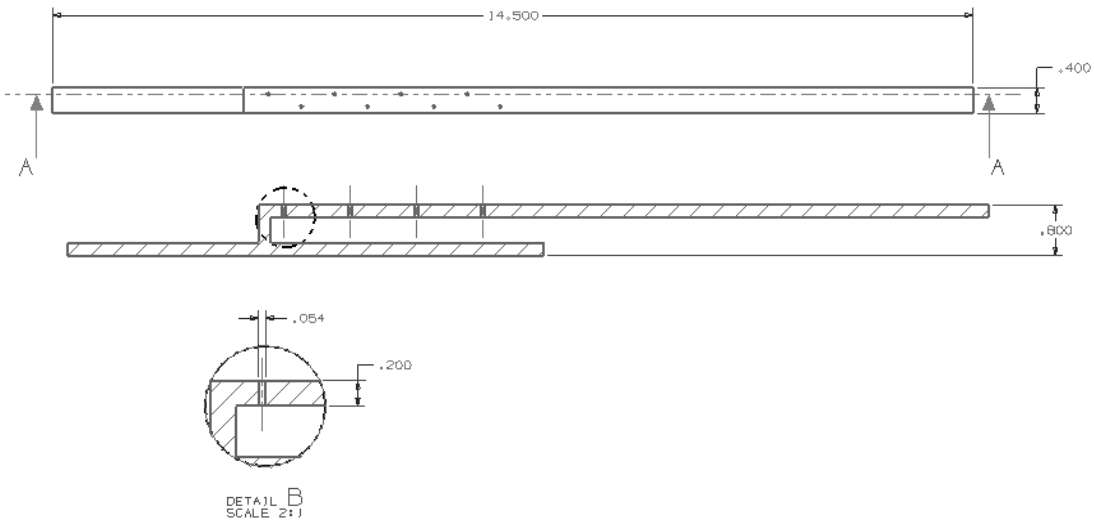

Fig. 8. Analyzed geometry - impingement details

Model was exported as a Parasolid® format what allowed importing it to program Gambit where next steps could be done. Preparing mesh (Fig. 9. and Fig. 10.) in Gambit occurred to be a hard and time-consuming task requiring lots of amendments. The most advantageous type of mesh is the mesh based on hexagonal elements, which gives the good quality mesh (low numerical

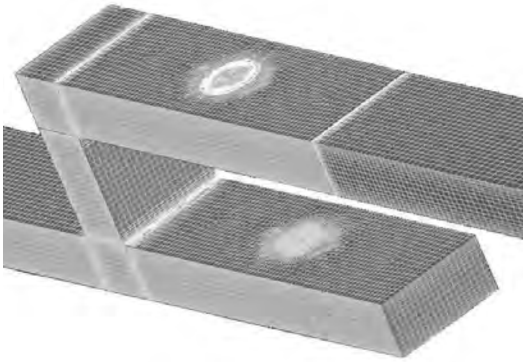

Fig. 9. Radial hole cooling system mesh

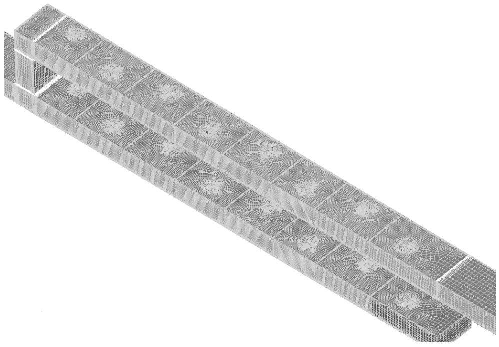

Fig. 10. Impingement cooling system mesh

diffusion) along with low number of elements in comparison to the tetragonal mesh. Relatively low number of elements reduces the required computing power and random access memory of the computer. In both cases the both metal and liquid was analyzed. For the radial hole model - liner was covered mainly by hexagonal mesh with the exception of hole and hole surrounding which was covered by hexagonal type of mesh. To make hexagonal mesh model was split into 29 pieces. To properly model the heat transfer between fluid and metal, the boundary layer was added to the metal geometry - all liner walls were covered with a boundary layer. Boundary layer consisted of 10 rows, with a growth factor of 1.2. Such parameters of BL allowed obtaining Y+ parameter in the most of the liner surfaces lower than 1. Case with radial hole mesh consist of 2690100 elements. Quality of the elements is very good, 80% of total elements skew is under 0.1, max value of skew is 0.9. The 0.1 skew are elements with the best quality and 1- the worst quality. All elements in the meshed model need to have skew under the 0.97, otherwise grid would not pass the grid check in Fluent program and may have problems with the convergence. For the impingement cooling model - liner was covered mainly by hexagonal mesh with the exception of holes surroundings which were covered by hex/wedge type of mesh. To make hexagonal mesh model was split into 63 pieces. To properly model the heat transfer between fluid and metal, the boundary layer was added to the metal geometry - all liner walls and cooling hole area were covered with a boundary layer with the same parameters as the radial hole model. Case with radial hole mesh consist of 3221334 elements. Quality of the elements is very good, 70% of total elements skew are under 0.1, max value of skew is 0.93.

Boundary conditions types were set in Gambit program.

- Both outer and inner passage inlets were set as **pressure inlets**. This type of boundary condition allows us to set a inlet pressure and temperature for the both cooling and hot gases, and the parameters that are needed to define the turbulence model -intensity and length scale,
- Both outer and inner outlets were set as **pressure outlets**. Pressure outlet boundary conditions are often used to define the static pressure at flow outlets (and also other scalar variables – for example temperature, in case of reverse flow). Using a pressure outlet boundary condition

instead of an outflow condition often results in a better rate of convergence when reverse flow occurs during iteration,

- Bottom wall was set as **symmetry** – for the symmetry boundary condition FLUENT is assuming flux of all quantities across a symmetry boundary equal to zero. There is no convective flux across a symmetry plane: the normal velocity component at the symmetry plane is equal to zero. There is also no diffusion flux across a symmetry plane: the normal gradients of all flow variables are thus zero at symmetry plane,
- Top wall was set as a **wall,**
- Sidewalls were set as rotational **periodic** walls - Periodic boundary conditions can be used when the physical geometry and the expected pattern of the flow solution have a periodically repeating nature.

The input values to calculate the boundary conditions were set as in the picture below. Rest of the parameters was calculated based on the Bernoulli equation.

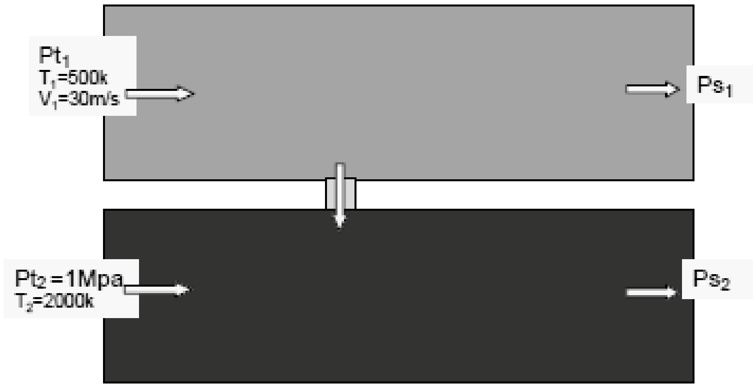

Fig. 11. Boundary conditions preliminary setting

4. Simulation Solver Settings

To reflect a combustor flow, the boundary conditions were set in Fluent program as: - Domain inlet- pressure inlet - Domain outlet – pressure outlet - Side walls – periodic - Bottom wall- symmetry The realizable k-epsilon model was used due to its superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation. This type of the k- epsilon model is usually used to predict accurately the spreading rate of planar and round jets.

Turbulence properties, after consultation with experienced engineers, were set as follows: Turbulent intensity -10% Turbulent length scale- 0.4 - this number reflects the width of the domain Enhanced wall treatment was used since the boundary layer has been used in the model. The enhanced wall treatment is designed to extend the validity of near- wall modeling beyond the viscous sublayer. To properly use the enhanced wall treatment model, the model grid must have the Y+ parameter equal to 1. The value of the Y+ parameter equal up to 5 can be accepted as long is it inside the viscous sublayer. One of the results of having Y+ parameter on such level is major grid elements increase, which results in the required CPU power and time of analysis increase.

5. Analysis description and results

Analyses were run for 10000 iterations, from which 2500 iterations were run as a first order upwind, and the 7500 iterations were run as a second order upwind. The analysis was considered as converged when the mass flow through the cooling hole/impingement cooling was stabilized. During calculations, a lot of different results were obtained. One of the calculations was the reverse flow occurrence on the pressure inlet and outlets.

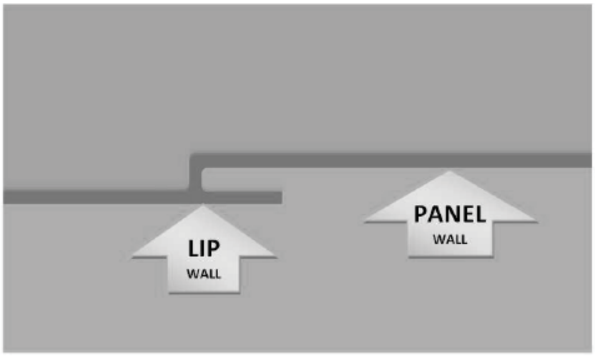

Fig. 12. Heat loaded combustor areas

The appearance of this flow caused the temperature and pressure field disorder and lead to unrealistic results. The way to eliminate the reverse flow occurrence was to increase the length of the liner and the domain, however increasing the length resulted in increase of the model mesh number of elements which increased the time and the CPU requirements to conduct the analysis. Main criterion to compare results of the cooling hole and impingement cooling, was to have the same amount of the mass flow of the cooling air used for cooling the panel in both cases. The baseline case was the case with the radial cooling hole, the impingement cooling holes geometry (diameter and number of holes) was selected to obtain the similar mass flow rate through the holes with accuracy of +/- 5%. The first approach was to model impingement holes that will have the same total area as the outlet area of the radial hole considering the CD parameter.

Below there are plots with results of both analyses: model with radial cooling hole and impingement cooling. Results of analyses, plots and charts are grouped by showed parameter, to allow comparison between both cases.

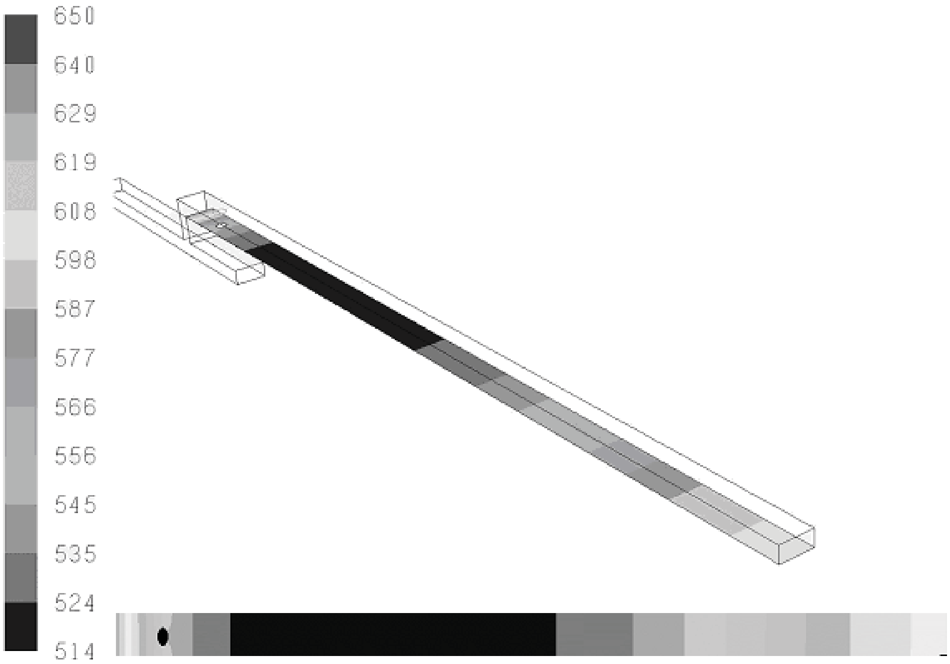

Fig. 13. Temperature layout on the liner panel – radial hole cooling

5. Summary

Running simulation often requires experience in using models to describe physical phenomena and applying a lot of simplifications, therefore results can differ from real test/field data, that is why it is highly important to validate used models, to compare results with existing test data. In

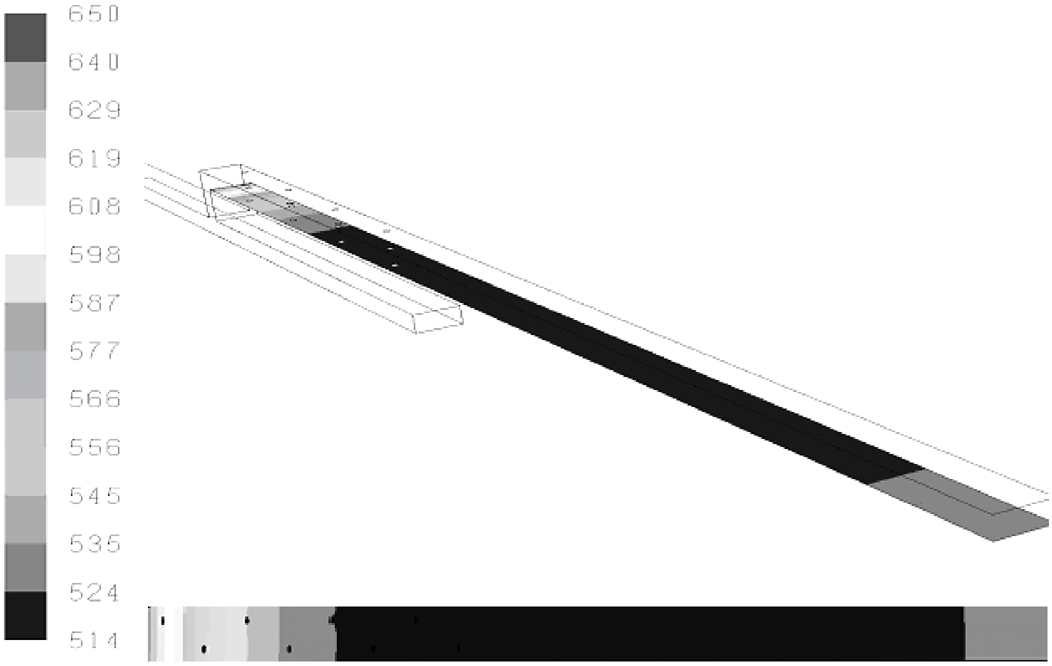

Fig. 14. Temperature layout on the liner panel – impingement cooling

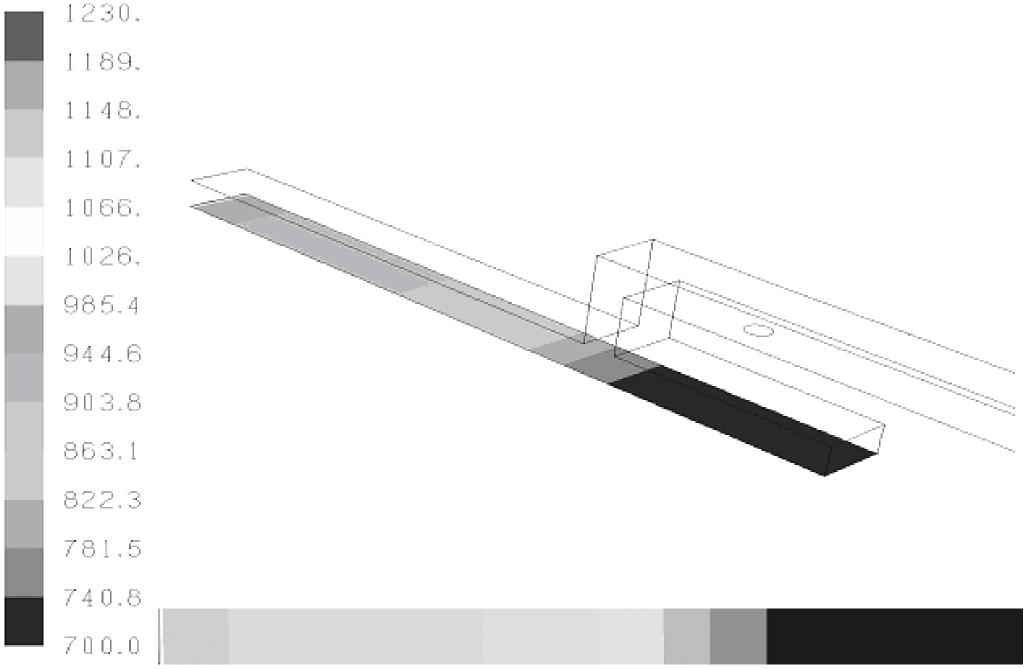

Fig. 15. Temperature layout on the liner lip – radial hole cooling

conducted work it was stated that enhanced wall treatment model was used. To properly use this model it was stated that Y+ parameter should not be higher than 5. However there are some places in my models in which Y+ parameter is between 5 and 14. These are mainly holes interiors and places where gases flowing out of holes outlets hit lip wall. Those are the places where highest velocity (up to 286 ft/s – radial hole, up to 310ft/s – impingement cooling) occur. In areas of Y+ between 5-15, applied enhanced wall treatment model can be improper, in those areas Fluent switches to standard wall function model. Decreasing Y+ parameter in those area would be combined with high grid elements increase therefore volume of the model would be to large to run analysis on home PC, and time of the analysis would be extremely increased. In conducted numerical analysis I performed several different calculations, from which only two, which gave the best, and reliable results, are described. Calculations showed in both cases that applied cooling seems to be sufficient, metal temperature does not exceed 1230 K for applied near wall hot gas temperature of 2000K. However to evaluate if this temperature of material surface (and temperature gradients) could be accepted, the stress analysis should be done (temperature is main source of combustor stress).

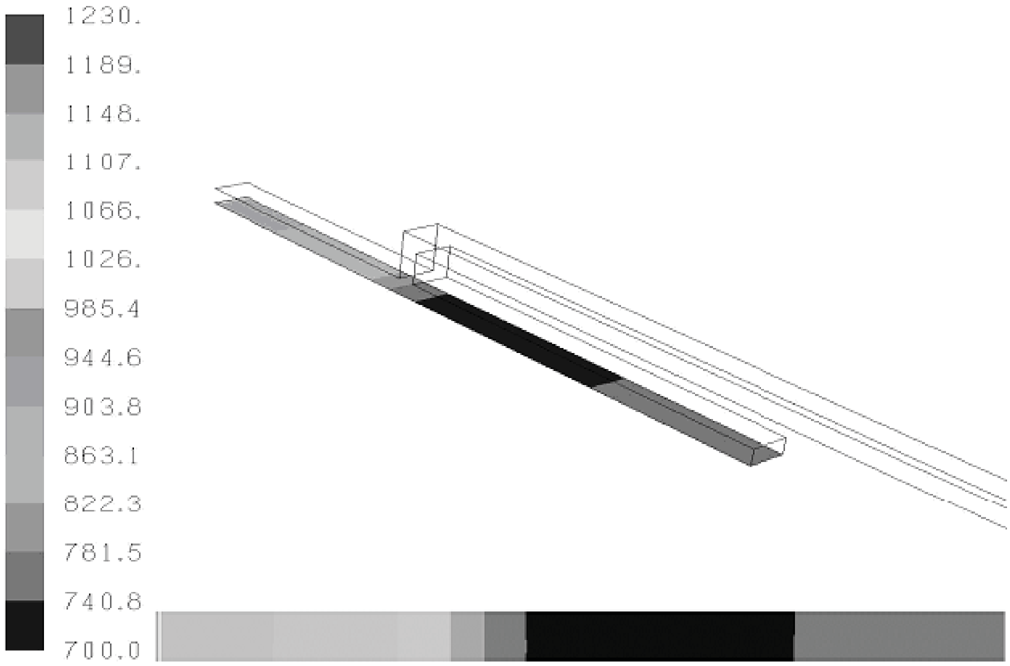

Fig. 16. Temperature layout on the liner lip impingement cooling

Most heat loaded part of the combustor liner with cooling hole nugget is end part of the lip, and downstream panel. We can see that end part of the lip is cooled better for the cooling slot with radial hole. Min temperature of the lip is cooled to 703 K while min temperature of the lip for impingement cooling is 736 K in this cross section. This is caused because radial hole has high diameter in comparison to impingement cooling hole, and all of the cooled air is concentrated in one point causing local "cold spot" while impingement cooling is more evenly distributed. Such occurrence of "cold spot (temp 680K)" surrounded by places with higher temperature (around 730K) generates temperature gradients at small area, which can lead to higher stress level. It is necessary to remember that for both cases the same amount of cooling air was used, moreover panel cooled by impingement cooling is two times longer than the one cooled by radial hole and in both cases lip temperature is relatively low, therefore it can be stated that impingement cooling is more effective to cool down lip. Fig. 3.60. shows also that in both cases there is temperature increase on the end part of the lip (up to 725K for radial cooling hole, and up to 778K for impingement cooling), however this increase of temperature is relatively low, probably because of too high amount of cooled air or to low cooling air temperature used in conducted simulations. In reality this is the most critical place, where hot spots are generated. Hot spot occurrence on the lip end can lead to metal deformation and nugget closure. The result of nugget closure is immediate downstream metal temperature increase followed by TBC spallation and burnthrough. It is necessary to remember that for those both cases same amount of cooling air was used, moreover lip cooled by impingement cooling is two times longer than the one cooled by radial hole.

For the downstream cooled panel we can see the impingement cooling advantage over nugget with radial cooling hole. With the same amount of air used, it can be observed that min temperature of downstream panel is equal in both cases (515K for impingement and 516K for radial cooling hole), however temperature distribution for impingement cooling is much more even. Panel temperatures for impingement cooling stays on the same level while panel temperature in radial cooling holes case is increasing along with the distance from radial hole. Temperature on the end of cooled panel is equal to 532K for impingement cooling and 673K for radial cooling hole. This means that temperature gradient over the panel for radial hole is higher than for impingement cooling applied, which can result in higher thermal stress. Based on panel temperature it can be stated that cooling film, which is formed in slot with impingement cooling, has better cooling properties than the film formed in nugget with radial cooling hole.

As for the summary for analyzed cases it seems that impingement cooling is a better and more efficient kind of cooling combustor liner nugget lip and downstream panel. It is also worth to know that we could apply preferential impingement cooling (use higher diameter holes at the end of the lip) to cool down hot spots that might occur on the end of the lip. Moreover temperature distribution on cooled metal for the impingement cooling is more even than for cooling with radial hole applied.

References

[1] Lefebvre, H. A., *Gas turbine combustion*, Proceedings of Taylor & Francis, pp. 275-309, Philadelphia 1999.

[2] Łapucha, R., *Komory spalania silników turbinowo-odrzutowych,* Proceedings of Institute of Aviation, pp. 153-170, Warsaw 2005.

[3] Łazarczyk, M., *Combustor liner cooling comparison using conjugated heat transfer analysis*, Proceedings of the Warsaw University of Technology Faculty of Power and Aeronautical Engineering, pp. 1-70, Warsaw 2010.