INFLUENCE OF COOLING AIR MASS FLUX VARIATION ON THE CONVECTIVE HEAT TRANSFER COEFFICIENT OF HPT ROTOR DISK

Roman Domański, Krzysztof Jedliński

Warsaw University of Technology, Division of Thermodynamics
Nowowiejska 21/25, 00-665 Warsaw, Poland
tel., fax: +48 22 8255270
e-mail: rdoma@itc.pw.edu.pl, krzysztof.jedlinski@onet.eu

Abstract

The paper is focused on the convective heat transfer phenomena on the High Pressure Turbine (HPT) rotor disk wall. The disk is a part of PW6000 turbojet engine (designed by Pratt & Whitney Company) and is a base of the 1st stage of HPT blades having very high temperature (combustion gases leaving combustion chamber have temperature around 1800 K). The analyzed disk is cooled by the air taken from one of the compressor's stages. The air gets to the gap between HPT disk and mini disk through discrete pump holes and then divides into two streams. One of them is used to decrease temperature of turbine blades and the second one is directed to the sequent turbine stages. Numerical analysis (made in Fluent) is based on the two dimensional aerodynamic model (in steady state) which is the cross section of the real geometry. There are three different sets of boundary conditions analyzed. Each of them has the same flow splits (40% of air is used to cool turbine blades and 60% gets to the next turbine stage) and all settings except cooling air mass flow on the inlet. Convective heat transfer coefficients are calculated using different method then proposed by Fluent (WFHTC and SHTC options give wrong results in this case). Coefficients are presented along disk wall and are compared with Fluent calculation methods.

Keywords: *CFD, Fluent, High Pressure Turbine, Rotor Disk, convective heat transfer coefficient*

1. Introduction

The first stage of High Pressure Turbine (HPT) is one of the most thermally loaded parts of turbine engines. It is exposed to high stresses and high temperatures. Turbine blades are subjected to the combustion gas flow which temperature is often higher then melting temperature of construction materials. This is the reason why sophisticated cooling methods are necessary (for both turbine blades and rotor disks). In high pressure turbines there is only one cooling medium available – the air subjected from compressor (transported through bypass ducts). To protect endangered parts, cooling air creates aerodynamic film on their walls or penetrates combustion gas decreasing its temperature. Metal temperature is also reduced by the air that is flowing directly on its surface (jet impingement) or that is passing through holes and channels inside it.

Aerodynamics around High Pressure Turbine rotor disk is so complex that it is not possible to determine analytically parameters such as temperature, heat flux, convection heat transfer coefficient, e.g. it is one of the reasons why Computational Fluid Dynamics (CFD) has become a contemporary designing method. Using the CFD analysis it is possible to calculate both aerodynamics of the process and heat transfer for complex geometries. It also permits to replace expensive tests on real parts (or whole turbine engines) with relatively cheap numerical models. Additionally computational methods allow to present results using visualization software.

CFD analysis consists of several steps. The first one is creation of the geometry (two or three dimensional) which reflects real part's shape. Next the proper grid should be built. Mesh elements should have size and shape which allows simulating vital aerodynamic effects. When grid is completed the boundary conditions should be set and calculation can be started. After fulfilling all convergence criteria, iterations can be stopped and the only step that is left is post processing.

2. Numerical model

The goal of this analysis is to calculate convective heat transfer coefficient on the High Pressure Turbine rotor disk wall. All results will be compared for cases with different boundary conditions – mass flow on the inlet will be changed.

2.1. Geometry of high pressure turbine rotor disk

Analysis is based on the cross-section of real geometry of High Pressure Turbine rotor disk (Fig. 1). This disk is part of the turbojet engine PW6000 designed by Pratt & Whitney Company. Rotor disk is connected by use of screws with mini disk which separates it from central part of the engine. Between the disks there is a space where the cooling air is delivered. Cooling air is brought inside through Discrete Pump Holes and later it is divided into two streams. One of them is used to decrease turbine blades' temperature and the second one gets to the sequent turbine stages (through the gap around the shaft).

On the analyzed drawing there are no contours of next stages of turbine. It is the reason why analysis has to be limited to the left rotor disk wall. There is also no information about number of holes on the circumferential of the disk so the calculation of effective areas of holes is not possible.

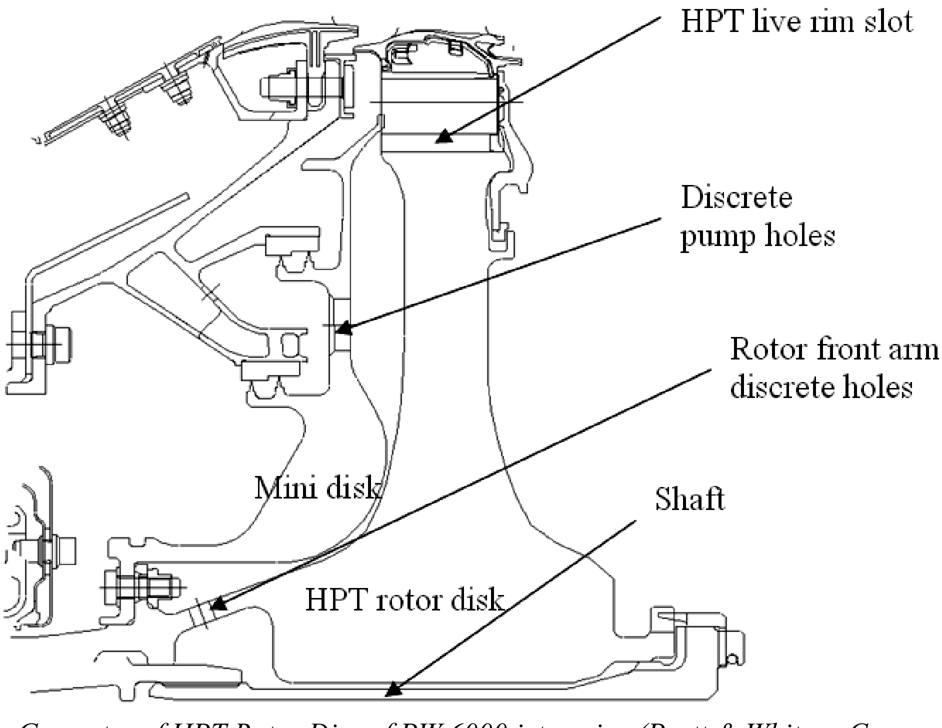

Fig. 1. Geometry of HPT Rotor Disc of PW 6000 jet engine (Pratt & Whitney Company)

2.2. Geometrical discretization

Analysis is based on simplified model of HPT rotor disk shown in Fig. 2. Wall's geometry in the space between rotor disk and mini disk is left unchanged but all elements on outer walls that don't have impact on heat transfer inside both disks were cut off. Additionally simplifying the geometry facilitates the grid creation (Fig. 3.) and decreases number of mesh elements.

The grid has two parts that are connected: the air and the metal. Air mesh has elements with regular quad shape as shown on Fig. 5. and has much higher density than metal grid. The metal is meshed with the pave scheme. Shape of the pave grid is additionally controlled by size functions which define minimum cell length, maximum cell length and growth rate.

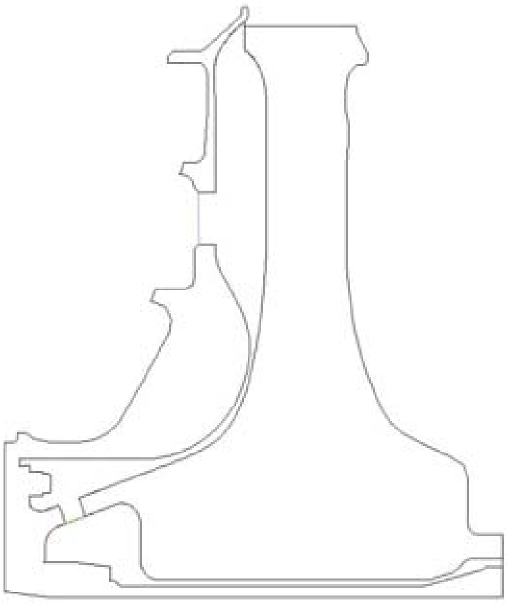

Fig. 2. Simplified model of HPT rotor disk

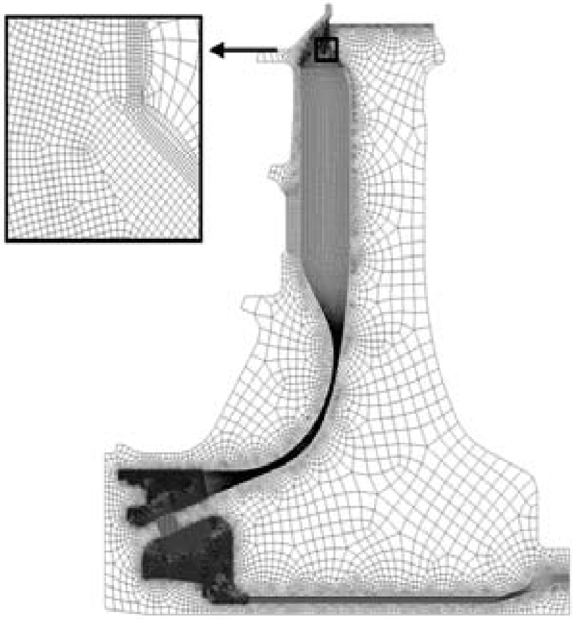

Fig. 3. Mesh used in the CFD analysis

2.3. Boundary conditions and material properties

Boundary conditions used in this model are assumed on the basis of different projects because information about operating conditions is protected by the manufacturer. Temperature of the cooling air on the inlet is assumed to be 760 K. Turbulence is described by intensity (10%) and hydraulic diameter (equal to discrete pump holes diameter). There are two outlets: 40% of the cooling air gets to the upper outlet to cool down blades of the High Pressure Turbine and 60% gets to the next turbine stages. Disk rotates with the angular speed equal to 17500 RPM. Temperature of the wall which is the base of turbine blades is equal to 1073 K. Remaining outer walls are adiabatic. Turbine rotor disk is made of steel with following parameters:

Tab. 1. Parameters of steel assumed in the CFD model

c_p	502.48 J/(kgK)
Density	8030 kg/m^3
Thermal conductivity	16.27 W/(mK)

Air is and ideal-incomessible gas with parameters presented in Tab. 2.

Tab. 2. Parameters of air assumed in the CFD model

c_p	1006.43 J/(kgK)
Thermal conductivity	0.0242 W/(mK)
Viscosity	1.7894e-05 kg/(ms)

There are three different cases analyzed. Each case has different mass flow on the inlet which is presented in Tab. 3.

Tab. 3. Mass flows on the inlet in three analyzed cases in kg/s and pps units

Case	Mass flow
1	0.145 kg/s (0.32 pps)
2	0.218 kg/s (0.48 pps)
3	0.290 kg/s (0.64 pps)

2.4. Calculation method

Numerical CFD analysis can be done using many programs. Few of them are: ADINA, ESI, NIKA, Numeca, AVL and many others. This case was calculated using Fluent. It is one of the most popular commercial systems.

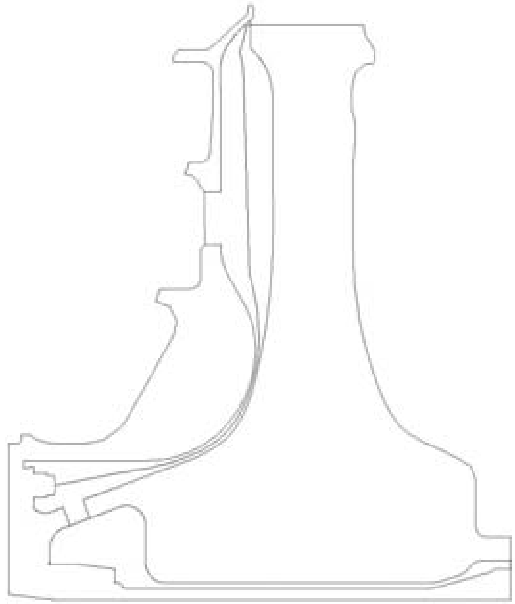

Fig. 4. Figure showing the position of reference line (green line)

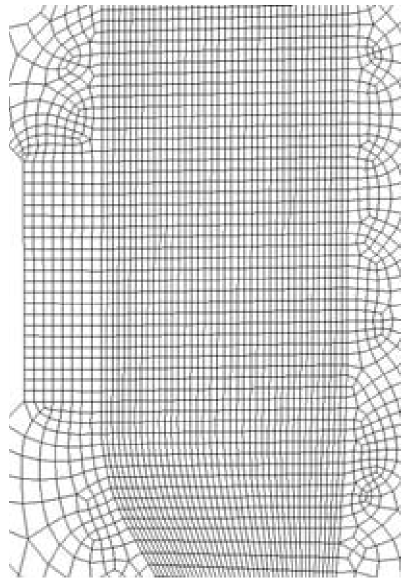

Fig. 5. Mesh close to discrete pump hole

Convection heat transfer coefficient can by determined directly by Fluent which has two different calculation models:

- Surface Heat Transfer Coefficient (SHTC).
- Wall Function Heat Transfer Coefficient (WFHTC).

Corresponding to definition of convection heat transfer coefficient, both options are wrong in case of this model. WFHTC is based on wall functions and strongly depends on grid density and gives different results for different cell wall distances in boundary layer. SHTC calculates coefficient using formula (1) where reference temperature (t_{ref}) is constant:

$$h = \frac{\dot{q}}{t_{wall} - t_{ref}}, \tag{1}$$

where:

h – Convective heat transfer coefficient [W/(m^{2}K)],

t_{wall} – Wall temperature [K],

t_{ref} – Reference temperature [K],

$\dot{q}$ – Heat flux [W/m^2].

In case of high pressure turbine rotor disk there is high radial temperature gradient and values of reference temperature along the wall can't be a constant. To avoid the error caused by averaging the reference temperature, the value of t_{ref} is calculated using the reference line. This line is placed in the middle of the air duct as shown on Fig. 4. Each cell along the wall has corresponding cell placed near the reference line which is used to read value of reference temperature in formula (1). Relative position of cells results from regular, mapped mesh shape inside the air volume (Fig. 5).

3. Convergence

To achieve sufficient convergence set of monitors were used. These monitors are:

- area weighted average, and maximum values of: analyzed wall temperature, analyzed wall total heat flux and reference line temperature,
- velocity magnitude in the chosen four points in the middle of air duct,
- mass flow balance on the inlet and outlets.

Additionally residual convergence criterion values were decreased in order not to stop iterations before the convergence is reached. Calculations in each case were stopped when all monitors and residuals were stable (after about 8,000 iterations).

4. Results

All results are presented as a function of length of the analyzed wall (the scale is shown on the Fig. 6). In the Fig. 7. there are plots of wall and reference temperatures (for the 0.145 kg/s case). The highest temperature differences are on the top of the model (over the discrete pump holes). There are also the highest gradients of temperature. In the middle part of the rotor disk both temperatures stabilizes. On the distance of 18 cm value of the reference temperature becomes very close to wall temperature. It has a great impact on heat fluxes and convection heat transfer coefficient.

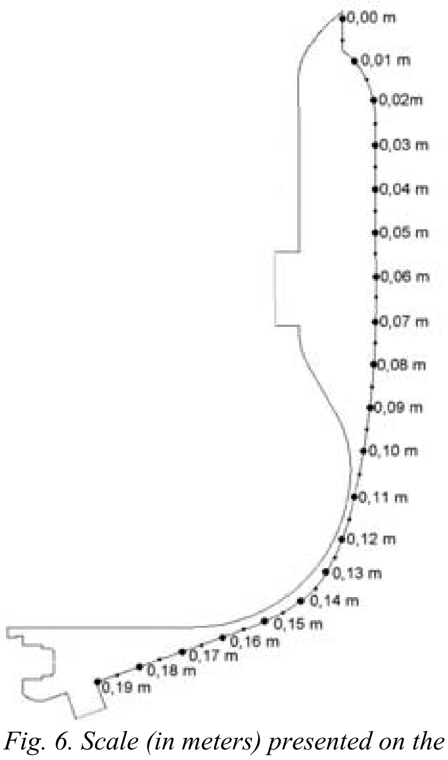

Fig. 6. Scale (in meters) presented on the analyzed rotor disk wall

In the Fig. 8. we can see the influence of boundary conditions change on the wall temperature. After increasing cooling air mass flow on the inlet, temperature of the disk decreases. Plots for all three cases are almost parallel to each other.

Heat fluxes (presented in the Fig. 9) have the highest values close to the upper cooling air outlet. Moving down to the shaft heat flux decreases and reaches small values on the end of analyzed wall. There are small heat flux oscillations on the top of the model which are caused by the turbulence and air separation near the sharply ended disk edge.

Change of the boundary conditions has significant impact on heat fluxes. It is the most noticeable over the discrete pump holes. Increasing the mass flow on the inlet causes the growth of the amount of heat taken from the disk.

Convective heat transfer coefficient values are presented in the Fig. 10. Value of this coefficient is highest near the turbine blades cooling holes outlet. The oscillations caused by turbulence and air separation are much more visible than it was on heat fluxes plot. Convective heat transfer coefficient is proportional to heat flux and opposite to temperature differences. There are two trends visible:

- Over the discrete pump holes increase of cooling air flow causes convective heat transfer coefficient growth.
- Below the discrete pump holes the increase of cooling air flow causes drop of the coefficient.

On the bottom of the disk there are very sharp oscillations. The reason of this is error caused by division by small values. The value of convective heat transfer coefficient growths sharply when the denominator in formula (1) comes to zero. Additionally when reference temperature becomes higher than wall temperature, convective heat transfer coefficient changes its sign. This solution is not physical but it is the result of computational error. This is the reason why convective heat transfer coefficient can not be calculated (using this method) on the distance grater then 16 cm.

Comparison of convective heat transfer coefficient calculated using all three formulas (for 0.64 pps mass flow inlet) is presented in Fig. 11. In the SHTC plot there is a peak on the distance of 5 cm and value of convective heat transfer coefficient changes its sign to negative. Reference temperature value is average of the gas temperature on the top and the bottom of the model (equal to 864 K). When wall temperature becomes smaller than reference temperature, denominator in formula (1) changes its sign and causes computational errors (for values close to zero).

WFHTC gives high values in comparison to other methods. It has maximum values greater than 3000 W/m^{2}K in the narrow channel between disks while there is no significant change in heat flux and temperature values.

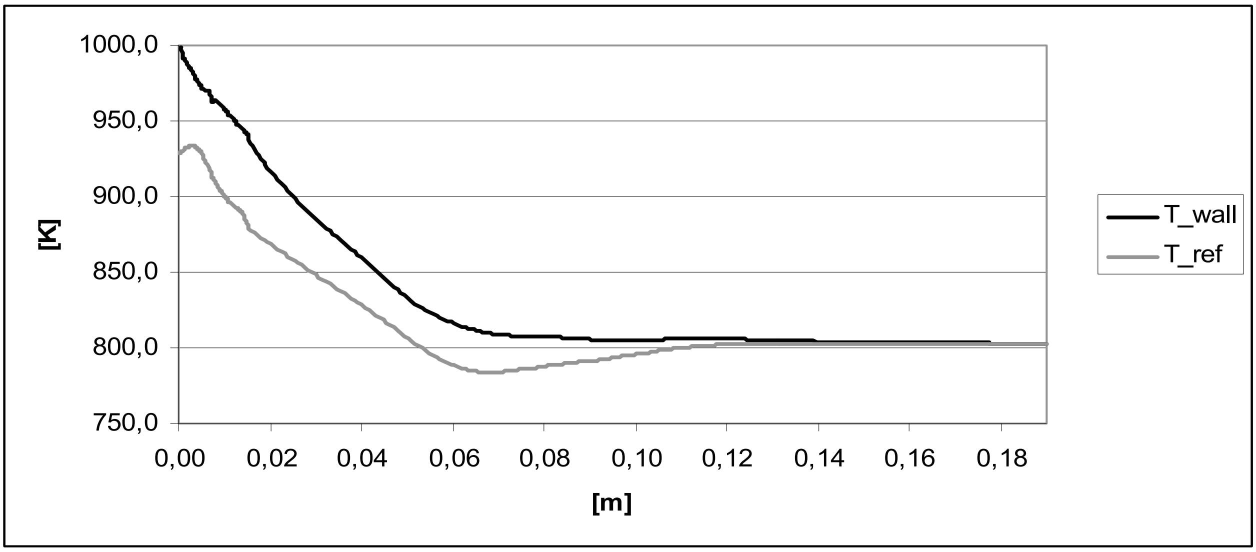

Fig. 7. Plot showing temperature of analyzed wall and reference temperature for 0.145 kg/s case

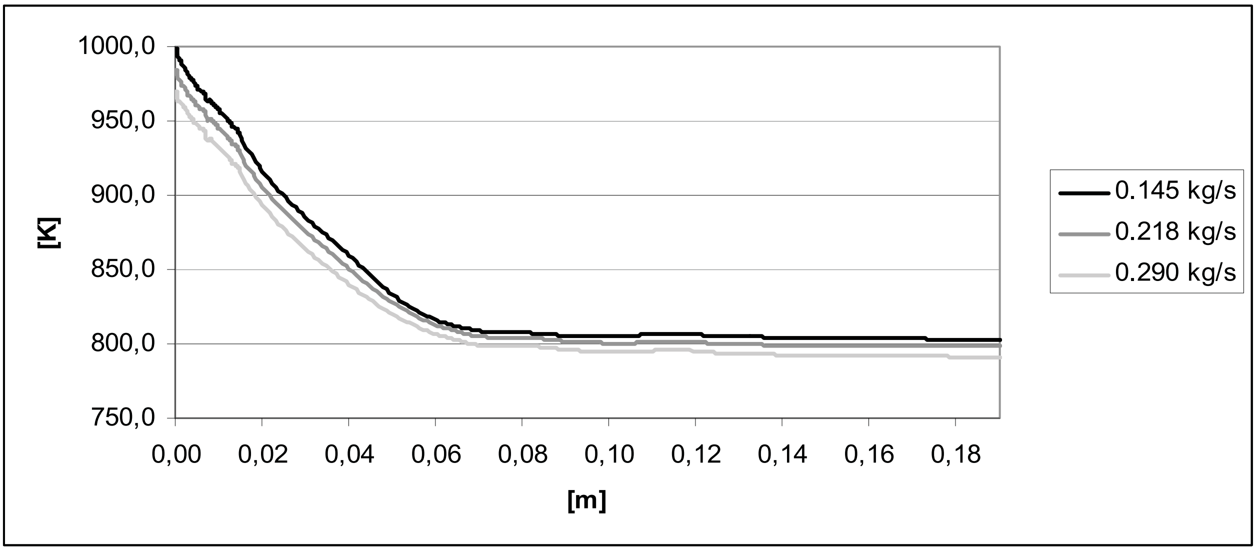

Fig. 8. Plot showing wall temperature for three different boundary conditions

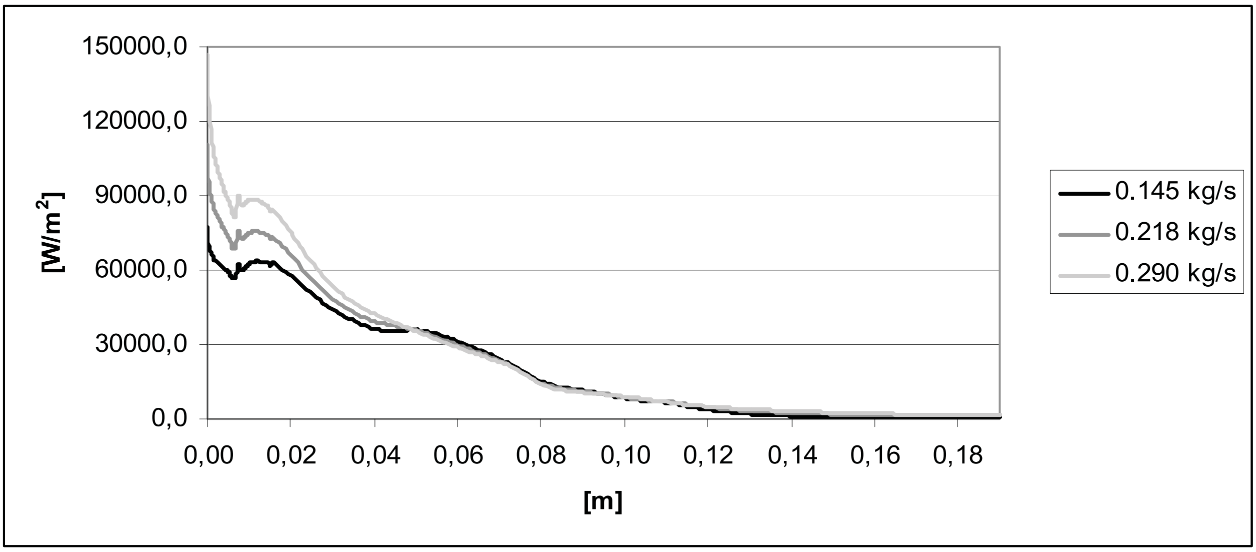

Fig. 9. Plot showing heat flux along rotor disk wall length

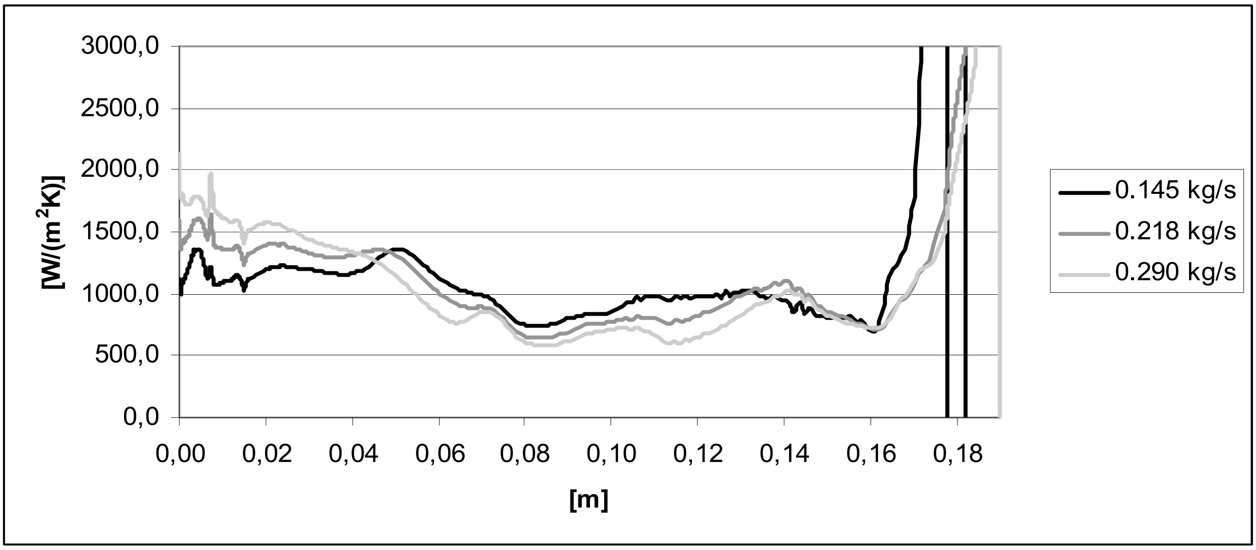

Fig. 10. Plot showing convective heat transfer coefficients along rotor disk wall

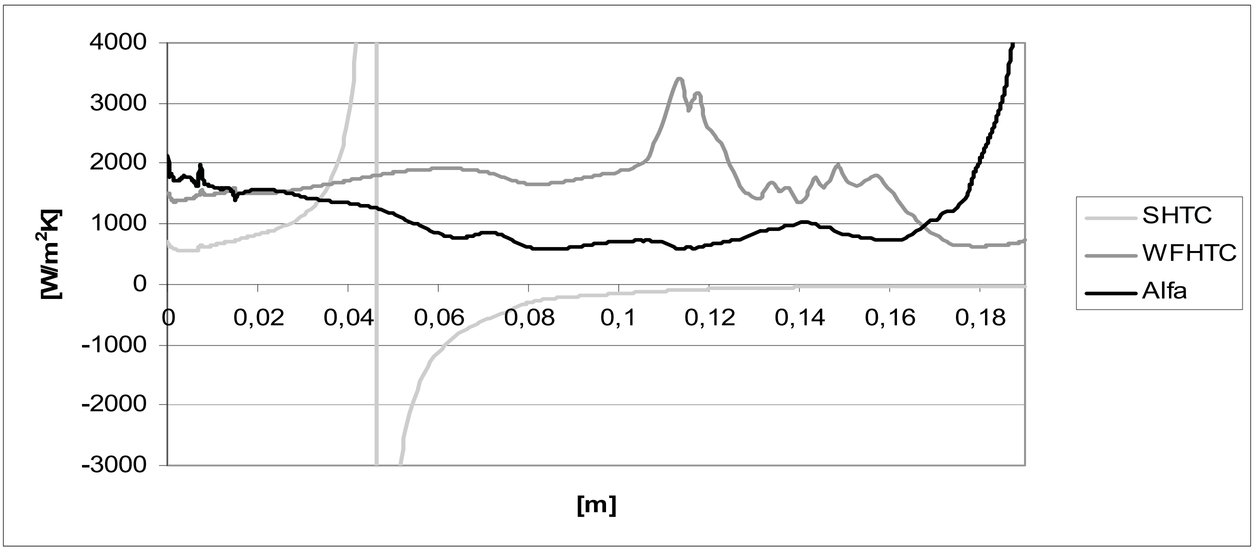

Fig. 11. Plot showing convective heat transfer coefficients along rotor disk wall calculated using three different methods (SHTC, WFHTC and method suggested in this article)

6. Conclusions

Information about convective heat transfer coefficient is crucial in thermal loads determination. Knowing inlet and outlet gas temperatures and local convective heat transfer coefficients it is possible to predict hot spots' localization on analyzed surfaces and values of their temperatures. Accurate prediction of hot spots' parameters is significant because of their impact on parts reliability.

However, there are many computational problems during heat transfer coefficient calculation using numerical methods. In the analyzed case both Fluent formulas used for heat transfer coefficient determination give wrong results. The method proposed in this analysis is closest to definition of convective heat transfer coefficient however gives high computational errors in case of small differences of temperature.

Despite all the numerical problems, CFD simulations allow calculation of thermal loads and heat transfer coefficients on the complex geometries' surfaces. Experimental measurements, in spite of their high price and low accuracy, are still very useful during numerical results validation. However their accuracy should be improved to achieve better convective heat transfer coefficient exactness (mainly in the inner parts of analyzed devices).

Nowadays in area of Propulsion it the 7^{th} EU Frame Work Program there is a series of research projects related to local convective heat transfer coefficient determination. It is the proclamation of the big interest in research of values of this coefficient. It also shows that the knowledge of local convective heat transfer coefficients is crucial for the development of contemporary turbojet engines and gas turbines.

References

[1] Boyce, M. P., *The gas turbine engineering handbook*, Gulf Publishing Company, 2006.
[2] Fluent Inc, *Documentation*, 2004.
[3] Incropera, F. P., DeWitt, D. P., *Fundamentals of Heat and Mass Transfer*, Fifth Edition, John Wiley & Sons, Inc., New York, Chichester, Weinheim, Brisbane, Toronto, Singapore 2002.
[4] Matthews, C., *Aeronautical Engineering's Data Book*, Butterworth-Neinemann, 2001.
[5] Mattingly, J. D., *Elements of Gas Propulsion*, Open University Pres, 1996.
[6] Muszyński, M., Orkisz, M., *Modelowanie turbinowych silników odrzutowych*, Institute of Aviation, Warsaw, 1997.
[7] Rolls-Royce plc, *The jet engine*, Great Britain, 1996.