© Strojniški vestnik 47(2001)2,70-82
ISSN 0039-2480
UDK 621.224.24:532.57
Izvirni znanstveni članek (1.01)
© Journal of Mechanical Engineering 47(2001)2,70-82
ISSN 0039-2480
UDC 621.224.24:532.57
Original scientific paper (1.01)

Napoved izkoristka francisove turbine z numeričnim izračunom toka

Using Numerical Flow Analysis to Predict the Efficiency of a Francis Turbine

Dragica Jošt - Leopold Škerget

V prispevku je predstavljena numerična analiza toka v francisovi turbini. Osredotočili smo se na napoved energijskih izgub v toku in napoved izkoristka turbine. Rezultate, dobljene z ločeno analizo toka v vsakem delu turbine, smo primerjali z rezultati ločenega izračuna toka v spirali in skupnega izračuna toka skozi preostalo turbino. Nato smo izračunali tok v francisovi turbini v večjem številu obratovalnih točk, narisali školjčni diagram izkoristka in ga primerjali z izmerjenim. Enak izračun je bil narejen še za primer, ko nimamo izmerjenih vstopnih podatkov. V prispevku je predstavljen tudi vpliv gostote mreže in izbire turbulentnega modela na rezultate.

(Ključne besede: turbine francisove, izkoristek turbin, analize toka, modeli turbulentni)

This paper presents a numerical analysis of flow in a Francis turbine. We concentrate on flow-energy losses and efficiency prediction. The results, obtained by a separate analysis of each turbine component, are compared with the results of a separate analysis of flow in a spiral casing and a simultaneous calculation of the flow through the other turbine parts. After this we analysed flow in a Francis turbine at several operating points, an efficiency hill-chart diagram was drawn and compared with the measured one. The same calculation was made for a case where no measured inlet conditions were avaible. The effect of the grid density and turbulence models on the results is also presented.

(Keywords: Francis turbine, turbine efficiency, flow analysis, turbulence models)

0 UVOD

Od rezultatov numerične analize toka v turbini pričakujemo natančno informacijo o toku, primerno točen izračun tokovnih izgub in izkoristka in napoved kavitacije. Pri oblikovanju novih gonilnikov in drugih delov turbinskih strojev je numerična analiza toka nepogrešljivo orodje. Mnogo laže je oblikovati veliko število lopatic gonilnika na računalniku in na podlagi numerične analize izbrati najboljšega, kakor pa izdelati številne modele in z meritvami izkoristka izbrati najboljšega. Pomembno pa je, da na podlagi numeričnih rezultatov res izberemo najboljši gonilnik. Bolj kot absolutna vrednost numerično dobljenega izkoristka nas zanimata lega optimalne točke obratovanja in oblika diagrama izkoristka.

V preteklosti smo računali vsak del turbine posebej. Rezultate analize toka skozi en del turbine smo uporabili za vstopne pogoje pri analizi naslednjega dela. Tak izračun ne upošteva vpliva

0 INTRODUCTION

Numerical results are expected to give detailed information about the flow in a turbine; to predict flow-energy lossses and efficiency with reasonable accuaracy; and to foresee the cavitation. In the design process of runners and other turbine components CFD is a useful tool: it is much easier to design a number of runner blades on a computer and numerically choose the best one than to do several models and model tests. But it is essential to choose the best runner. More than the absolute value of efficiency, it is important to accurately obtain the position of the best-efficency point and the shape of the efficiency diagram.

In the past, each part of a turbine was analysed separately. The results of the flow through one part of a turbine were used as inlet boundary conditions for the analysis of the next part. However, such a flow calculation does not take into account the influence of one turbine component on the previene komponente turbine na poprejšnjo. Tako izgubimo vpliv gonilnika na tok v dvojni kaskadi in vpliv sesalne cevi na tok v gonilniku. Kljub temu pa so bili rezultati ločene analize toka pogosto uspešno uporabljeni pri izboljšanju hidravličnih oblik vseh delov turbine ([1] in [2]).

V zadnjih letih je bil v numeričnem obravnavanju toka tekočin dosežen izreden napredek. Eden najpomembnejših dosežkov je skupni izračun toka v rotirajočih in nerotirajočih delih stroja. Tako je zdaj mogoče skupaj računati tok od vstopa v spiralo do izstopa iz sesalne cevi z vsemi predvodilnimi, vodilnimi in gonilnimi lopaticami. Tako upoštevamo medsebojni vpliv statorja, gonilnika in sesalne cevi in se izognemo nenatančnim robnim pogojem med komponentami. Slaba stran takega izračuna je veliko število vozlov, počasna konvergenca in dolgi računski časi. Kompromisna rešitev je ločen izračun toka v spirali in skupen izračun toka od vstopa v predvodilnik do izstopa iz sesalne cevi, območja računanja pri kaskadah predvodilnika, vodilnika in gonilnika pa so skrčena na en perodični del ([3] in [4]). Pogosto pa je tudi tak izračun prezamuden in se moramo odločiti za ločeno analizo toka. Zanesljivost numeričnih rezultatov je odvisna tudi od gostote mreže. V primeru premajhnih računalniških zmogljivosti je vprašanje, ali je bolje računati celotno turbino na redki mreži ali pa vsak del turbine posebej na zgoščeni mreži. Rezultati so odvisni tudi od izbire turbulentnega modela.

V tem prispevku skušamo prikazati razlike med rezultati ločenega in skupnega izračuna, vpliv gostote mreže in izbire turbulentnega modela na rezultate in zanesljivost numeričnega izračuna izkoristka turbine.

1 LOČENA, DELNO SKLOPLJENA IN SKLOPLJENA ANALIZA TOKA

Numerična analiza toka je bila narejena za model francisove turbine s specifično vrtilno frekvenco n_S=300 (n_s=3.65 $n\ Q^{1/2}\ H^{-3/4}$), ki je bila izmerjena na Turboinštitutu. Turbina sestoji iz spirale z 12 predvodilnimi lopaticami, 24 vodilnih lopatic, iz 13-lopatičnega gonilnika in kolenaste sesalne cevi z navpičnim rebrom v izstopnem delu.

Numerična analiza toka je bila narejena s programskim paketom CFX-TASCflow s standardnim modelom k - ε.

Obratovalna točka turbine je določena s padcem, pretokom in vrtljaji. Namesto pretoka in padca raje uporabljamo pretočno število φ ($\varphi=Q/(\pi\omega r^3)$) in tlačno število ψ ($\psi=2gH/(\omega r)^2$). Tu sta φ in ψ brezdimenzijski števili in sta neodvisni od velikosti stroja. Numerična analiza toka je bila narejena za pet obratovalnih točk pri nominalnem ψ. Pretok pri določenem odprtju vodilnika in vrtljajih je bil dobljen iz meritev.

ous one. So the influence of a runner on the flow through the distributor and the influence of a draft tube on the flow in the runner were lost. In spite of this, results of separate analyses were used successfully to improve the hydraulic shapes of all turbine parts ([1] and [2]).

Recently, there has been a rapid development in CFD and one of the most important achievements is simultaneous calculation of the flow in rotating and non-rotating parts. It is now possible to calculate the flow from the spiral casing inlet to the draft-tube outlet with all the stay and guide vanes and the runner blades, simultaneously. In this way we take into account the interaction of the stator, the rotor and the draft tube and avoid inaccurate boundary conditions between the turbine components. The disadvantages of such a calculation are the large number of nodes, the slow convergence and the long CPU time. The compromise solution is a separate analysis of the spiral casing and a simultaneous calculation of the flow from the stay-vanes inlet to the draft-tube outlet, while the domain for the stay and guide vanes and the runner-blades cascades is reduced to one periodic part ([3] and [4]). Often, even this kind of calculation is too time consumming and a separate analysis has to be performed. The reliability of the numerical results also depends on the grid density. In the case of insufficient computer capacity there is a question as to whether it is better to calculate the whole turbine on a coarse grid or to calculate each part individualy on a fine grid. The results also depend on the turbulence model.

In this paper the difference between the results of separate and coupled analysis, the effect of grid density and the turbulence model on the results and the reliability of numerically predicted efficiency are presented.

1 SEPARATED, PARTLY COUPLED AND COUPLED-FLOW ANALYSIS

A numerical analysis was made for a model of a Francis turbine with specific speed n_S=300, (n_s=3.65 $n\ Q^{1/2}\ H^{-3/4}$), which was tested on the test rig at the Turboinstitute. The turbine consists of a spiral casing with 12 stay vanes, 24 guide vanes, a 13-blades runner and an elbow draft tube with a vertical pier.

The numerical analysis was made with the CFX-TASCflow computer code using the standard k - ε model.

The turbine operating point is determined by head, discharge and speed. Often, instead of discharge and head, a discharge coefficient φ ($\varphi=Q/(\pi\omega r^3)$) and a pressure coefficient ψ ($\psi=2gH/(\omega r)^2$) are used. Here φ and ψ are dimensionless numbers independent of the turbine dimensions. The numerical analysis was made for five operating points for a nominal ψ. A discharge corresponding to a certain-guide vane opening and speed was obtained from measurements.

Najprej je bila narejena analiza toka v spirali s predvodilnimi lopaticami. Območje računanja je razširjeno do izstopa iz gonilnika, vendar brez vodilnih in gonilnih lopatic. V mreži je 280 000 vozlov (sl. 1). Iz rezultatov izračuna toka v spirali smo dobili vstopne pogoje za nadaljnje izračune. Izračunali smo tudi izgube v spirali.

Numerična analiza toka v dvojni kaskadi, gonilniku in sesalni cevi je bila narejena na tri načine. Prvi način je bil ločen izračun toka v kaskadi, gonilniku in sesalni cevi. Območje računanja za dvojno kaskado je del med dvema predvodilnima in tremi vodilnimi lopaticami in med vencem in pestom, toda brez lopatic gonilnika. V mreži je 114 000 vozlov. Območje računanja za gonilnik je med dvema lopaticama, v mreži je 72 000 vozlov. Mreža za sesalno cev vsebuje 170 000 vozlov. Vstopni pogoji so dobljeni iz analize prejšnje komponente. Območja računanja se prekrivajo, ker smo želeli zmanjšati vpliv nenatančnih izstopnih robnih pogojev. Drugi način je delno sklopljena analiza toka. Tok skozi predvodilne in vodilne lopatice ter gonilnik računamo skupaj (166 000 vozlov), tok v sesalni cevi pa posebej (170 000 vozlov). Tretji način je skupen izračun toka od vstopa v predvodilnik do izstopa iz sesalne cevi. Mreža vsebuje 332 000 vozlov (sl. 2). Mreže za sklopljeno analizo so bile dobljene z združevanjem mrež ločenega izračuna, izpuščeni so bili le deli, ki se prekrivajo. Zato je struktura in gostota mreže enaka za ločen in sklopljen izračun.

Iz rezultatov numeričnega izračuna lahko izračunamo izgube v toku, navor na os turbine in izkoristek. V nerotirajočih delih turbine razlika med totalnim tlakom na vstopu in izstopu pomeni izgube:

First, a numerical analysis of the spiral casing with stay vanes was performed. The computational domain was extended to the runner outlet, but the guide vanes and runner blades were not modeled. The grid consisted of 280 000 nodes (Fig. 1). The results were used as the inlet conditions for subsequent calculations. At the same time the flow-energy losses in the spiral casing were calculated.

A numerical analysis of the flow in the tandem cascade, the runner and the draft tube was made in three stages. The first stage was a separate analysis of the flow through the tandem cascade, the runner and the draft tube. The computational domain for the tandem cascade is the region between two stay vanes and three guide vanes and between the hub and crown, but without runner blades. The domain consists of 114 000 nodes. The computational domain for the runner analysis is the region between two blades, it consists of 72 000 nodes. In the draft tube there are 170 000 nodes. The inlet conditions were obtained from numerical results of the upstream component. In order to minimize the influence of the inaccurate outlet boundary conditions the computational domains overlapped. The second stage was a partly coupled analysis. Flow through the stay vanes, the guide vanes and the runner were calculated simultaneously (166 000 nodes), while the draft tube was analysed separately (170 000 nodes). Finally, the flow from the stay-vanes inlet to the draft-tube outlet was analysed simultaneously. The grid consisted of 332 000 nodes (Fig. 2). The grids for the coupled analysis were obtained by attaching the grids of separate analyses and omitting the parts which overlapped, so the grid structure and grid density were the same for the separate and coupled analyses.

From the numerical results the flow-energy losses, the torque on the shaft and the efficiency can be calculated. Flow-energy losses in the non-rotating turbine parts are calculated as the difference between the total pressure at the domain inlet and outlet

$$\Delta E = \frac{1}{\rho.Q}\left(\int_{S_1} p_{tot} v_t dS - \int_{S_2} p_{tot} v_t dS\right) \quad (1),$$

pri čemer je

where

$$p_{tot} = \frac{1}{2}\rho.v^2 + p \quad (2),$$

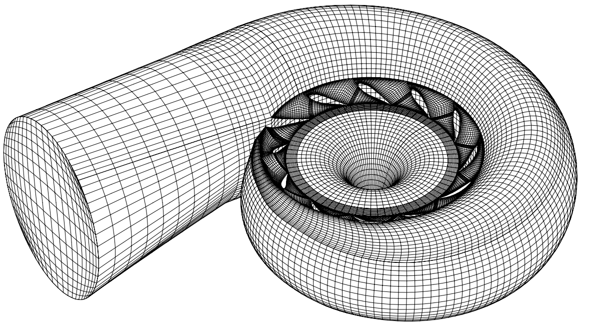
v_t je transportna komponenta hitrosti, S_1 in S_2 pa vstopni in izstopni prerez. Če ΔE delimo s težnostnim pospeškom g, dobimo izgube, izražene kot del padca, ki ni bil izkoriščen. V gonilniku večino razlike v totalnem tlaku pomeni delo gonilnika, majhen del pa izgube v toku. Izkoristek gonilnika izračunamo po obrazcu:

v_t is transport velocity component, S_1 and S_2 are the inlet and outlet cross-sections. If ΔE is divided by the acceleration due to gravity g, the flow-energy losses can be expressed as a head, which was not utilized. In the runner most of the difference in total pressure is converted to runner work, while a small part represents flow-energy losses. The turbine efficiency can be calculated by:

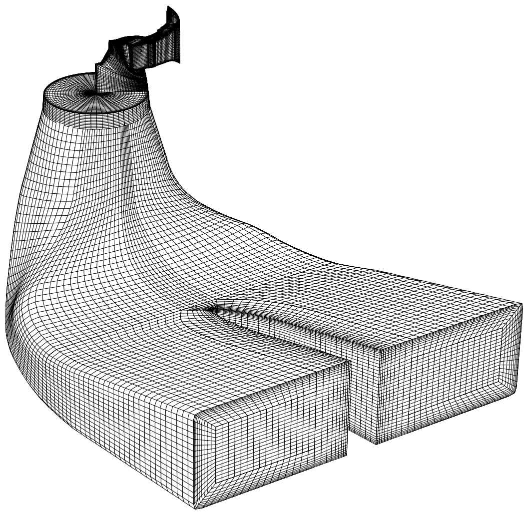
$$\eta = \frac{M.\omega}{\rho.Q.\Delta E} \quad (3),$$

pri čemer je M navor na os turbine, ω pa kotna hitrost. V primeru ločene numerične analize dobimo ΔE kot vsoto prispevkov posameznih komponent.

where M is the torque on the shaft, and ω is the angular velocity. In the case of a separate numerical analysis ΔE is obtained as the sum of the contributions of all the turbine parts.



Sl. 1. *Območje računanja in mreža za izračun toka v spirali*
Fig. 1. *Computational domain and grid for flow analysis in the spiral casing*



Sl. 2. *Območje računanja pri skupnem izračunu toka skozi predvodilnik, vodilnik, gonilnik in sesalno cev*
Fig. 2. *Computational domain for simultaneous calculation of the flow through the stay and guide vanes, the runner and the draft tube*

Podroben prikaz rezultatov ločene, delno sklopljene in sklopljene analize je prikazan v [5]. Pokazalo se je, da ločena analiza toka napove prevelike izgube v vseh delih turbine, izračunani navor na os turbine pa je skoraj enak pri ločeni, delno sklopljeni in sklopljeni analizi toka. Zato ločena analiza toka napove bistveno manjši izkoristek, kakor je bil izmerjen. Tudi lega optimalne točke obratovanja je pomaknjena k večjemu pretoku. Rezultati delno sklopljenega izračuna so nekoliko bliže izmerjenim. Oblika diagrama izkoristka, dobljenega s sklopljenim izračunom, se dobro ujema z izmerjenim, vrednosti izkoristka pa so za okoli 3% manjše od izmerjenih (sl. 3).

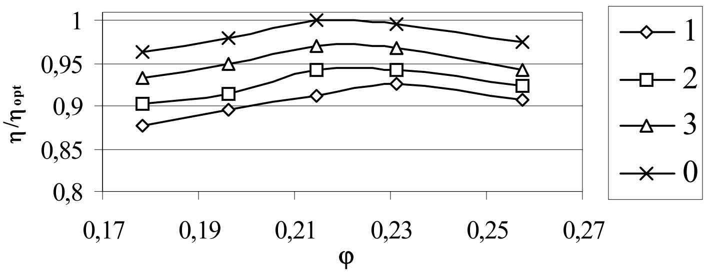
2 ŠKOLJČNI DIAGRAM IZKORISTKA

Na podlagi rezultatov za nominalno tlačno število smo ugotovili, da je le sklopljena analiza primerna za izračun izkoristka turbine. Zato smo s sklopljeno analizo izračunali izkoristek v naslednjih

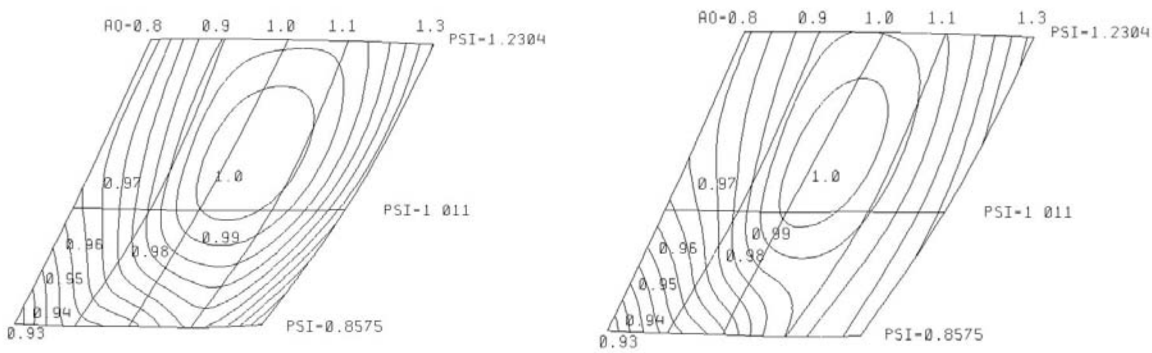
A detailed comparison of the results of the separate, the partly coupled and the coupled analysis is presented in [5]. We found that the separate flow analysis overestimates the flow-energy losses in all the turbine parts, while the calculated torque on the shaft is nearly the same for the separated, the partly coupled and the coupled calculations. In other words, the separate analysis predicts a much lower efficiency than the measured value. Also, the position of the best-efficiency point is shifted to a higher discharge. The results of the partly coupled calculation are closer to the measured values. The shape of the efficiency curve obtained with the coupled analysis is in good agreement with the measured value, but the calculated efficiency is about 3% lower than the measured values (Fig. 3).

2 HILL-CHART DIAGRAM

On the basis of the results for the nominal pressure coefficient it was concluded that only the coupled analysis is suitable for the prediction of turbine efficiency. Therefore, only the coupled analysis was



Sl. 3. *Diagram izkoristka za nominalno tlačno število ψ=1,011*
1 - ločen izračun, 2 - delno sklopljen izračun, 3 - sklopljen izračun, 0 - meritve
Fig. 3. *Efficiency diagram for the nominal pressure coefficient ψ=1.011*
1 - separated calculation, 2 - partly coupled calculation, 3 - coupled calculation, 0 - measurement



Sl. 4. *Školjčna diagrama izkoristka na temelju izračunanih in izmerjenih vrednosti v 15 točkah*
Fig. 4. *Hill-chart efficiency diagram based on the calculated and measured efficiency at 15 points*

desetih točkah obratovanja, pet za nižje in pet za višje tlačno število. Izračunane in izmerjene krivulje izkoristka se po obliki dobro ujemajo, izračunani izkoristek je za okoli 3% manjši, le pri velikem pretočnem številu in majhnem tlačnem številu (φ=0,2336, ψ=0,8575), smo dobili odstopanje okoli 5%.

Na temelju izračunanega izkoristka v 15 obratovalnih točkah narišemo školjčni diagram izkoristka. Za primerjavo narišemo še diagram na podlagi izmerjenega izkoristka v 15 obratovalnih točkah (sl. 4). Numerične vrednosti izkoristka so deljene z največjim izračunanim izkoristkom, izmerjene vrednosti pa z največjim izmerjenim izkoristkom. Obravnavane točke obratovanja so na presečiščih krivulj, ki pomenijo nespremenljivo odprtje vodilnika in vodoravnih črt, ki pomenijo stalen ψ. Lega optimalne točke obratovanja se dobro ujema z meritvami, prav tako tudi oblika krivulj s stalnim izkoristkom. Izračunani diagram je v okolici optimalne točke obratovanja nekoliko bolj položen, dlje od optimalne točke obratovanja pa bolj strmo pade kakor izmerjeni. Razlika je največja v desnem spodnjem delu diagrama, zaradi večjega odstopanja med izmerjenim in izračunanim izkoristkom pri φ=0,2336, ψ=0,8575.

made for the additional ten operating points, five for the lower and five for the higher pressure coefficient. The calculated and measured efficiency curves have the same shape, however, the calculated efficiency is about 3% lower. The exception is the operating point, with a large discharge coefficient and a small pressure coefficient (φ=0.2336, ψ=0.8575), where the discrepancy is 5%.

On the basis of the calculated efficiency for the 15 operating points a hill-chart diagram was drawn. For comparison, a diagram based on the measured efficiency at 15 operating points was also drawn (Fig. 4). The numerically obtained values were divided by the highest calculated efficiency, while the experimental values were divided by the highest measured efficiency. The treated operating points were at the cross-sections of the curves of constant guide-vane opening (A_0) and the lines of constant ψ. The position of the best-efficiency point was quite accurately predicted. The shape of the efficiency contours was also in quite good agreement. Near the best-efficiency point the calculated diagram is flatter than the measured one, but further from the best-efficiency point the efficiency decreases quickly. The descrepancy is largest at the bottom right-hand part of the diagram, because of the larger disagreement between the measured and calculated efficiency at point φ=0.2336, ψ=0.8575.

3 VPLIV GOSTOTE MREŽE IN TURBULENTNIH MODELOV NA REZULTATE

Diagram izkoristka za nominalno tlačno število je bil dobljen iz rezultatov numerične analize, izvedene s standardnim modelom k-ε na precej redki mreži. Da bi preučili vpliv gostote mreže in turbulentnega modela na rezultate, je bila ločena analiza toka v optimalni točki obratovanja narejena z različnimi turbulentnimi modeli na redki in zgoščeni mreži.

Porazdelitve tlaka in hitrosti, dobljene na mrežah različne gostote, so kakovostno zelo podobne. Pri redkih mrežah so energijske izgube v toku večje predvsem na račun precenjenih izgub zaradi trenja na stenah. Z zgostitvijo mrež se izgube zmanjšajo, zlasti v dvojni kaskadi in v gonilniku, medtem ko je vpliv zgostitve mreže v sesalni cevi manjši. Vpliv zgostitve mreže je enak za vse uporabljene turbulentne modele.

3.1 Turbulentni modeli

Pri izbiri turbulentnih modelov smo se omejili na dvoenačbna modela k-ε in k-ω. Poleg standardnega modela k-ε smo računali tudi z modelom RNG. Ta model je dobljen s teorijo renormalizacijskih grup (RNG), uporabljeni na Navier-Stokesovih enačbah. Transportni enačbi za k in ε sta enaki kakor pri standardnem modelu k-ε, razlikujejo se le koeficienti, s katerimi sklenemo sistem [6].

Turbulentni model k-ω je bil razvit z namenom, da bi bolj natančno napovedali odlepljanje toka na gladkih stenah. V CFX-TASCflow so vključeni trije modeli k-ω: standardni Wilcoxov model k-ω [7], model BSL (Baseline model) in model SST (Shear Stress Transport model). Standardni model k-ω je zelo občutljiv za vstopne pogoje za ω. Model BSL skuša ohraniti prednosti modelov k-ε in k-ω. Pri tem modelu je Wilcoxov model pomnožen s funkcijo F_1, model k-ε pa je najprej transformiran v obliko k-ω, nato pa pomnožen s $(1-F_1)$. F_1 je definirana tako, da imamo zunaj mejne plasti standardni model k-ε, ob steni pa preidemo na model k-ω [8]. Model SST upošteva prenos turbulentnih strižnih napetosti in najbolje popiše odlepljanje toka na stenah [9].

Eden od problemov dvoenačbnih modelov je obnašanje v okolici zastojnih točk. Pogosto opazimo pred zastojnimi točkami zelo visok nivo turbulence, ki se nato porazdeli okoli telesa. Problem sta rešila Kato in Launder s spremembo produkcijskega člena v enačbi za turbulentno kinetično energijo [10].

Pri turbulentnem modelu k-ε tok ob stenah najpogosteje modeliramo s standardnimi stenskimi funkcijami z logaritmičnim profilom. Da bi se izognili nedoslednosti pri zelo gostih mrežah, so razvili stenske funkcije s fiksnim y^+ [11]. Pri modelu k-ω je

3 THE EFFECT OF GRID DENSITY AND TURBULENCE MODELS ON THE RESULTS

The efficiency diagram for a nominal pressure coefficent was obtained from the results of a numerical analysis performed by the standard k-ε model on coarse grids. In order to study the effect of the grid density and the turbulence model on the results, a separate numerical analysis at the best-efficiency point (BEP) was performed using different turbulence models for the coarse and refined grids.

The pressure and velocity distribution obtained for grids of different density are qualitatively similar. For the case of coarse grids the flow-energy losses are too high, mostly due to overprediction of the friction losses. With grid refinement the flow-energy losses decrease, especially in the tandem cascade and the runner, while in the draft tube the effect is small. The same effect of grid refinement was obtained with all the turbulence models.

3.1 Turbulence models

The calculations were made with two-equational models k-ε and k-ω. Besides the standard k-ε model the RNG model was also used. This model is obtained from Renormalized Group Theory applied to Navier-Stokes equations. The transport equations for k and ε are the same as for the case of the standard k-ε model, but the closure coefficients are different [6].

The k-ω turbulence model was developed to predict the onset of separation on a smooth surface more accurately. In CFX-TASCflow three k-ω models are available: the standard Wilcox model [7], the BSL (Baseline) model and the SST model (Shear Stress Transport model). The standard k-ω model is very sensitive to the inlet conditions for ω. The BSL model combines the advantages of the k-ε and k-ω models. The Wilcox model is multiplied by a blending function F_1. The k-ε model is at first transformed to the k-ω formulation and then multiplied by $(1-F_1)$. F_1 is defined in such a way that outside the boundary layer the standard k-ε model is used, while inside the boundary layer the k-ω model is used [8]. The SST model accounts for the transport of the turbulent shear stress and therefore predicts the separation most accurately [9].

One of the problems with the two- equational models is the behavior near stagnation points. It is frequently observed that very high turbulence levels are predicted upstream of a stagnation point and then transformed around the body. This problem was solved by Kato and Launder, who changed the production term in the equation for the turbulent kinetic energy [10].

When the k-ε turbulence model is used, the flow in the near-wall region can be modeled by standard log-law wall functions. To avoid inconsistencies for the case of fine grids, fixed y^+ wall functions were developed [11]. For the k-ω model problem nedoslednosti rešen s formulacijo, ki pri zgoščenih mrežah avtomatično preide iz stenskih funkcij za visoka Re števila na model za nizka Re števila [12].

V CFX-TASCflow je vključen tudi dvoslojni turbulentni model. Pri tem modelu tok dovolj stran od sten modeliramo z modelom k-ε, tok ob stenah pa z enoenačbnim modelom. Za visoka Reynoldsova števila mora biti ob stenah mreža zelo zgoščena [12]. Zaradi premajhnih računalniških zmogljivosti tega modela nismo uporabili.

3.2 Izračun toka z različnimi turbulentnimi modeli

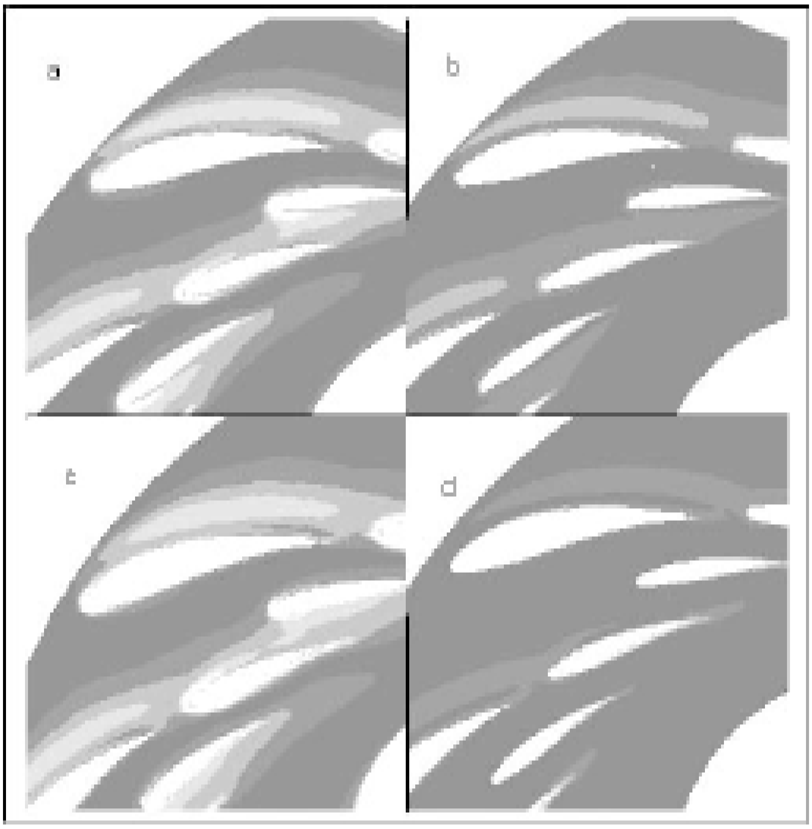
Tok v dvojni kaskadi smo računali s standardnim in z modelom RNG k-ε. Z modelom RNG dobimo nekoliko manjše energijske izgube v toku kakor s standardnim modelom. Izračun je bil ponovljen s standardnima modeloma k-ω in SST k-ω. S standardnim modelom k-ω dobimo predvsem zaradi nekoliko večje sence za lopaticani za malenkost večje izgube kakor s standardnim modelom k-ε. Na sliki 5 je prikazana porazdelitev turbulentne kinetične energije, dobljene z različnimi modeli na gosti mreži. S standardnima modeloma k-ε (sl. 5a) in k-ω (sl. 5c) modeloma dobimo veliko

the problem of inconsistencies in the case of fine grids is solved by a formulation which automatically switches from wall functions to a low-Re near-wall formulations as the grid is refined [12].

In CFX-TASCflow a two-layer turbulence model is also available. The standard k-ε model is used away from the wall, while the one-equation model is used near the wall. For high Reynolds numbers a very fine grid near the walls is required [12]. Due to insufficient computer capacity, we did not use this model.

3.2 Flow calculation with different turbulence models

Flow in the tandem cascade was calculated with the standard and the RNG k-ε models. With the RNG model, smaller flow-energy losses were obtained than with the standard k-ε model. The calculation was repeated with standard k-ω and SST k-ω models. With the standard k-ω model, due to larger wakes behind the vanes, slightly larger flow-energy losses were obtained than with the standard k-ε model. The distribution of turbulent kinetic energy, obtained with the different models on the refined grid is presented in Fig. 5. With the standard k-ε (Fig. 5a) and the standard k-ω (Fig. 5c) models, an increase in the turbulent kinetic energy near the stagnation



Sl. 5. *Porazdelitev turbulentne kinetične energije v dvojni kaskadi*
Fig. 5. *Distribution of the turbulent kinetic energy in the tandem cascade*
a – standardni model k-ε / standard k-ε model, b - model k-ε, Kato-Launder , c – standardni model k-ω / standard k-ω model, d – model SST k-ω, Kato-Launder

zvečanje turbulentne kinetične energije v okolici zastojnih točk in okoli lopatic. Pri obeh modelih s Kato-Launderjevim produkcijskim členom dobimo veliko bolj enakomerno porazdelitev turbulentne kinetične energije (sl. 5b in 5d). Zato dobimo tudi manjše izgube, in sicer pri modelu k-ε na redki mreži za 2,5%, na gosti mreži pa za 4%, pri modelu k-ω pa na redki mreži za 4%, na gosti mreži pa za 9,7%. Najmanjše izgube dobimo z modelom SST k-ω, tam je tudi porazdelitev turbulentne kinetične energije najbolj enakomerna (sl. 5d). Pri modelu k-ε se izgube nekoliko zmanjšajo še z uporabo stenskih funkcij s fiksnim y^+.

Pri izračunu toka v gonilniku smo primerjali navor na os turbine in izkoristek gonilnika. Razlike v navoru so majhne, pod 0,35%. Izkoristek gonilnika je najmanjši pri standardnem modelu k-ε in največji pri modelu SST k-ω. Z zgostitvijo mreže se pri vseh modelih izkoristek poveča za približno 1%.

Tok v sesalni cevi smo računali z obema dvoenačbnima modeloma. Primerjali smo izgube in koeficient rekuperacije tlaka C_p. Koeficient rekuperacije tlaka predstavlja razmerje med razliko tlaka na izstopu in vstopu sesalne cevi in kinetično energijo na vstopu. Definiran je z enačbo:

points and around the vanes can be observed. When using the Kato-Launder production term the distribution of turbulent kinetic energy is much more uniform (Fig. 5b and 5d). This results in a reduction of the flow-energy losses in the case of the k-ε model by 2.5% on the coarse grid and 4% on the refined grid, while in the case of the k-ω model the reduction was 4% on the coarse grid and 9.7% on the refined grid. The smallest flow-energy losses were obtained with the SST k-ω model, where the distribution of turbulent kinetic energy was also the most uniform. In the case of the k-ε model some reduction in the flow-energy losses was also obtained by the use of wall functions with a fixed y^+.

As a result of the runner analysis, the torque on the shaft and the runner efficiency were compared. The difference in torque on the shaft is small, less than 0.35%. The runner efficiency is the smallest in the case of the standard k-ε model and the largest in the case of the k-ω SST model. With grid refinement, the runner efficiency increases by approximately 1%.

The flow in the draft tube was calculated with both two-equational models. The flow-energy losses and the coefficient of pressure recovery (C_p) were compared. C_p represents the ratio of the difference in pressure at the draft-tube outlet and inlet and the kinetic energy at draft tube inlet. C_p is defined by the equation:

$$C_p = \frac{\int_{S_2} p v_t dS - \int_{S_1} p v_t dS}{\frac{\rho}{2} \int v^2 v_t \, dS} \tag{5}$$

pri čemer je S_1 vstopni, S_2 pa izstopni prerez sesalne cevi, p je tlak, v_t je transportna komponenta, v pa absolutna hitrost. Z modelom k-ε dobimo manjše izgube in višji C_p kakor z modelom k-ω. S Kato-

where S_1 and S_2 are the inlet and outlet draft-tube cross-sections, respectively, p is the pressure, v_t the transport velocity component, v the absolute velocity. The flow-energy losses obtained with the k-ε model are smaller

Preglednica 1. *Energijske izgube v toku v kaskadi, dobljene z različnimi turbulentnimi modeli na redki in gosti mreži*
Table 1. *Flow-energy losses in the tandem cascade obtained by several turbulence models on coarse and refined grids*
A - standardne log. stenske funkcije / standard log.-law wall functions
B - stenske funkcije s fiksnim y^+ / fixed y^+ wall functions
C - kombinacija stenskih funkcij za nizka in visoka Re števila / combined low and high Re wall functions

Turbulentni model Turbulence model	Model za tok ob steni Near-wall model	$\Delta E/g$ m 113 580 vozlov 113 580 nodes	$\Delta E/g$ m 374 130 vozlov 374 130 nodes
k-ε, standardni model /k-ε, standard model	A	0,4289	0,3728
k-ε, standardni model /k-ε, standard model	B	0,4114	0,3616
k-ε, Kato-Launder	A	0,4184	0,3578
k-ε, Kato-Launder	B	0,4054	0,3450
k-ε, RNG	A	0,4135	0,3590
k-ε, RNG, Kato-Launder	A	0,4023	0,3547
k-ε, RNG, Kato-Launder	B	0,4023	0,3662
k-ω, standardni model /k-ω, standard model	C	0,4338	0,3828
k-ω, Kato-Launder	C	0,4163	0,3455
k-ω, Kato-Launder, SST	C	0,4033	0,3087

Preglednica 2. *Navor na os turbine in izkoristek gonilnika, dobljena z različnimi turbulentnimi modeli na redki in zgoščeni mreži*
Table 2. *Torque on the shaft and runner efficiency obtained by several turbulence models on coarse and refined grids*
A - standardne log. stenske funkcije / standard log.-law wall functions
B - stenske funkcije s fiksnim y^+ / fixed y^+ wall functions
C - kombinacija stenskih funkcij za nizka in visoka Re števila / combined low and high Re wall functions

Turbulentni model Turbulence model	Model za tok ob steni Near-wall model	72 000 vozlov, 72 000 nodes		243 000 vozlov, 243 000 nodes	
		M Nm	η %	M Nm	η %
k-ε, standardni model /k-ε, standard model	A	368,84	90,49	371,87	91,61
k-ε, Kato-Launder	A	368,98	90,72	372,12	91,83
k-ε, RNG, Kato-Launder	A	369,03	90,67	372,26	91,82
k-ε, RNG, Kato-Launder	B	369,10	90,67	372,53	91,85
k-ω, standardni model /k-ω, standard model	C	369,99	90,88	372,75	91,89
k-ω, Kato-Launder	C	370,08	91,03	373,03	92,04
k-ω, Kato-Launder, SST	C	370,15	91,13	373,11	92,14

Preglednica 3. *Energijske izgube v toku in koeficient rekuperacije tlaka v sesalni cevi, dobljene z različnimi turbulentnimi modeli na redki in zgoščeni mreži*
Table 3. *Flow-energy losses and coefficient of pressure recovery in the draft tube obtained by several turbulence models on coarse and refined grids*
A - standardne log. stenske funkcije / standard log.-law wall functions
C - kombinacija stenskih funkcij za nizka in visoka Re števila / combined low and high Re wall functions

Turbulentni model Turbulence model	Model za tok ob steni Near-wall model	170 000 vozlov 170 000 nodes		452 000 vozlov 452 000 nodes	
		$\Delta E/g$ m	C_p %	$\Delta E/g$ m	C_p %
k-ε, standardni model, k-ε, standard model	A	0,257	51,02	0,251	51,9
k-ε, Kato-Launder	A	0,2528	52,02	0,2421	53,43
k-ε, RNG, Kato-Launder	A	0,2667	51,21	0,2689	50,39
k-ω, standardni model /k-ω, standard model	C	0,2616	49,31	0,2535	51,58
k-ω, Kato-Launder	C	0,2665	49,53	0,2588	50,33
k-ω, Kato-Launder, SST	C	0,272	47,52	0,273	48,37

Preglednica 4. *Izkoristek turbine v optimalni točki obratovanja za različne turbulentne modele in za dve gostoti mrež*
Table 4. *Turbine efficiency at the best-efficiency point for different turbulence models and for coarse and refined grids*
A - standardne log. stenske funkcije / standard log.-law wall functions
C - kombinacija stenskih funkcij za nizka in visoka Re števila / combined low and high Re wall functions

Turbulentni model Turbulence model	Model za tok ob steni Near-wall model	η/η_{opt} redke mreže coarse grids	η/η_{opt} goste mreže refined grids
k-ε, standardni model / k-ε, standard model	A	0,9126	0,9292
k-ε, Kato-Launder	A	0,9159	0,9332
k-ε, RNG, Kato-Launder	A	0,9146	0,9313
k-ω	C	0,9158	0,9311
k-ω, Kato-Launder	C	0,9182	0,9349
k-ω, Kato-Launder, SST	C	0,9196	0,9376

Launderjevim popravkom dobimo pri modelu k-ε manjše izgube, pri modelu k-ω pa se izgube celo povečajo. Z modelom k-ω SST dobimo največje izgube in najnižji C_p.

Z upoštevanjem izgub v posameznih delih turbine in navora na os turbine smo za oba dvoenačbna modela na redki in gosti mreži izračunali izkoristek turbine. Izkoristek, deljen z izmerjenim izkoristkom v optimalni točki obratovanja, je prikazan v preglednici 4. Pri vseh modelih se z zgoščanjem mreže izkoristek poveča za od 1,5 do 1,7%. Največji izkoristek dobimo z modelom k-ω SST, najmanjšega pa s standardnim modelom k-ε.

4 IZRAČUN TOKA V TURBINI BREZ PODATKOV IZ MERITEV

Pri vseh do sedaj prikazanih rezultatih je bil pretok pri določenem odprtju vodilnika in vrtljajih podatek, dobljen iz meritev. Kadar pa razvijamo novo turbino, tega podatka nimamo. Poznamo le razpoložljivi padec in vrtljaje, pri katerih bo turbina obratovala. Zato se v tem primeru naloge lotimo drugače. Na vstopu v turbino podamo le smer toka in totalni tlak, na izstopu pa statični tlak. Med izračunom se oblikuje pretok, ki ustreza dani tlačni razliki. Če računamo spiralo posebej, je postopek nekoliko bolj zapleten, saj ne vemo, kolikšen del tlačne razlike pomenijo izgube v spirali in kolikšen del odpade na preostalo turbino. Postopek postane iterativen. Za prvi približek vzamemo, da izgube v spirali pomenijo 1% celotnega razpoložljivega padca. Vstopne in izstopne robne pogoje za preostali del turbine predpišemo tako, da ustrezajo preostalim 99% razpoložljivega padca. Kot rezultat numerične analize toka dobimo neki pretok. Za ta pretok izračunamo tok v spirali in izgube v njej. Seštejemo tlačno razliko v spirali in v preostali turbini. Če se ta seštevek razlikuje od razpoložljivega padca, ustrezno popravimo vstopne in izstopne pogoje za izračun toka od predvodilnika do izstopa iz sesalne cevi. Po nekaj korakih se v okviru predpisane natančnosti približamo razpoložljivemu padcu. Ker se izgube v spirali večajo s kvadratom pretoka, tok v spirali računamo le enkrat, nato pa le še preračunamo izgube, ki ustrezajo novemu pretoku.

Tako smo izračunali tok v petih obratovalnih točkah pri nominalnem padcu. Računali smo z modelom k-ε s Kato-Launderjevim popravkom. Primerjava med izmerjenim in izračunanim pretokom je prikazana na sliki 6. Pri bolj zaprtem vodilniku je izračunani pretok nekoliko manjši od izmerjenega, pri večjih odprtjih vodilnika pa večji od izmerjenega. Največje odstopanje je

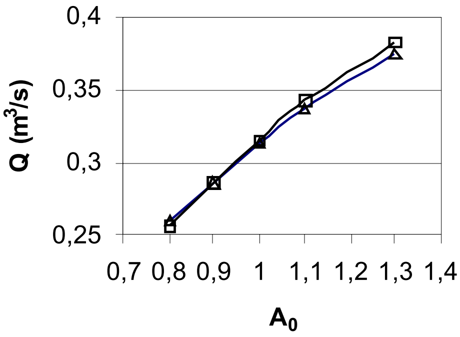
than those obtained by the k-ω model. With the Kato-Launder change of production term the flow-energy losses in the case of the k-ε model are reduced, while in case of the k-ω model they increase. With the k-ω SST model the largest flow-energy losses and the smallest C_p are obtained.

From the flow-energy losses in all the turbine parts and from the torque on the shaft, the turbine efficiency was calculated. The efficiency values, divided by the measured efficiency at the best-efficiency point, are presented in Table 4. For all the turbulence models used the efficiency obtained on the refined grid is from 1.5% to 1.7% higher than those obtained on the coarse grids. The highiest efficiency is obtained with the k-ω SST model and the smallest with the standard k-ε model.

4 ANALYSIS OF THE FLOW IN A TURBINE WITHOUT MEASURED DATA

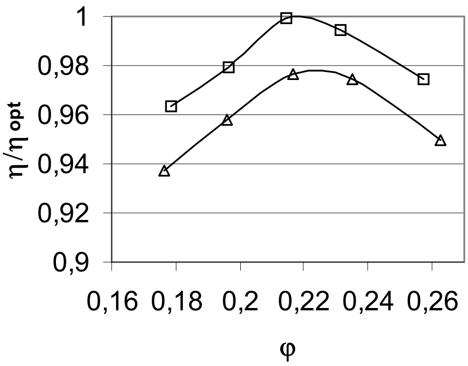
All the results presented so far were obtained from calculations where the discharge coresponding to a certain guide-vane opening and speed was obtained from measurements. When a new turbine is being developed, the discharge for a certain operating point is not known: we know only the avaible head and speed. Therefore, we have to solve the problem in a different way. At the turbine inlet the direction of the flow and the total pressure is prescribed, while at the outlet, the static pressure is prescribed. During the calculation, the discharge, which corresponds to the difference in pressure, is calculated. When the spiral casing is calculated separately, the procedure is a bit more complicated, because we do not know, how much of the head coresponds to the spiral casing and how much to the other turbine parts. The procedure becomes iterative. For the first approximation it can be assumed that the flow-energy losses in the spiral casing are 1% of the avaible head. Then we prescribe the inlet and outlet boundary conditions for the other parts of the turbine in such a way that the difference in the total pressure corresponds to 99% of the avaible head. A discharge is obtained as a result of a numerical analysis. For this discharge flow analysis of the spiral casing is made and we sum up the difference in the pressure in the spiral casing and in the other turbine parts. If the sum is not equal to the avaible head, we change the boundary conditions for the analysis of the flow from the stay-vane inlet to the draft-tube outlet. In a few steps, we obtain the discharge for which the calculated head is equal to the avaible head. The flow-energy losses in the spiral casing increase quadratically with the discharge, therefore, the flow in the spiral casing is calculated only once. For a new discharge only the flow-energy losses are recalculated.

In this way the flow at five operating points for the nominal head was calculated. The calculation was made with the Kato-Launder k-ε model. The calculated and measured discharge are compared in Fig. 6. For small guide-vane openings the calculated discharge is smaller than the measured one, while for large guide-vane openings it is larger. The largest descrepancy is 1.9%. The calculated efficiency is



□ - izračun - calculation
Δ - meritve – measurement

Sl. 6. *Izmerjeni in izračunani pretok pri dani geometrijski obliki, padcu in vrtljajih*
Fig. 6. *Measured and calculated discharge for a given geometry, head and speed*



□ - izračun - calculation
Δ - meritve – measurement

Sl. 7. *Izračunani in izmerjeni izkoristek pri dani geometrijski obliki, padcu in vrtljajih*
Fig. 7. *Calculated and measured efficiency for a given geometry, head and speed*

1,9%. Izračunani izkoristek je za okoli 2,5% manjši od izmerjenega, oblika krivulje se dobro ujema z meritvami (sl. 7).

5 SKUPNI IZRAČUN TOKA V CELOTNI TURBINI

Pri dosedanjih izračunih smo računali spiralo posebej, pri kaskadah predvodilnika, vodilnika in gonilnika pa smo se omejili na en periodični del. S tem smo predpostavili, da je tok med poljubnima lopaticama enak. To zlasti v primeru predvodilnika ne drži. V obravnavanem primeru celo lopatice predvodilnika niso enake, ampak se njihova dolžina manjša od vstopnega dela proti ostrogi spirale. Da bi ugotovili, kolikšen je vpliv ločenega izračuna toka v spirali in vpliv omejitve kaskad na en periodični del, smo izračunali tok v celotni turbini od vstopa v spiralo do izstopa iz sesalne cevi z vsemi predvodilnimi, vodilnimi in gonilnimi lopaticami. Zaradi premajhnih računalniških zmogljivosti je uporabljena računska mreža zelo redka, ima okoli 510 000 vozlov. Za primerjavo rezultatov v optimalni točki obratovanja smo tudi ločen in sklopljen izračun ponovili na enako redkih mrežah. Primerjali smo izračunane izkoristke, deljene z izkoristkom v optimalni točki obratovanja. Z ločenim izračunom vsakega dela turbine smo dobili vrednost 0,851, z ločenim izračunom spirale in skupnim izračunom toka v preostalih delih 0,929, s skupnim izračunom toka v celotni turbini pa 0,9398. Izkoristek pri skupnem izračunu toka v celotni turbini je največji predvsem zaradi manjših izgub v kaskadah predvodilnika in vodilnika. Te izgube so se zmanjšale zaradi skupnega izračuna spirale in kaskade. Pri izračunih, pri katerih smo se omejili na en periodični del, smo vzeli najdaljšo predvodilno lopatico in smo tudi zato dobili prevelike izgube.

about 2.5% less than the measured one (Fig. 7). The shape of the calculated efficiency curve is in good agreement with the measurements.

5 COUPLED CALCULATION OF THE FLOW IN A COMPLETE TURBINE

In all the calculations so far, the flow in the spiral casing was calculated separately and the stay vane, the guide vane and the runner cascades were reduced to one periodical part. It was assumed that the flow between any two blades or vanes is equal. However, this is not true, especially for stay vanes. The stay vanes are not equal, their length decreases from the inlet part to the nose of the spiral casing. To see effect of a separate calculation of the spiral casing and the effect of the reduction of the cascades' domains to one periodical part, the flow in the whole turbine, from the spiral casing inlet to the draft-tube outlet with all the stay and guide vanes and all the runner blades was calculated simultaneously. Due to insufficient computer capacity the grid is very coarse, it consists of about 510 000 nodes. To compare the results at the best-efficiency point, the separate and coupled calculation was also repeated on equally coarse grids. The calculated values of efficiency, divided by the measured efficiency at the BEP were compared. By a separate analysis of each turbine part a value of 0.851 was obtained, by a separate analysis of the spiral casing and coupled analysis of the other turbine parts we got a value of 0.929, and by a simultaneous calculation of the whole turbine a value of 0.9398 was obtained. The efficiency was the highest for a simultaneous calculation of the flow in the whole turbine, mainly due to the smaller flow-energy losses in the stay- and guide-vane cascades. The reason for the smaller losses in the tandem cascade is the coupled calculation of the spiral casing. In the calculation where the domains were reduced to one periodical part the flow between the longest two stay vanes was calculated and that was also the reason for the too high flow-energy losses.

6 SKLEP

Iz prikazanih rezultatov lahko povzamemo, da z ločenim izračunom toka v spirali in s skupnim izračunom toka v dvojni kaskadi, gonilniku in sesalni cevi dovolj natančno napovemo lego optimalne točke obratovanja, tudi oblika diagrama izkoristka se dobro ujema z izmerjeno. Izračunani izkoristek je za okoli 3% manjši od izmerjenega, vendar se z zgostitvijo mrež izmerjenim rezultatom zelo približamo. Pri ločenem izračunu smo z zgostitvijo mrež in vključitvijo Kato-Launderjevega popravka v model k-ε dobili za 2% višji izkoristek. Enako izboljšanje lahko pričakujemo tudi pri sklopljenem izračunu na gosti mreži, ki pa zaradi premajhnih računalniških zmogljivosti ni bil izveden. Sklepamo, da se s sklopljenim izračunom na zgoščeni mreži s Kato Launderjevim modelom k-ε izmerjenemu izkoristku lahko približamo na 1%.

S skupnim izračunom toka v celotni turbini na zelo redki mreži smo dobili za 1% večji izkoristek kakor pri ločenem izračunu spirale in skupnem izračunu toka v preostalih delih. S tem smo pokazali, da je izračunani izkoristek v prejšnjih izračunih premajhen tudi zaradi ločenega izračuna toka v spirali in omejitve kaskad na en periodični del. Zato lahko bolj kakor na 1% natančne vrednosti izkoristka pričakujemo le z izračunom celotne turbine na zelo gosti mreži.

V postopku razvoja novih gonilnikov je pomembno, da tudi v primeru, ko ne poznamo pretoka pri danem odprtju vodilnika, tega lahko dokaj natančno izračunamo. Tudi v tem primeru se lega optimalne točke obratovanja in oblika diagrama izkoristka dobro ujemata z izmerjenimi rezultati.

6 CONCLUSION

From the results presented it can be concluded that by a separate analysis of the flow in a spiral casing and a coupled analysis in a tandem cascade, runner and draft tube, the position of the best-efficiency point and the shape of the efficiency diagram are accurately predicted. The calculated efficiency is about 3% lower than the measured one, but with grid refinement the calculated results approach the measured ones. With grid refinement and by including the Kato-Launder production term in the k-ε model the efficiency of the separate analysis increased by 2%. The same improvement can be expected for a coupled calculation on a refined grid. This calculation was not performed due to computer capacity. We would expect that the efficiency obtained with a coupled calculation on a refined grid would be within 1% of the measured value.

The efficiency obtained from a simultaneous calculation of the flow in a complete turbine with a very coarse grid is 1% higher than that obtained by a separate analysis of the spiral casing and a coupled analysis of the flow in the other turbine parts. This result shows that the calculated efficiency is also too low because of a separate analysis of the spiral casing and the reduction of cascades domains to one periodical part. Therefore, less than a 1% difference between the calculated and measured efficiency can be expected only with a coupled calculation of the whole turbine on a very fine grid.

In the design process it is very important that when the discharge coresponding to a certain guide-vane opening is not known, it can be calculated accurately enough. Also, in this case the position of the best-efficiency point and the shape of efficieny diagram is in good agreement with measured results.

7 LITERATURA
7 REFERENCES

[1] Jošt, D., A. Lipej, K. Oberdank, M. Jamnik, B. Velenšek (1996) Numerical flow analysis of a Kaplan turbine; *Hydraulic Machinery and Cavitation*, ed. E Cabrera, V. Espert, F. Martinez, Dortrecht: Kluver.

[2] Troha, J., M. Bajd, A. Oberdank, A. Lipej, D. Jošt (1997) Refurbishment and uprating hydro powerplants with model test; *Hydropower into the next Century*, Portorož, 1997; *The International Journal on Hydropower & Dams*; Sutton.

[3] Sick, M., M.V. Casey, P.F. Galpin (1996) Validation of a stage calculation in a Francis turbine; *Hydraulic Machinery and Cavitation*, ed. Cabrera, E et all, Vol. I.

[4] Jošt, D., L. Škerget (2000) Separate and coupled CFD simulation of a flow in a Francis turbine; *Hydro, Techology and Enviroment for New Century, Hydraulic Machinery and Systems*, 20th IAHR Symposyum, Charlotte, USA.

[5] Jošt, D., L. Škerget (1999) Numerična analiza toka v francisovi turbini, Kuhljevi dnevi 99, *Slovensko društvo za mehaniko.*

[6] Yakhot, V., S.A. Orszag, S. Tangham, T.B. Gatski, C.G. Speciale (1992) Development of turbulance models for shear flows by a double expansion technique; *Phys. Fluids*, Volume 7, 1510-1520.

[7] Wilcox, D. C. (1986) Multiscale model for turbulent flows; AIAA 24th Aerospace Science Meeting, *American Institute of Aeronautics and Astronautics*.

[8] Menter, F. R. (1993) Multiscale models for turbulent flows; 24th Fluid Dynamics Conference, *American Institute of Aeronautics and Astronautics*.
[9] Menter, F. R. (1996) A comparison of some recent eddy - viscosity turbulence models; *Journal of Fluids Engineering,* ASME, Vol. 118, 514-519.
[10] Kato, M., B.E. Launder (1993) The modelling of turbulent flow around stationary and vibrating square cylinder; *9th Symposium on Turbulent Shear Flows*, Kyoto, Japan, 10-4-1 - 10-4-6.
[11] Grotjans, H., F.R. Menter (1998) Wall functions for general application CFD codes; ed. Papailiou, *ECOMAS 98 Proceedings of the Fourth European Computational Fluid Dynamics Conference*, 1112-1117.
[12] CFX-TASCflow Computational Fluid Dynamics Software, Primer Documentation, Version 2.10.

Naslova avtorjev: mag. Dragica Jošt
Turboinstitut
Rovšnikova 7
1210 Ljubljana Šentvid

prof. dr. Leopold Škerget
Fakulteta za strojništvo
Univerza v Mariboru
Smetanova ulica 17
2000 Maribor

Authors' Addresses: Mag. Dragica Jošt
Turboinstitute
Rovšnikova 7
1210 Ljubljana Šentvid

Prof. Dr. Leopold Škerget
Faculty of Mechanical Eng.
University of Maribor
Smetanova ulica 17
2000 Maribor, Slovenia

Prejeto:
Received: 22.1.2001

Sprejeto:
Accepted: 27.6.2001