© Strojniški vestnik 49(2003)9,445-457
ISSN 0039-2480
UDK 621.311.238:536.23
Izvirni znanstveni članek (1.01)
© Journal of Mechanical Engineering 49(2003)9,445-457
ISSN 0039-2480
UDC 621.311.238:536.23
Original scientific paper (1.01)

Konvektivni prenos toplote v vrtečem se kanalu kaskade z ravnimi lopaticami

Convective Heat Transfer Inside Rotational Cascades with Flat Blades

Mitja Mori - Lovrenc Novak - Mihael Sekavčnik

V prispevku je obravnavana odvisnost konvektivnega prenosa toplote od vrtilne frekvence na vroči steni turbinske lopatice. Prenos toplote poteka v križnem sistemu iz aksialnega toka vročih dimnih plinov na hladilni zrak v kanalu lopatice. Z numeričnim modelom smo simulirali tokovne in energijske razmere v aksialni krožni kaskadi z ravnimi lopaticami pri različnih vrtilnih frekvencah in količini hladilnega zraka, rezultate pa smo zapisali v brezdimenzijski obliki. Izkazalo se je, da ima v obravnavanem primeru razmernik hladilnega zraka bistveno večji vpliv od rotacijskega števila. Rezultate numeričnih simulacij smo primerjali z rezultati enodimenzijskih, polizkustvenih modelov konvektivnega prenosa toplote z različnimi nastavki, ki so v praksi najpogosteje uporabljeni. V primerjavi z rezultati numeričnih analiz vsi trije uporabljeni modeli kažejo pravilno usmeritev, naboljše ujemanje pa dobimo z uporabo modela prenosa toplote skozi ravno steno.

(Ključne besede: turbine plinske, hlajenje konvektivno, števila Nusselt, analize numerične)

In this paper we present a numerical approach to determining the variation of rotational speed with the convective heat transfer of blade structures. The investigated structure, which is cooled by air, was subjected to an axial hot-air stream in a cross-flow system. In order to exclude the influence of undesirable, aerodynamic flow forces, a simplified geometry was chosen, and the influence of varying the rotational speed was studied. The distributions of heat-transfer rate on the hot side of the structure's wall were determined with CFD calculations. The non-uniformity of the heat transfer increased with the rotational speed due to the increasing impact of centrifugal flow forces on the boundary-layer thickness. The impact of rotational speed on the heat-transfer rate was found not to be significant. The results of the studied structure were also used to validate various one-dimensional, empirical, convective heat-transfer models that are commonly applied in engineering practice. In all cases, the flat-wall heat-transfer treatment showed the best agreement with the results of the CFD calculations.

(Keywords: gas turbines, convective cooling, Nusselt number, numerical analysis)

0 UVOD

Iz osnovnih zakonov termodinamike izhaja, da lahko izboljšamo izkoristek poljubnega krožnega procesa s povečevanjem srednje temperature dovoda toplote. Posledica tega je težnja po višji temperaturi delovnega sredstva pred začetkom ekspanzije. V primeru Joulovega krožnega procesa, kot primerjalnega procesa v plinski turbini, je prva turbinska stopnja najbolj toplotno obremenjena, zato morajo biti posamezni deli konstrukcije, kakor so lopatice vodilnika in

0 INTRODUCTION

The laws of thermodynamics state that the heat-to-power conversion efficiency of any cycle can be improved by increasing the mean temperature of the heating process, which leads to a higher temperature of the working fluid before the expansion process. Thus, taking into consideration the Joule-cycle, which takes place in a gas-turbine plant, the first turbine stage is exposed to extreme thermal loadings. In order to obtain the high temperature of the input gas and the sufficient strength and metallurgical stability of the turbine structure, gonilnika, hlajene. Najpogosteje uporabljena tehnika hlajenja je konvektivno hlajenje lopatic, pri katerem je del zraka iz kompresorja uporabljen kot hladivo, ki teče v kanalu znotraj lopatice. Z namenom, da bi zadostili omejitvam zaradi toplotnih in mehanskih obremenitev gradiva lopatic, potekajo raziskave v smeri zmanjševanja prestopa toplote z dimnih plinov na lopatico in izboljševanja prestopa toplote z lopatice na hladivo. Poleg tega mora biti masno razmerje med hladivom in plini čim manjše, saj se zaradi nepovračljivosti pri mešanju hladiva s plinom znižuje termodinamični izkoristek procesa.

V zadnjih desetletjih je bilo razvitih veliko metod za identifikacijo vplivnih parametrov in odvisnosti pri prenosu toplote v plinskih turbinah z eksperimentalnimi in numeričnimi raziskavami. Celovit pregled rezultatov omenjenih raziskav je podan v [1] in [2]. Rezultati obsežnega eksperimentalnega raziskovanja omogočajo boljše razumevanje specifičnih tokovnih in energijskih pojavov za značilne obratovalne okoliščine, vendar ne omogočajo splošne in natančne kvantitativne ocene prenosa toplote. V večini primerov so eksperimentalno dobljeni podatki rabili za oblikovanje enodimenzijskih polizkustevnih modelov z omejenim območjem uporabe.

V toplotnih turbinskih strojih, kjer so posamezni elementi (npr. stator in rotor) v relativnem gibanju, so eksperimentalna opazovanja tokovnih in energijskih razmer zelo omejena, pogostokrat celo nemogoča. Študij posameznih pojavov se opravlja na prilagojenih preizkusih, katerih rezultati imajo zelo omejeno območje veljavnosti [3], čedalje večjo uporabnost pa dobiva uporaba numeričnih modelov na ločeno popisanih geometrijskih območjih.

V prispevku je obravnavan stacionarni prenos toplote v aksialnem kanalu plinske turbine z vročih dimnih plinov na steno lopatice z uporabo numerične analize. Preučevan je bil vpliv spreminjanja vrtilne frekvence na prenos toplote. Da bi izvzeli dodatne vplive pospeškov in površinske ukrivljenosti na strižne sile v mejni plasti [4], je bil uporabljen kanal z ravnimi, vzporednimi stenami. Uporabljeni so bili stacionarni robni pogoji z namenom izvzeti vpliv turbulence zaradi medsebojnega vpliva rotorja in statorja. Na podlagi dobljenih rezultatov je narejena primerjava z rezultati polizkustvenega enodimenzijskega modela s tremi različnimi nastavki.

1 TEORETIČNE OSNOVE

Obstaja nekaj procesov za oceno mehanizma prenosa toplote na konvektivno hlajeni lopatici plinske turbine. Numerični proces zahteva rešitev vodilnih tokovnih enačb. Natančnost te metode je odvisna od numerične sheme, modeliranja turbulence

it is necessary to cool the stator and rotor blades of the gas turbine. One of the most commonly used cooling techniques for rotor blades is convective cooling, which is performed using the air extracted from the compressor as a coolant. To satisfy the metallurgical temperature limit of the blade structure, design solutions that provide good convective heat transfer on the cold side and low convective heat transfer on the hot side of the blade structure are required. It is also clear that the mass-flow ratio between the coolant and the gas has to remain as low as possible, due to mixing irreversibilities and the consequent efficiency drop of the cycle after the coolant is injected into the gas stream. Thus, the optimum solution is a compromise between several parameters that affect heat transfer.

In past decades, many approaches to heat-transfer parameter identification and correlation modelling in turbomachinery were developed, both experimentally and numerically. A comprehensive overview of this area was provided by [1] and [2]. Although the large amount of experimental data obtained has improved our understanding of fluid dynamics and heat-flow phenomena in gas turbines, no single technique has been developed, as yet, that would provide generally reliable results. Thus, many correlations have been derived from experimental data that contribute useful guidelines for engineering practice. They are mostly based on a one-dimensional heat-transfer treatment, which yields only the first approximation of the temperature distribution.

In turbomachinery, where the fluid flow and the heat-transfer phenomena occur in both the stationary and rotating passages (nozzle and rotor), experimental observation is often quite limited [3]. For this reason, CFD simulations, which yield a numerically obtained solution to the governing Navier-Stokes equations in terms of the different turbulent models and boundary conditions, become very great important.

In this paper, the stationary heat transfer from the hot gases to the blade surface in an axial cascade is studied by means of numerical analyses. Rotational speed, as the variable parameter, was investigated. In order to exclude any additional influences affecting the shear stresses in the boundary layer caused by the passage curvature [4], a cascade with flat walls was chosen. An additional assumption, that of stationary boundary conditions, was taken into account, in order to avoid turbulence due to the stator-to-rotor interaction. The results obtained were compared to those obtained with three different semi-empirical one-dimensional correlations.

1 THEORETICAL BACKGROUND

There are several approaches to predicting the heat-transfer performance of convectively cooled blades. The numerical approach requires the solving of the equations governing fluid flow. The accuracy of this method depends on the numerical technique, the in prehoda na lopatici, modela prenosa toplote in kakovosti mreže.

Vodilne enačbe za obravnavani tok tekočine so stacionarne Navier-Stokesove enačbe v ohranitveni obliki. Enačbe za ohranitev mase, gibalne količine in energije za rotirajoči sistem lahko zapišemo kot [7]:

turbulence and transition modelling, the heat-flux model, the grid resolution and the effect of artificial dissipation.

The equations governing fluid flow in the presented case are steady, Navier-Stokes equations in a conservation form. The instantaneous equations for mass, momentum and energy conservation for rotational systems can be written as follows [7]:

$$\nabla\left(\rho w\right)=0 \quad (1),$$

$$\nabla\left(\rho w\times w\right)=\nabla\left(-p\delta+\mu\left(\nabla w+\left(\nabla w\right)^{T}\right)\right)-\rho\omega\times\left(\omega\times r\right)-2\rho\omega\times w \quad (2),$$

$$\nabla\left(\rho w h_{\mathrm{tot}}\right)=\nabla\left(\lambda\nabla T\right)+S_{E} \quad (3).$$

Navier Stokesove enačbe (1), (2) in (3) popisujejo tako laminarne kakor turbulentne tokove. Neposredna numerična rešitev teh enačb (diskretiziranih) za turbulentne tokove, poznana kot direktna numerična simulacija (DNS), zahteva ogromne računalniške zmogljivosti že za razmeroma preproste geometrijske primere, zato jih v bližnji prihodnosti v inženirskih uporabah ne gre pričakovati. Alternativni proces k DNS je sprememba Navier-Stokesovih enačb z uvedbo povprečn in spreminjajočih se komponent v obliki Reynoldsovih povprečenih Navier-Stokesovih enačb (RPNS). Da sistem vodilnih enačb postane rešljiv, morajo biti neznane spreminjajoče se komponente v RPNS modelirane z dodatnimi enačbami, tako imenovanimi turbulenčnimi modeli. Najpogosteje uporabljena skupina turbulenčnih modelov je model vrtinčne viskoznosti, ki omogoča dobro usklajenost med potrebnimi računalniškimi zmogljivostmi, robustnostjo in natančnostjo izračuna. Alternativa k modelu vrtinčne viskoznosti so modeli navora, ki pa pomenijo dodatno breme v numerični shemi in zato zahtevajo ogromne računalniške zmogljivosti [5].

Neznana člena v RPNS, ki se rešujeta s turbulenčnimi modeli, sta člen Reynoldsovih napetosti in člen turbulentnega toplotnega toka. Modeli vrtinčne viskoznosti povežejo ta dva člena s povprečenimi komponentami spremenljivk prek skalarne veličine, imenovane turbulentna viskoznost. Turbulentna viskoznost je nadalje podana kot zmnožek karakteristične hitrosti in dolžinske skale turbulence, ti dve veličini pa sta običajno modelirani z dvema enačbama, zato v tem primeru govorimo o dvoenačbnih turbulenčnih modelih. Karakteristična hitrost turbulence je računana iz turbulentne kinetične energije (k), za katero je predpisana prva transportna enačba. Dolžinska skala turbulence je ocenjena iz dveh lastnosti, običajno iz turbulentne kinetične energije in njenega raztrosa, ki je določena z drugo transportno enačbo. Najbolj razširjen dvoenačbni turbulenčni model je model $k-\varepsilon$ ((4) in (5)), kjer ε pomeni disipacijsko hitrost turbulentne kinetične energije, pogosto uporabljan pa je tudi model $k-\omega$ ((6) in (7)), kjer ω pomeni frekvenco turbulence ([7] in [12]).

The Navier-Stokes equations (1), (2) and (3) describe both laminar and turbulent flows. A direct numerical solution of these equations (discretized) for turbulent flows, also known as a direct numerical simulation (DNS), would require enormous computing power, which is not foreseeable for engineering applications in the near future. A common alternative approach to DNS is to modify the Navier-Stokes equations by introducing averaged and fluctuating components to produce Reynolds Averaged Navier-Stokes (RANS) equations. In order to achieve the closure of the system of governing equations, the unknown fluctuating terms in RANS have to be modelled by additional equations called turbulence models. The most widely used group of turbulence models is that of eddy-viscosity models, since they offer a good compromise between computing-power requirements, robustness and the accuracy of the solution. An alternative to eddy-viscosity models are second-moment closure models. However, they impose a large burden on the numerical scheme and therefore require significant computer resources [5].

The unknown terms in RANS, solved by turbulence models, are the Reynolds stress term and the turbulent heat-flux term. Eddy-viscosity models relate these two terms with averaged variables through another scalar variable, called the turbulent viscosity. Turbulent viscosity is defined as the product of the turbulent velocity and the turbulent length scale, which are modelled by two separate transport equations, hence the term two-equation models. The turbulent velocity is computed from the turbulence kinetic energy (k), which is defined by the first transport equation. The turbulent length scale is estimated from two properties of the turbulence field, usually the turbulence kinetic energy and its dissipation rate, which is defined by the second transport equation. Two of the most widely known two-equation turbulence models are the k–ε model ((4) and (5)), where ε represents the turbulence eddy dissipation, and the k–ω model ((6) and (7)), where ω represents the turbulent frequency ([7] and [12]).

$$\frac{\partial(\rho k)}{\partial t}+\nabla(\rho w k)=\nabla\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\nabla k\right]+P_k-\rho\varepsilon \quad (4),$$

$$\frac{\partial(\rho \varepsilon)}{\partial t}+\nabla(\rho w \varepsilon)=\nabla\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\nabla \varepsilon\right]+\frac{\varepsilon}{k}\left(C_{\varepsilon 1}P_k-C_{\varepsilon 2}\rho\varepsilon\right) \quad (5),$$

$$\frac{\partial(\rho k)}{\partial t}+\nabla(\rho w k)=\nabla\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\nabla k\right]+P_k-\beta'\rho k\omega \quad (6),$$

$$\frac{\partial(\rho \omega)}{\partial t}+\nabla(\rho w \omega)=\nabla\left[\left(\mu+\frac{\mu_t}{\sigma_\omega}\right)\nabla \omega\right]+\alpha\frac{\omega}{k}P_k-\beta\rho\omega^2 \quad (7).$$

$C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k, σ_ε, β', β, α, σ_ω so konstante in P_k porajanje turbulence zaradi viskoznih in vzgonskih sil.

Če uporabljamo CFD orodja za analizo prenosa toplote v vrtečih se kanalih, moramo vzeti v zakup, da imajo na strukturo turbulentnega polja velik vpliv ukrivljenost površine, vrtenje, prehodni pojavi, odlepljanje itn. V [1] je pokazano, da prvi polizkustveni turbulenčni modeli, npr. $k-\varepsilon$, ne popišejo dobro viskozne in temperaturne mejne plasti, kar ima za posledico precejšnje napake v oceni prenosa toplote. V primerjavi z drugimi klasičnimi modeli vrtinčne viskoznosti, omogoča turbulenčni model strižne napetosti (TSN - SST) v kombinaciji z ustrezno obravnavo toka ob steni za zdaj najboljši kompromis med natančnostjo izračuna in potrebnimi računalniškimi zmogljivostmi ([5] in [6]).

V našem primeru je bila numerična simulacija (CFD) narejena s programskim paketom CFX-5.5.1, v katerem je model TSN oblikovan kot kombinacija klasičnega dvoenačbnega modela $k-\omega$ in $k-\varepsilon$. ω – enačba, ki omogoča boljšo oceno strižnih napetosti ob steni, je uporabljena za določanje toka ob steni in enačba ε – za določanje prostega toka. Izbiranje med obema modeloma je izvedeno s funkcijami, ki upoštevajo razdaljo do najbližje stene in tokovne spremenljivke [7]. Dodatno vsebuje ω – enačba zelo preprost izraz za nizek-Re, kar je zajeto v 'izbiri stenskih funkcij' [7], ki samodejno izbira med stenskimi funkcijami in nizkim-Re modelom. Stenske funkcije predpisujejo tok ob steni z izkustvenim funkcijami, kar omogoča uporabo bolj grobe mreže ob steni in s tem prihranek pri potrebnem računskem času. V nasprotju s tem metoda nizkega-Re obravnava tok v viskoznem sloju mejne plasti in zato zahteva gosto mrežo in več računske moči. Samodejno izbiranje med stenskimi funkcijami in nizkim-Re modelom se izvaja glede na razdaljo prvega vozla mreže od stene. V predstavljenih izračunih to omogoča večjo natančnost in manjšo odvisnost rešitve glede na kakovost mreže ob steni.

Zadnja dva izraza v en. (2) pomenita centrifugalno in Coriolisovo silo. Za neviskozne tokove je pokazano, da v nasprotju z radialnimi v aksialnih turbinskih strojih Coriolisova sila ne vpliva

$C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k, σ_ε, β', β, α, σ_ω are constants and P_k is the turbulence production rate due to viscous and buoancy forces.

When using CFD tools to predict the heat transfer in rotating cascades, it should be taken into account that the turbulence structure may be considerably affected by many circumstances, such as the passage curvature, rotation, unsteadiness, flow separation, etc. It has been shown [1] that the viscous and thermal boundary layer cannot be sufficiently predicted by means of earlier, semi-empirical turbulence models, such as the $k–\varepsilon$ model, which leads to substantial invalidating of the heat-transfer predictions. In contrast to other standard eddy-viscosity models, the shear-stress turbulence (SST) model in combination with an optimum wall treatment provides the best compromise between accuracy and computational requirements ([5] and [6]).

The CFD calculations for this study were done using CFX-5.5.1 software, which includes the SST model. The SST model blends between the standard $k–\varepsilon$ and the $k–\omega$ two-equation models, whereby an ω-equation is used for determining the flow near the wall and an ε-equation is used to model the free-stream flow. The blending between both models is done by functions that are based on the distance to the nearest surface and on the flow variables [7]. The ω-equation has significant advantages near the wall and enables improved wall-shear-stress and heat-transfer predictions. Furthermore, it has a very simple low-Re formulation, which is employed in an 'automatic near-wall treatment' [7], a feature of CFX that automatically blends between the wall functions and the low-Re model. The wall functions are empirical formulas that impose suitable conditions near to the wall without resolving the boundary layer equations, thus saving computational resources. In contrast, the low-Re method uncovers details of the boundary layer and therefore requires a fine mesh near the wall and increased computational resources. The automatic blending between both methods is done automatically, depending on the distance of the first node from the wall. This feature, which offers better accuracy and reduced sensitivity of the solution to the near-wall grid resolution, was also used for the calculations in this study.

The last two terms in Eq. (2) represent the centrifugal force and the Coriolis force, respectively. For inviscid flows, it can be shown that in contrast to radial turbomachinery, the Coriolis force in axial na tok skozi kanal med lopaticami [1]. Kljub temu pa je vpliv Coriolisove sile opazen zaradi viskoznih pojavov v toku, ki povzročajo sekundarne tokove. Ti mehanizmi povzročajo tokove znotraj mejne plasti v radialni smeri, kar ima za posledico gradient hitrosti in tlaka v obodni smeri. To povzroča spremembo v strukturi mejne plasti, ki je najpomembnejši del v obravnavi konvektivnega prenosa toplote. Zato vrtenje lahko posredno vpliva na prenos toplote tudi v aksialnih kanalih. V radialnem hladilnem kanalu (npr. centrifugalna turbina, notranje hlajenje lopatice) je vpliv vrtenja na prehod toplote očiten in je bil obširno raziskan [2].

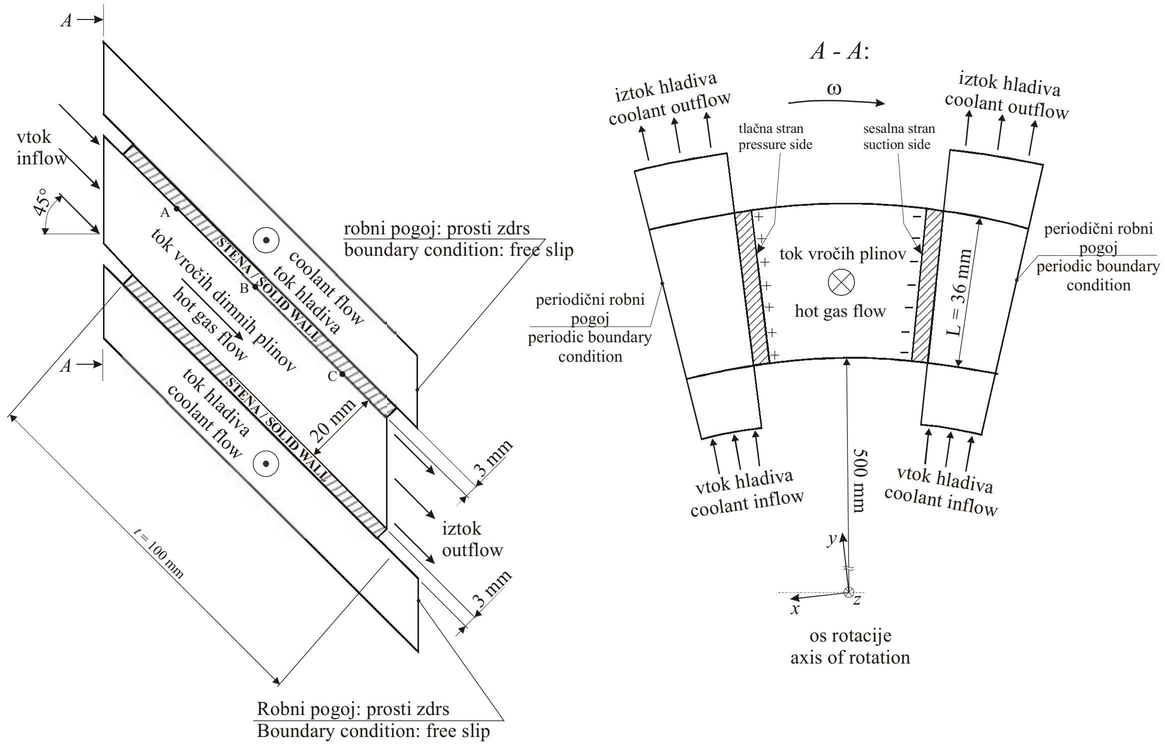
2 NUMERIČNI PREIZKUS

Numerični preizkus je bil narejen s programskim paketom CFX 5.5.1. Geometrijska oblika računskega območja je bila izbrana tako, da je predstavljala sistem konvektivno hlajenih rotorskih lopatic v aksialni plinski turbini. Ker je bil numerični preizkus osredotočen le na vpliv vrtenja pri različnih masnih tokovih hladiva, je bila geometrijska oblika dejanske lopatice močno poenostavljena. Zaradi izločitve vplivov pospeševanja toka in ukrivljenosti površin, ki so običajno navzoči pri obtekanju aerodinamičnih profilov, je bila geometrijska oblika lopatice poenostavljena na kanal med dvema popolnoma ravnima stenama. Rezultat tega je preprosta tri-dimenzijska oblika križnega toka vročega in hladnega plina, ki sta ločena s trdno steno (sl. 1).

turbomachinery does not impact on the flow through the blade passage [1]. However, due to fluid viscous effects, which cause secondary flows, the flow can actually be influenced by Coriolis force mechanisms. These mechanisms introduce radial, outward flows inside the boundary layers resulting in pressure and velocity gradients in the tangential direction. This can cause changes in the structure of the boundary layer, which plays a key role in convective heat transfer. Therefore, the rotational effects can also indirectly influence heat-transfer rates in axial passages. In radial cooling passages (e.g. centrifugal turbine, internal blade cooling), the influence of rotation on the heat transfer is evident, and has been widely researched [2].

2 NUMERICAL EXPERIMENT

The numerical experiment was made with the CFX 5.5.1 software package. The geometry of the numerical model was designed to represent a row of convectively cooled rotor blades in an axial gas turbine. Since the numerical experiment was focused only on the effects of rotation for different coolant mass flows, the real blade geometry was significantly simplified. The effects of flow acceleration and surface curvature, which are normally present around airfoils, were minimized by reducing the blade geometry to a channel between two completely flat walls. This resulted in a simple, 3-dimensional configuration of a hot and cold gas cross-flow with a solid wall at the interface of both gases (Fig. 1). Such a setup allowed the study of a combination of convective heat



Sl. 1. *Geometrijska oblika modela*
Fig. 1. *Model geometry*

Taka postavitev je omogočila preučevanje vpliva vrtilne frekvence kaskade na konvektivni prenos toplote z vročega plina na steno in s stene na hladen plin ter prevoda toplote znotraj trdne stene lopatice.

Smer toka vročega in hladnega plina je bila vzporedna s trdno steno, kar pomeni odsotnost obračanja toka zaradi geometrijske oblike. Kanal vročega plina je poenostavljen kanal med lopaticama aksialne turbine z nespremenljivim kotom 45° glede na obodno smer. Hladilni kanal je usmerjen radialno, tako da omogoča tok hladiva v centrifugalni smeri. Pretočna prereza hladilnega in vročega kanala nista sorazmerna s prerezom v dejanski turbinski kaskadi. Zaradi tega je kot spremenljivka za definiranje različnih primerov numeričnega preizkusa namesto razmerja masnega toka vročega plina in hladiva uporabljena geometrijsko neodvisna hitrost prostega toka hladilnega zraka. Dodatne prilagoditve geometrijske oblike modela (podaljšanje hladilnih kanalov v radialni in aksialni smeri in kanala vročih dimnih plinov v aksialni smeri) so bile namenjene zmanjšanju vpliva robnih pogojev na tokovne razmere ob trdnih stenah. Vroči in hladni kanal sta bila na vstopu podaljšana za razvoj mejne plasti vzdolž stene.

Za diskretizacijo računskega področja, sestavljenega iz enega hladilnega kanala, enega vročega kanala in dveh trdnih sten (sl. 1), je bila izdelana hibridna mreža, sestavljena iz nestrukturiranih prizmatičnih in tetraedričnih elementov. Prizmatični elementi so bili uporabljeni na omočenih površinah trdnih sten, kjer je ustrezna rešitev enačb mejne plasti pomembna za izračun prenosa toplote. Za zmanjšanje numerične difuzije in zagotovitev neodvisnosti rezultatov od mreže je bila izvedena primerjava različno gostih mrež.

Za hladno in vročo tekočino je bil uporabljen model zraka kot realnega plina. Dinamična viskoznost (η), toplotna prevodnost (λ) in specifična toplota (c_p) zraka so bile temperaturno odvisne in modelirane s polinomi petega in šestega reda. Koeficienti polinomov za η, λ in Pr so bili povzeti po [8] ter za c_p po [9]. Toplotna prevodnost snovi trdnih sten je bila izbrana 60,5 W/mK, kar je pogosto uporabljena vrednost pri jeklu.

2.1 Robni pogoji

Vrste robnih pogojev, uporabljenih na mejnih površinah, so prikazane na sliki 1 in v preglednici 1. Vse površine na robovih z izjemo omočenih površin trdnih sten so bile definirane kot "adiabatne". Prosti zdrs tekočine je bil omogočen na pestu in vencu vročega kanala in na stranskih stenah hladilnega kanala, kjer je deformacija tokovnega profila zaradi mejne plasti nezaželena. Preostale površine strukture so bile obravnavane kot hidravlično gladke stene.

transfer from hot gas to solid wall and from solid wall to cold gas, as well as conduction inside the blade's solid wall.

Both hot and cold gas flows were parallel to the solid walls, so that there was no flow turning as a result of the geometry. The hot-gas channel was a simplified axial blade passage with a constant circumferential inclination at 45°. The cooling channel was oriented radially, whereas the coolant flow was centrifugal. The flow cross-sections of the cooling and hot-gas channels were not proportional to the conditions in the actual gas-turbine cascade. Therefore, the use of the hot-gas-to-coolant mass-flow ratio for defining various cases of the numerical experiment was avoided and a geometry-independent variable, the free-stream velocity, was used. Some further modifications of the model's geometry (extensions of the cooling channel in the radial and axial directions and the hot-gas channel in the axial direction) were made in order to minimize the influence of the boundary conditions on the flow conditions over solid walls. Extensions were used at both the hot-gas and cold-gas inflows in order to allow the development of a boundary layer upstream of the solid walls.

An unstructured hybrid mesh consisting of prismatic and tetrahedral elements was made to discreticize the computational domain composed of one cooling channel, one hot-gas channel and two solid walls (Fig. 1). Prismatic elements were used on the wetted surfaces of the solid walls, where an appropriate boundary-layer equations solution is important for the calculation of the heat transfer. A grid-refinement study was performed in order to minimize the numerical diffusion and thus achieve grid-independent results.

An air, real-gas model was used for both hot and cold fluids. The dynamic viscosity (η), the thermal conductivity (λ) and the specific heat capacity (c_p) of air were set as temperature dependent and modelled with polynomials of the 5th and 6th orders. The coefficients of the polynomials for η, λ and Pr were obtained from [8], and for c_p from [9]. The thermal conductivity of the solid-wall material was set to 60.5 W/(mK), a typical value for steel.

2.1 Boundary conditions

The types of boundary conditions applied to the bounding surfaces are shown in Fig. 1 and Table 1. All the surfaces, with the exception of the wetted surfaces of the solid walls, were set as "adiabatic". The free slip of fluid was allowed at the hub and the shroud of the hot channel and at the sidewalls of the cooling channel, where flow-profile distortions due to the boundary layer were undesirable. Other surfaces were treated as hydraulically smooth walls with no slip influence on the flow.

Preglednica 1. *Stenski robni pogoji*
Table 1. *Wall boundary conditions*

adiabatna stena adiabatic wall	$\dot{q}_{\mathrm{w}} = 0$
hidravlično gladka stena hydraulically smooth wall	$w_{\mathrm{n,w}} = 0$
prosti zdrs ob steni free-slip wall	$w_{\mathrm{n,w}} = 0,\ \tau_{\mathrm{w}} = 0$

Kakor prikazuje preglednica 2, je bilo za ugotovitev vpliva vrtenja na prenos toplote izračunanih več primerov pri različnih vrtilnih frekvencah in različnih vstopnih hitrostih hladiva. Vrtilno hitrost izrazimo v brezdimenzijski obliki z uporabo vrtilnega števila, [1]:

In order to determine the influence of rotation on heat transfer, a series of cases for variable rotational speeds and velocities of cooling air at the inflow was calculated, as shown in Table 2. The rotational velocity is expressed in terms of the rotational number, [1]:

$$\mathrm{Ro} = \frac{\omega \cdot L}{w} \tag{8}$$

kjer so ω kotna hitrost rotorja, L značilna dolžina (dolžina tetive) in w značilna hitrost (aksialna hitrost dimnih plinov). V preglednici 2 je χ masno razmerje hladiva in dimnih plinov:

where ω is the angular speed of the rotor, L is the characteristic length (cord length) and w is the characteristic speed (axial speed of gases). In Table 2, χ is the coolant-to-gas mass-flow ratio:

$$\chi = \frac{\dot{m}_{\mathrm{c}}}{\dot{m}_{\mathrm{g}}} \tag{9}$$

Za vse primere je bil vtok vročega plina izbran pri statični temperaturi 1000 °C in hitrosti 100 m/s v smeri, vzporedni stenam, medtem ko je bila statična temperatura hladilnega zraka na vstopu vedno 400 °C. Tako hladen kakor vroči zrak sta vstopala pri intenzivnosti turbulence Tu = 5 %. Iztoka vročega in hladnega plina sta bila definirana kot površini pri 15 bar statičnega tlaka. Vsi izračuni so se izvajali tako dolgo, dokler se niso normalizirani največji ostanki kontinuitetne, gibalne in energijske enačbe zmanjšali pod vrednost 1×10^{-3}.

For all cases, the hot-gas inflow was set at a steady temperature of 1000 °C and a velocity of 100 m/s in the direction parallel to the walls, whereas the steady temperature of the cooling air at the inflow was maintained at 400 °C. Both the hot and cold air were entering the domain at a turbulence intensity of Tu = 5 %. The outflows of the hot gas and the cold gas were defined as areas at a static pressure of 15 bar. All the simulations were performed until the normalized maximum residuals of the mass, the momentum and the energy equation were below a value of 1×10^{-3}.

3 POLIZKUSTVENI MODEL

3 SEMI-EMPIRICAL MODEL

Pri prenosu toplote se pogosto upoštevajo naslednje poenostavitve: Temperaturna porazdelitev po stenah lopatic je popisana s povprečjem pri določeni višini lopatice. Zanemarimo prevod toplote vzdolž stene. Temperatura stene je nespremenljiva po vsej debelini stene zaradi relativno tanke stene.

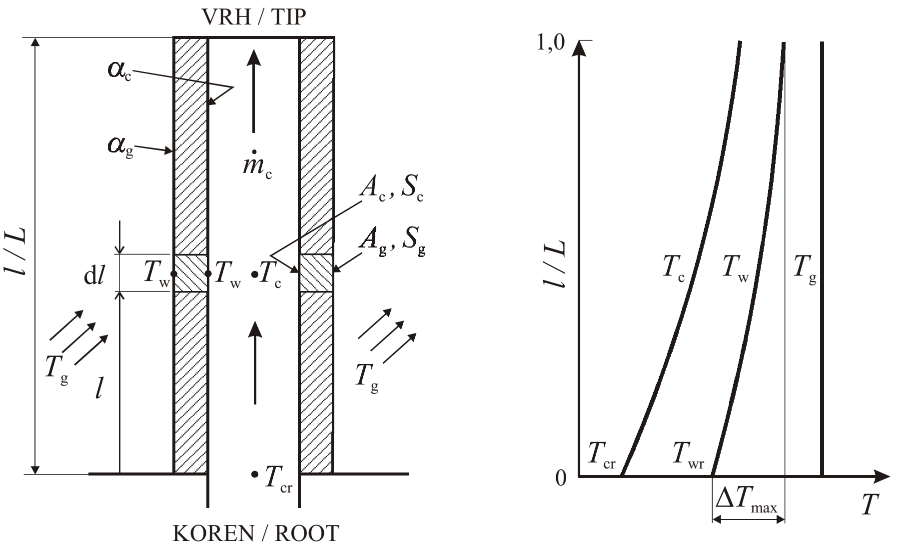
Tako se problem prevede v enodimenzijskega, odvisnost srednjih temperatur hladiva in stene od višine turbinske lopatice pa dobimo iz energijske bilance [10]:

In order to obtain a first approximation for the convective heat transfer in a cooled turbine blade, the problem is often reduced to a one-dimensional one, which yields an averaged temperature distribution along the blade's height. In this case, certain additional model simplifications are taken into account: conductive heat transfer in the span-wise direction is neglected, and the wall temperature is considered to be constant across the entire thickness due to the relatively thin wall.

The variations of coolant and wall temperatures vs. blade height can be derived from the heat balance [10]:

$$T_{\mathrm{c}}(l) = T_{\mathrm{g}} - \left(T_{\mathrm{g}} - T_{\mathrm{cr}}\right) \cdot e^{-\mathrm{M}\frac{l}{L}} \tag{10}$$

$$T_{\mathrm{w}}(l) = T_{\mathrm{g}} - \frac{T_{\mathrm{g}} - T_{\mathrm{cr}}}{1 + \dfrac{\alpha_{\mathrm{g}} S_{\mathrm{g}}}{\alpha_{\mathrm{c}} S_{\mathrm{c}}}} \cdot e^{-\mathrm{M}\frac{l}{L}} \tag{11}$$



Sl. 2. *Shematični prikaz hlajene turbinske lopatice*
Fig. 2. *Schematic view of the cooled turbine blade*

kjer je T_{cr} temperatura hladiva v korenu lopatice in izraz M v potenci pomeni:

where T_{cr} is the coolant root temperature and M represents:

$$\mathrm{M} = \frac{\alpha_g S_g L}{\dot{m}_c c_{pc} \left[1 + \left(\alpha_g S_g / \alpha_c S_c\right)\right]} \tag{12}$$

Koeficienta toplotne prestopnosti α_g in α_c sta lahko določena eksperimentalno ali z uporabo povezav, ki so po navadi za konvektivni prenos toplote podane v obliki Nusseltovih števil [1]:

The heat-transfer coefficients α_g and α_c can be determined experimentally or by using correlation relationships, which are usually given as the Nusselt number for convective heat transfer [1]:

$$\mathrm{Nu}_g = \frac{\alpha_g t}{\lambda_g}, \qquad \mathrm{Nu}_c = \frac{\alpha_c D}{\lambda_c} \tag{13}$$

kjer D in t pomenita značilni dolžini: hidravlični premer hladilnega kanala znotraj lopatice in dolžino tetive. λ je koeficient toplotne prevodnosti.

where D and t represent characteristic lengths: the hydraulic diameter of the internal cooling passage and the blade chord length, respectively. λ is the thermal conductivity.

Za tok hladiva skozi kanal poljubnega radialnega prereza lahko zapišemo [1]:

For coolant flow through a pipe of an arbitrary cross section, the following equation can be written [1]:

$$\mathrm{Nu}_c = \mathrm{A}\left(\mathrm{Re}\right)^{\mathrm{m}}\left(\mathrm{Pr}\right)^{\mathrm{n}}\left(\frac{L}{D}\right)^{\mathrm{E}}\left(\frac{T_c}{T_w}\right)^{\mathrm{F}} \tag{14}$$

kjer so: L dolžina cevi, D hidravlični premer, T_c temperatura hladiva, T_w temperatura stene in A, E, F, m, n konstante modela.

where L is the pipe length, D is the hydraulic diameter, T_c is the coolant temperature, T_w is the wall temperature and A, E, F, m, and n are model constants.

Za prenos toplote na strani vročih plinov za tokove v turbinskih strojih lahko zapišemo Nusseltovo število v obliki [1]:

For external heat transfer, the Nusselt number correlation for turbine flows can be written in the following form [1]:

$$\mathrm{Nu}_g = \mathrm{A}\left(\mathrm{Re}\right)^{\mathrm{m}}\left(\mathrm{Pr}\right)^{\mathrm{n}}\left(\frac{T_g}{T_w}\right)^{\mathrm{C}} F\left(\mathrm{Tu}\right) F\left(K\right) \tag{15}$$

kjer so: T_w temperatura stene, Tu intenzivnost turbulence, K faktor pospeška in A, C, m, n konstante modela.

where T_w is the wall temperature, Tu is the turbulence intensity, K is the acceleration factor and A, C, m, and n are model constants.

En. (10), (11) in (13) sestavljajo sistem enačb, ki se rešuje iterativno z upoštevanjem primernih povezav in vstopnih pogojev. V obravnavanem primeru so bile toplotne prestopnosti določene s

Eqs. (10), (11) and (13) constitute a system of equations that has to be solved iteratively, applying the appropriate correlation relationships and initial values. In our case, the heat-transfer coefficients were

pomočjo treh različnih povezav:
- prenos toplote na turbinski lopatici [10];
- obravnava prenosa toplote na ravni steni [11];
- prenos toplote za tok preko ravne stene [9].

Posamezni nastavki v polizkustvenem modelu so bili medsebojno primerjani na testnem primeru lopatice za različna razmerja masnih tokov χ, pri čemer se drugi robni pogoji niso spremenili. Rezultati primerjave kažejo podobno obnašanje modela ob uporabi različnih povezav.

calculated from three different correlations:
- the general turbine-blade treatment [10];
- the flat-plate heat-transfer treatment [11];
- the flat-plate treatment [9].

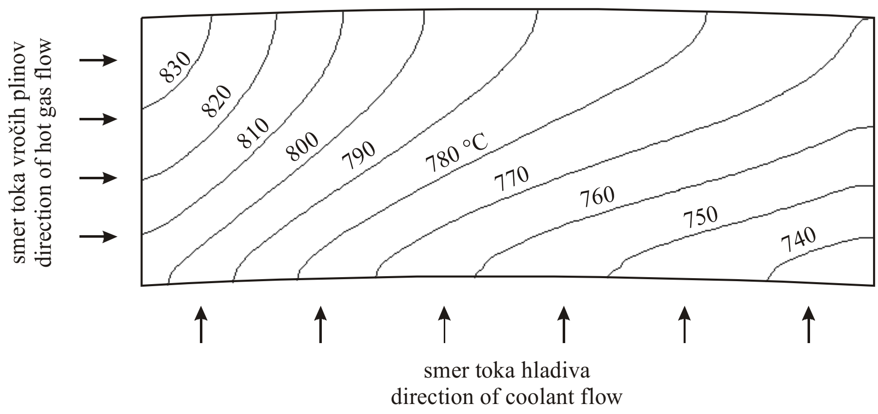
All three heat-transfer correlations were introduced as a heat-transfer model within the blade at the same boundary conditions in order to analyse the impact of the coolant mass-flow ratio on the temperature distribution. Good agreement between all the applied models is evident.

4 REZULTATI

Pri analizi rezultatov smo se osredotočili na tlačno stran lopatice, ki je termično bolj obremenjena od nasprotne, sesalne strani (sl. 1). Slika 3 prikazuje izračunane izoterme na tlačni površini lopatice za mirujoči model. Porazdelitev izoterm ustreza pričakovani porazdelitvi za prikazani sistem križnega toka.

4 RESULTS

All of the below analyses focus only on one side of the model, i.e. the wall representing the blade pressure surface (Fig. 1). Figure 3 shows the computed temperature contours on the blade-pressure surface for a non-rotating model. The distribution shown is in agreement with the expected distribution for the presented cross-flow system.



Sl. 3. *Temperaturna porazdelitev na tlačni strani numeričnega modela lopatice (primer Ro = 0, χ= 4,3)*
Fig.3. *Temperature distribution on the numerical-model blade-pressure surface (case at Ro = 0, χ = 4.3)*

Ustreznost mreže ob steni je bila preverjena s pomočjo brezdimenzijske razdalje do stene (y^+). Ugotovljena je bila povprečna vrednost y^+ = 21,8 na vroči površini in y^+ = 15,4 na hladni površini trdne stene. Ob upoštevanju avtomatične obravnave toka ob steni [7] in dobljenih povprečnih vrednosti y^+ nad 11, kjer je meja med viskoznim podslojem (model nizkih Re) in slojem logaritmičnega zakona (stenske funkcije), sledi, da je na izračune ob steni prevladujoče vplivala metoda stenskih funkcij.

V preglednici 2 so predstavljena povprečna Nusseltova števila na vroči strani stene, izračunana z numeričnim preizkusom. Polizkustveni model je bil usklajen z numeričnim modelom tako, da so bila dosežena enaka povprečna Nusseltova števila na vroči strani. Ker polizkustveni model ne upošteva vpliva vrtenja, so bile uporabljene vrednosti Nu za primer brez vrtenja.

Slika 4 prikazuje z numeričnim preizkusom dobljene temperaturne porazdelitve po višini lopatice

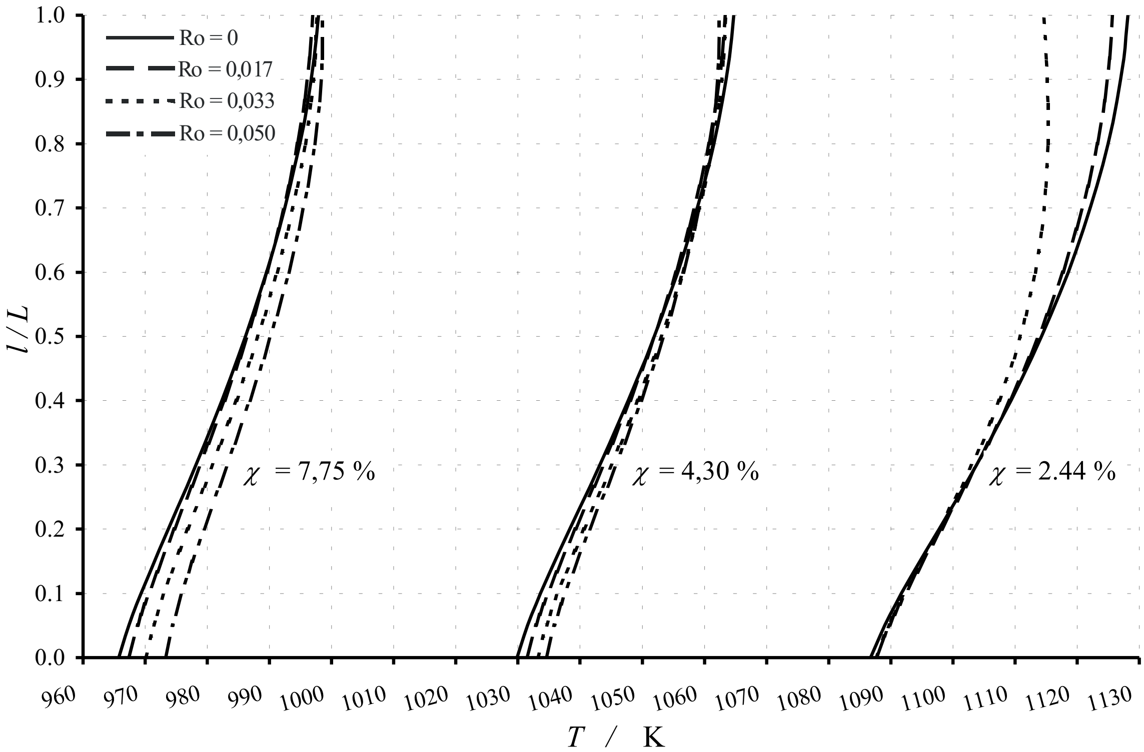
In order to verify the near-wall grid resolution, the dimensionless distance from the wall (y^+) was checked. The average values were found to be 21.8 on the hot surface and 15.4 on the cold surface of the solid wall. Considering the use of automatic near-wall treatment [7] and the resulting average values of y^+ above 11, which is at the border between the viscous sublayer (low-Re model) and the log law layer (wall function), calculations near the wall were predominantly influenced by the wall-function method.

The semi-empirical model was synchronized with the numerical one in order to achieve an identical average Nusselt number on the hot side of the wall. Table 2 shows the average Nusselt numbers obtained from a numerical experiment on the hot side of the wall. Since the semi-empirical model does not account for rotational effects, Nu values for a 0 rotation case were used.

Fig. 4 shows the average temperature distributions along the blade height for different

Preglednica 2: *Povprečna Nusseltova števila na vroči strani stene*
Table 2. *Average Nusselt numbers on the hot side of the wall*

χ \ Ro	0	0,017	0,033	0,050
2,44	274,6	275,5	275,8	
4,30	275,1	277,3	275,9	276,0
7,75	276,4	276,6	277,1	277,2



Sl. 4. *Porazdelitve povprečne temperature na vroči površini stene*
Fig. 4. *Average hot-surface temperature distributions*

pri različnih vrtilnih frekvencah in različnih razmerjih masnega toka hladiva.

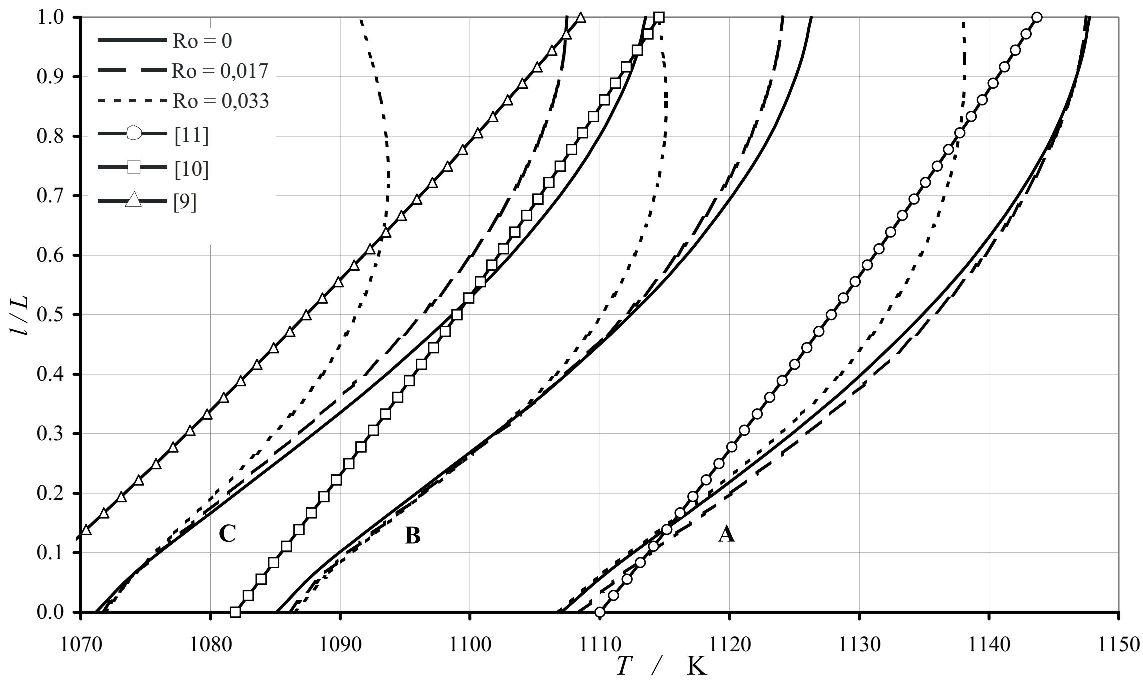
Vpliv razmerja masnega toka χ in vrtilne frekvence na temperaturno porazdelitev je očiten. S povečevanjem χ se izboljšuje prenos toplote na hladni strani, zato se temperatura stene zmanjša. Temperatura v korenu stene se v primeru vrtenja nekoliko poveča (preglednica 2 in sl. 4), medtem ko na temperaturo na vrhu stene poleg vrtenja vpliva tudi odlepljanje toka v hladilnem kanalu, ki ni modeliran v skladu z dejanskimi razmerami. Zaradi tega je temperatura v korenu stene, dobljena z numeričnim preizkusom, bolj pomembna od temperature na vrhu lopatice. V plinskih turbinah so toplotne in mehanske obremenitve v korenu lopatice kritične, zato je poznavanje temperaturnih porazdelitev v korenu zelo pomembno [2].

Na sl. 5 so predstavljene temperaturne porazdelitve na treh različnih legah pri razmerju mase hladiva χ = 2,44. Položaji A, B in C se nahajajo pri 20 %, 51 % in 83 % tetive lopatice (sl. 1). Narejena je primerjava med temperaturnimi porazdelitvami,

rotational speeds and for different coolant mass-flow ratios obtained with the numerical experiment.

The effect of the mass-flow ratio χ and the rotational number on the temperature distribution is evident. As χ increases, the heat transfer on the cold side improves, therefore the wall temperatures decrease. The temperature at the wall root slightly increases with rotational speed (Table 2 and Fig. 4), whereas the temperature at the wall tip is affected by the flow separation in the coolant channel, which is not modelled in accordance with real circumstances. Therefore, the temperature at the wall root obtained with the numerical model is assumed to be more relevant than the wall-tip temperature. In gas turbines, the thermal and mechanical loads at the blade root are critical, therefore, a knowledge of the temperature distributions at the blade root is very important [2].

Fig. 5 presents the temperature distributions for three different positions in the case of a flow ratio χ = 2.44. The positions A, B and C are located at 20 %, 51 % and 83 % blade chord, respectively. A comparison is made between the temperature



Sl. 5. *Temperaturne porazdelitve pri χ = 2,44*
Fig. 5. *Temperature distributions for χ = 2.44*

dobljenimi z numeričnim modelom in polizkustvenim modelom z različnimi zgoraj navedenimi modeli za Nusseltovo število.

Največje temperature se pojavijo na položaju A, ki je prva lega za vtokom vročega plina (sl. 1). Temperaturne porazdelitve, izračunane s polizkustvenim modelom pri različnih povezavah za Nusseltovo število, so istega velikostnega razreda kakor numerično dobljeni rezultati. Temperaturni gradienti vzdolž višine stene za numerični in polizkustveni model se dobro ujemajo. Razlike med posameznimi polizkustvenimi porazdelitvami so le posledica različnih povezav za Nusseltovo število za vročo in hladno površino stene. Z nastavkom prenosa toplote skozi ravno steno [11] smo izračunali najvišje temperature vzdolž vroče stene lopatice.

Kakor je bilo že rečeno, vrtenje poveča prenos toplote, kar se kaže s povišanjem temperatur na steni lopatice. S slike 5 je razvidno, da ima v primeru modela z upoštevanjem vrtenja odlepljeni tok ob iztočnem robnem pogoju za hladivo znaten vpliv na temperaturno porazdelitev, še posebej pri vrhu stene. Zato je primer pri razmerju masnega toka hladiva 2,44 manj primeren za izhodišče za nadaljnje raziskave vpliva vrtenja v predstavljenem numeričnem modelu.

distributions obtained with the numerical model and those obtained with the semi-empirical model for different Nusselt number models.

The maximum temperatures occur at position A, i.e. the first position that is downstream of the hot-gas inlet (Fig. 1). The temperature distributions calculated with the semi-empirical model using different Nusselt number correlations are of the same magnitude as the numerically obtained results. The temperature gradients along the wall height for the numerical and the semi-empirical models show good agreement. The differences between individual semi-empirical distributions occur only because of the different Nusselt-number correlations for the hot and cold surfaces of the wall. In the studied case, the semi-empirical model with the Nusselt-number correlation that introduces the flat-plate heat-transfer treatment [11] predicts the highest temperature distributions along the wall.

As was already mentioned, rotation tends to increase the wall temperature. Fig. 5 clearly shows that the outflow boundary condition has a significant influence on the temperature distribution when rotation is applied to the model, especially at the wall tip. Therefore, the case with a coolant mass-flow ratio of 2.44 is less appropriate for use as a starting point for further investigations of the rotational effects in the presented numerical model.

5 SKLEPI

V prispevku je bil predstavljen numerični proces pri opazovanju stacionarnega prehoda toplote na konvektivno hlajenih ravnih lopaticah v vrteči

5 CONCLUSIONS

This paper presents a numerical approach to stationary heat-transfer determination of convectively cooled flat blades within a rotating axial cascade. The aksialni kaskadi. Raziskava je bila osredotočena na določitev vpliva razmerja masnega toka hladiva in vrtilne frekvence na prenos toplote in posledično na temperaturne porazdelitve v modeliranem sistemu križnega toka. Kakor je bilo pričakovano, vrtilna frekvenca v aksialnem turbinskem kanalu nima pomembnega vpliva na temperaturne porazdelitve. Z uporabo numeričnih analiz so bili ocenjeni trije polizkustveni modeli. V vseh obravnavanih primerih je bilo ujemanje z numeričnim modelom najboljše pri modelu, ki obravnava prenos toplote na ravni steni.

focus of our research was to determine the influence of varying coolant mass-flow ratios and rotational speeds on the heat transfer, and consequently on the temperature distribution, in the modelled cross-flow system. As expected, the rotational speed in the axial turbine passage has no significant impact on the temperature distributions. Using the results of numerical analyses, three semi-empirical models were evaluated. In all the tested cases, the flat-plate heat-transfer treatment yielded the best agreement with the numerical model.

6 SIMBOLI
6 SYMBOLS

specifična toplota	c_p	J/kgK	specific heat capacity
hidravlični premer hladilnega kanala	D	m	hydraulic diameter of internal cooling passage
statična entalpija	h	kJ/kg	static enthalpy
turbulentna kinetična energija	k	m^2/s^2	turbulence kinetic energy
faktor pospeška	K	-	acceleration factor
višina lopatice	L, l	m	blade height
masni tok	$\dot{m}$	kg/s	mass-flow rate
Nusseltovo število	Nu	-	Nusselt number
statični tlak	p	bar	static pressure
Prandtlovo število	Pr	-	Prandtl number
radij	r	m	radial distance from the axis
Reynoldsovo število	Re	-	Reynolds number
vrtilno število	Ro	-	rotational number
obseg	S	m	wetted perimeter
vir oz. ponor toplote	S_E	W	heat source and heat sink
dolžina tetive, čas	t	m, s	blade chord length, time
temperatura	T	°C	temperature
intenzivnost turbulence	Tu	%	turbulence intensity
brezdimenzijska razdalja od stene	y^+	-	non-dimensional distance from wall
relativna hitrost	w	m/s	relative velocity
Grške črke			**Greek letters**
toplotna prestopnost	α	W/m^2K	heat-transfer coefficient
Kroneckerjeva delta funkcija	δ	-	Kronecker tensor
temperaturna razlika	ΔT	°C	temperature change
raztros turbulentne kinetične energije	ε	m^2/s^3	turbulence eddy dissipation
dinamična viskoznost	η	Pas	dynamic viscosity
toplotna prevodnost	λ	W/mK	thermal conductivity
turbulentna viskoznost	μ_t	kg/ms	turbulence viscosity
gostota tekočine	ρ	kg/m^3	fluid density
strižna napetost	τ	Pa	shear stress
masno razmerje hladiva	χ	-	coolant mass-flow ratio
kotna hitrost, frekvenca turbulence	ω	s^{-1}	angular velocity, turbulence frequency
Indeksi			**Subscripts**
glavnina	b		bulk
hladivo	c		coolant
plin	g		gas
največje	max		maximum
normalna smer	n		normal direction
koren	r		coolant root
celotni	tot		total
stena	w		wall

7 LITERATURA
7 REFERENCES

[1] Lakshminarayana, B. (1996) Fluid dynamics and heat transfer of turbomachinery, *John Wiley & Sons*.
[2] Han, J.-C., S. Dutta, S. V. Ekkad (2000) Gas turbine heat transfer and cooling technology, *Taylor & Francis*.
[3] Florjančič, S. (2002) Gas turbine cooling, manufacturing and life interaction, *VDI-Berichte 1721*, Düsseldorf.
[4] Bradshaw, P. (ed.) (1970) Turbulence: Topics in applied physics, Vol. 12, *Springer-Verlag*, New York.
[5] Vieser, W., T. Esch, F. Menter (2002) Heat transfer predictions using advanced two-equation turbulence models, CFX Validation Report, CFX-VAL 10/1002.
[6] Esch, T. (2003) Heat transfer predictions using advanced two-equation turbulence models, *The 6th ASME-JSME Thermal Engineering Joint Conference,* march 2003.
[7] CFX-5 Documentation (2001) Mathematical models and solution algorithms.
[8] Brandt, F. (1985) Wärmeübertragung in Dampferzeugern und Wärmeaustauschern, Essen.
[9] Holman, J.P. (1992) Heat transfer, *McGraw-Hill*.
[10] Cohen, H., GFC Rogers, HIH Saravanamuttoo (1996) Gas turbine theory, 4th Edition.
[11] Gnielinski, V. (1975) Forsch. –Ing. Wes. 41, 145/50.
[12] Škerget, L. (1994) Mehanika tekočin, *Tehniška fakulteta v Mariboru in Fakulteta za strojništvo v Ljubljani*, Ljubljana.

Naslov avtorjev: Mitja Mori
Lovrenc Novak
doc. dr. Mihael Sekavčnik
Fakulteta za strojništvo
Univerza v Ljubljani
Aškerčeva 6
1000 Ljubljana
mitja.mori@fs.uni-lj.si
lovro.novak@fs.uni-lj.si
mihael.sekavcnik@fs.uni-lj.si

Authors' address: Mitja Mori
Lovrenc Novak
Doc. Dr. Mihael Sekavčnik
Faculty of Mechanical Eng.
University of Ljubljana
Aškerčeva 6
1000 Ljubljana, Slovenia
mitja.mori@fs.uni-lj.si
lovro.novak@fs.uni-lj.si
mihael.sekavcnik@fs.uni-lj.si

Prejeto:
Received: 10.6.2003

Sprejeto:
Accepted: 12.9.2003

Odprto za diskusijo: 1 leto
Open for discussion: 1 year