Proceedings of ASME Turbo Expo 2004
Power for Land, Sea, and Air
June 14-17, 2004, Vienna, Austria

GT2004-54228

FLOW PHENOMENON IN STEAM TURBINE DISK-STATOR CAVITIES CHANNELED BY BALANCE HOLES

L. Moroz
SoftInWay, Inc.
35 Corporate Dr., 4th floor, Burlington, MA 01803
lm@softinway.com

A. Tarasov
SoftInWay, Inc.
35 Corporate Dr., 4th floor, Burlington, MA 01803
tars@ua.softinway.com

ABSTRACT

Computational fluid dynamic (CFD) analysis of the region the secondary flow path of high pressure steam turbine was conducted. Region included two adjacent to disc cavities and section of main flow path. The cavities were channeled by balance holes located in the disk. Two geometrical models were considered: single disk cavity and both cavities with balance holes. 2D axi-symmetric and 3D analyses were conducted for the first model, however only 3D study was performed for the second one.

Analysis of the single disk cavity has revealed appearance of vortexes in circumferential direction even in case when all boundary conditions and geometry features were close to the axi-symmetric case. The trend of the average pressure distributions along radius was found for 2D and 3D models.

The transient analysis of the two cavity model revealed vortexes movement in circumferential direction with higher than disk rotation velocity. It induces periodic conditions at the inlet and outlet of balance hole and periodic steam mass flow rate through the holes. The flow pattern in the two cavities with balance holes is so complicated that common 1D net flow representation should be considered as significant simplification

INTRODUCTION

Steam/gas mass flow distributions in the secondary turbine flow path is usually treated with method replacing real hydraulic system with equivalent graph. Branches of such graph are 1D models of flow in elements like tubes, annulus, valves and so on. Existing 1D models are essentially correlation dependencies of hydraulic resistance upon various parameters.

Mass flow rate and pressure distributions in the whole hydraulic system of the secondary flow path are calculated using mass conservation law at each graph node and pressure drops on each branch.

The most of the correlations in simple cases are reliable for the wide range of operation and geometry parameters. The complicated flow such as flow in the disk-stator axial seals couldn't be treated with the same accuracy as simple ones, and unfortunately its influence on the whole secondary flow path is significant. In reality flow in the axial seal is substantially 3D, especially when adjacent to seal cavities are channeled with balance holes. In spite of that, often such flows are treated with simple methods when whole turbine secondary path is analyzed for the lack of more accurate methods.

Turbine design requires development of reliable and rapid methods of secondary flow path prediction. Such methods must be based on deep understanding of processes in axial disk-stator seals and should extract accurate 1D models applicable for turbine designer needs.

An alternative way is axi-symmetric CFD modeling of the turbine secondary flow path with several additional assumptions related to labyrinth seals, balance holes and others elements that induce 3D effect [1]. This approach promises more accurate results but several assumptions used in reduction of the problem from 3D to 2D axi-symmetric formulation need to be validated.

Flow in cavities between rotating disk and fixed stator is a subject of many investigations [2, 3, 4, 6, 7, 8] and remains a point of considerable interest due to a variety of disk cavity and rim seal geometries that are explored in current gas and steam turbines. Influence of geometrical details on the flow inside cavity is very strong and therefore slight variations in seal geometries lead to dramatic differences in flow patterns. That is why researchers often conduct specific design studies using general understanding of flow phenomenon. The latest studies [2, 3, 4] include numerical/experimental analyses and only a few old papers [5, 9] were devoted to developing simple methods that could be used for the conceptual design of steam/gas turbine secondary flow path

One of the challenging problems of the secondary flow path modeling is predicting of pump effect caused by rotating disk.

Known methods for calculating pump effect rely on the solution of momentum equations in 1D formulation evolved for fluid volume enclosed in between flat rotating and fixed disks [5, 9]. It is assumed that fluid in the boundary layer on the disk moves centrifugally, meanwhile it moves centripetally on the stator. Besides, disk and stator surfaces are supposed to be smooth that provides continuous flow. In practice, in order to meet the design and technology requirements, the disk/stator must have more complex surfaces (in gas turbines, especially); therefore, the flow will be substantially different from idealized one. Moreover, above mentioned approach doesn't include neither the seal at the disk's band, nor its shape, although it's known that these factors influence the flow dramatically. Steam migration through balance holes adds even more complexity to the situation.

This paper doesn't claim to provide a general solution. It describes flow modeling in near disk adjacent cavities of a steam turbine high pressure cylinder in 2D and 3D formulations, complemented with analysis of the differences and comparison with reduced-order modeling results.

The analysis has shown a very intricate flow pattern in considered system. Flow through the balance holes exhibit significant swirling and heterogeneity. Two kinds of stream were observed in the system: the first strong stream is directed from the cavity with higher pressure to the cavity with lower pressure. The second weaker stream runs in opposite direction, i.e. in the direction of adverse pressure gradient. It was revealed that balance holes generate near regular vortex structures that move in the cavities mainly circumferentially. The flow appeares transient and can be characterized as practically periodic one. Due to this reason, mass exchange between cavities and main flow path was also periodic, i.e. in the same band seal, main steam flow ingress was alternated in time with secondary steam flow injection into the turbine main flow path. Changes in other characteristics were also periodic in time. For instance, static pressure at inlet/outlet of balance hole changed periodically, so that pressure drop was either positive or negative. It is obvious that the periodic flow pattern dramatically differs from existing models. At the same time, 2D CFD analysis didn't demonstrate any instability that could be identified as periodicity which leads to noticeable difference from results of 3D CFD analysis.

The paper shows that an averaged in time flow pattern in the disc-rotor cavities couldn't be correctly determined by means of existing relevant equations used in a framework of net model.

NOMENCLATURE

A	Area or coefficient in Equation (5)
G	Mass flow rate
K	Rotation factor
Re	Reynolds number
d	Balance hole diameter
p	Pressure
r	Radius of the disk
s	Width of the cavity
u	Axial velocity
x	Reduced radius, $x = r / r_1$
v	Tangent velocity
z	Number of balance hole

Greek

β	Rotation factor, ratio of average tangent velocity of fluid to disk velocity at the current radius
ζ	Resistance factor
φ	Flow factor
ν	Kinematic viscosity

Subscripts

1	Hub of the disk
2	Rim of the disk
op	Operational pressure
rotation	Rotation case
stationary	Stationary case

3D MODEL

Model

An object of the study was steam flow in two adjacent near disk cavities of the first stage of a large steam turbine high pressure cylinder. All disks of the rotor have 7 balance holes of 45mm in diameter. It was possible to model only 1/7 of the cavity volume (Fig.1). Disk thickness was 50 mm. Average (reference) value of steam pressure was 16.3 MPa. Steam temperature was 750K and density 47-50 kg/m^3.

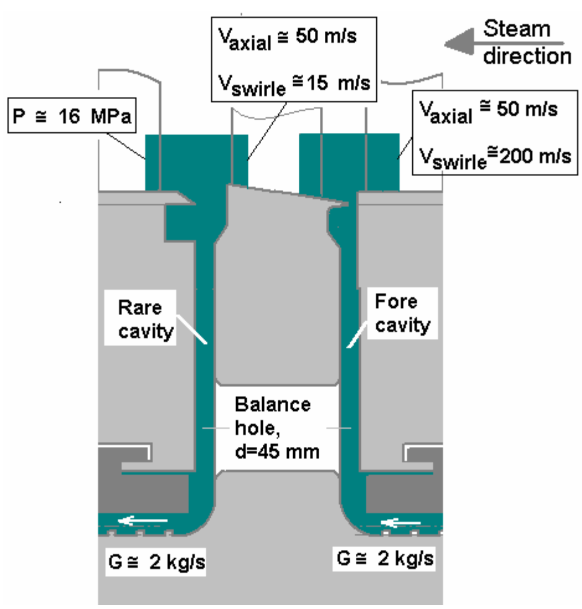

Figure 1. Model of the disk cavities (green sectors) chanelled with balance holes (white sector)

Analysis was performed by using FLUENT solver version 6.0. Grids were generated by Gambit. All models were meshed with boundary layer refinement near the walls

Grid study was conducted by varying the number of elements in 2D model by two times, and essentially it didn't change the results. Wall cell Y^+ for 2D model was in range of 40-500.

A full 3D model comprised of 224,230 hexahedral elements that included detailed meshing near wall regions (Fig. 2). The range of Y^+ was about 40-1000. Standard wall functions were used for treatment near wall region.

All computations were conducted according to the scheme of second-order approximation. The iterations were stopped when residuals of all quantities were approximately 10^{-5}.

Boundary conditions

Boundaries conditions were determined from existing results of modeling of steam turbine main and secondary flow path that were carried out with traditional engineering methods. Thus the pressure and velocities upstream and downstream blade's root zone were extracted from flow path prediction, Fig.1.

We had to tolerate some inaccuracies in assigning boundary conditions. For example, it is obviously that stationary nozzles and rotating blades intensify interaction between the main and the secondary steam flows. However, we had to omit these details in order to be able to run the problem on commercially available PC. Therefore, the flow path zone was simplified in the model and represented by hollow cylinders above the hub and the diaphragm, Fig. 1.

Boundary conditions before/after labyrinth seals

Surface **A,** Fig. 2, located upstream can be used for specification of inflow parameters and surface **F,** located downstream can be used for specification of outflow parameters. Boundary conditions on these surfaces shell be discussed in details. It seems that pressure (total/static) is the most appropriate kind of BC to assign, because pressure is known on each node of hydraulic net system of secondary flow path. However,, by doing so the pump (centrifugal) effect will be pre-determined. In reality we don't know how to estimate correctly the pump effect and study of which is one of the goals of this work. Therefore, the mass flow rate was assigned to both surfaces. This allowed studying pressure development under centrifugal forces. Values of inflow and outflow were about the same and equal to 2/7 kg/s, etc. ~1% of the main steam flow. (Full mass flow rate was 2 kg/s, only one sector, i.e. 1/7 of the flow path was modeled, thus 2/7 kg/s). This value was determined by the previous net calculation of secondary path. Besides mass flow rate at the surface **A,** a tangential velocity was assigned on that surface that was about 0.3-0.4 of rotor rotational speed.

Main steam flow boundary conditions

To mimic interaction between flows inside disc cavities and steam main flow there were created two zones extending into turbine flow path. Zones were sized in such a way to avoid direct influence of the location of boundaries on mass exchange between main/secondary flows.

Velocity components and steam temperature were assigned on **B** and **D** surfaces and static pressure was applied to **C** and **E** surfaces (Fig.1).

Tangential component of velocity at the surface **B** was ~200 m/s and practically axial flow was specified at the surface **D**. Axial component was about 50 m/s through both surfaces while radial component was null. The relative static pressure was set at the surfaces **C** and **E** pressure (taking into account radial effect.

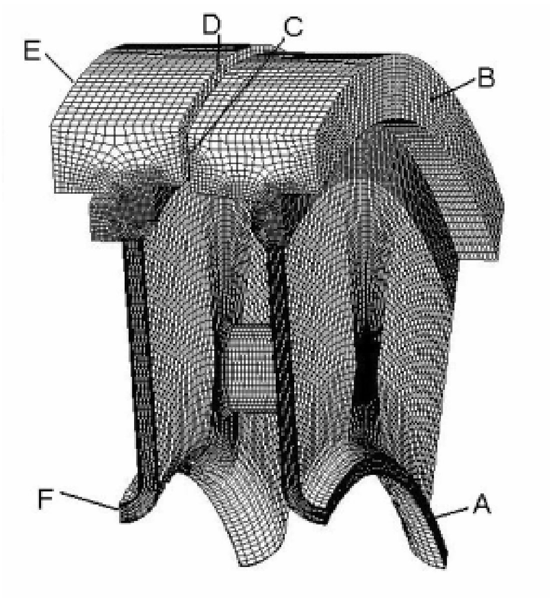

Figure 2. CFD model of near disk cavities

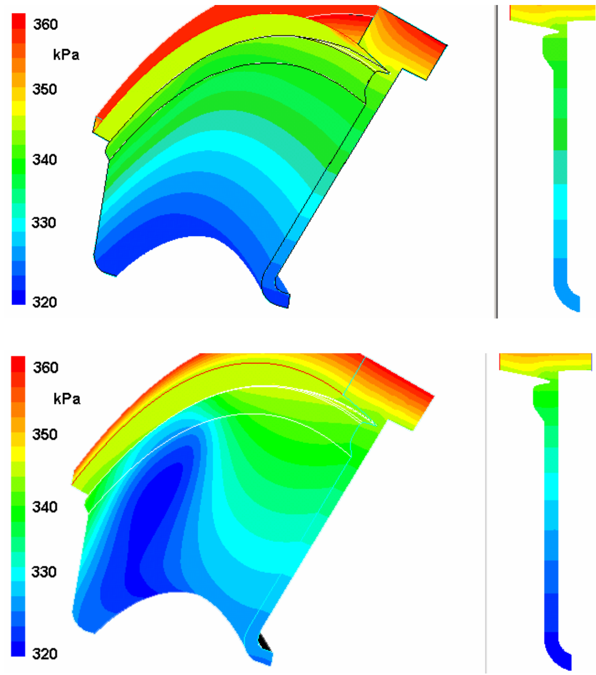

Figure 3. Pressure distribution with 3D (left) and 2D (right) models of the fore near-disk cavity. Above - P_{op}=1MPa , below P_{op}=16.3 MPa

Boundary conditions at the walls

"No slip" conditions were assigned at the all wall surfaces. The model rotated with angular velocity 314 rad/s; hence, rotor surfaces were stationary in relative motion, while stator surfaces were stationary in absolute motion.

Periodicity

Periodic boundary conditions were applied to surfaces of the segments cross-sections (Fig.2).

Single disk cavity

A single disk cavity without balance hole was taken as an object of investigation. The results of analysis allowed to draw some important conclusions with respect to further study of the full model.

Flow nature in the disk-stator systems is extremely unstable, especially, in the case of steam forced injection into the gaps from labyrinth seal of upstream diaphragm. At this point, steam moves predominantly from axis of symmetry to the disk rim under adverse pressure gradient. Small obstacles or great value of pressure gradient can cause flow separation that has substantially 3D nature. In other words, under certain conditions, the problem can be adequately treated only as 3D one.

It was a matter of interest to compare two kinds of models. With this goal, axi-symmetric 2D model and 3D model were developed for the single disk seal (only fore cavity of whole model shown on Fig.1). Boundary conditions were the same as in the whole model, however fluid was ideal gas (air). It allowed analysis for different values of operation pressure (air density). We studied flow with forced injection of air from the labyrinth seal and without it.

The computations showed unexpected results. At relatively small value of operation pressure (1 to 3 MPa), axi-symmetric computations completely coincided with 3D analysis results (Fig.3, above). Air moved circumferentially and its streamlines remained undistorted. At pressure above 3 MPa, the powerful vortex structures were generated. The process lost its stationery nature due to transition of those structures in tangential direction. At the same time, pressure distribution averaged in time along radius in 3D computations was almost similar to results of stationery 2D analysis. To avoid mistakes, these results were validated by numerous computations.

The following reasoning can serve as the foundation for understanding this phenomenon. Pressure growth leads to density increase; correspondingly, pump effect is intensified together with increasing of adverse pressure gradient. It promotes fluid separation around the rim and distortion of axis-symmetric character of the flow. As the result powerful vortexes appear and move along circumference of the seal.

Full model

Complete analysis of the flow in two near disk cavities channeled by balance hole was performed under reference pressure of 16.3 MPa only, i.e. for real conditions of steam turbine HPC. Computations were carried out for non-stationery formulation.

Transient analysis is very computationally expensive, nevertheless, corresponding computations have been conducted to cover the period of attainment of the solution's periodicity.

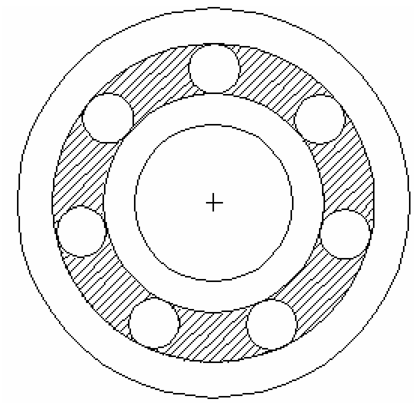

Fig. 4 Disk surface of averaging (hatched area)

Transient behavior of circumference vortex structure was revealed in the fore cavity that caused significant irregularity of static pressure. For instance, static pressure fluctuation of ±0.25MPa was observed at balance holes radius, meanwhile the difference of static pressure values averaged within the annulus that envelopes the balance holes (Fig.4) on both sides of the disk fluctuated approximately within the same range of values (Fig.5).

Fluctuation in pressure inside the cavities leads to periodic variation of steam flow rate through the balance holes (Fig.5). Flow rate fluctuation frequency is close to frequency of rotor (and disk) rotation and is in the range 55 - 80Hz.

As to rear cavity, a pressure distribution pattern there appeared not as explicit as it was in the fore cavity and vortex structures are less pronounced. If in the rear cavity steam is pumped in from main flow path, then an inverse process is performed in the fore cavity, i.e. very often periodic blow-out to main flow path has been occurred. In that case, flow in the fore cavity had hub-to-tip direction and inverse motion was in the rear cavity where steam suction from turbine flow path was observed.

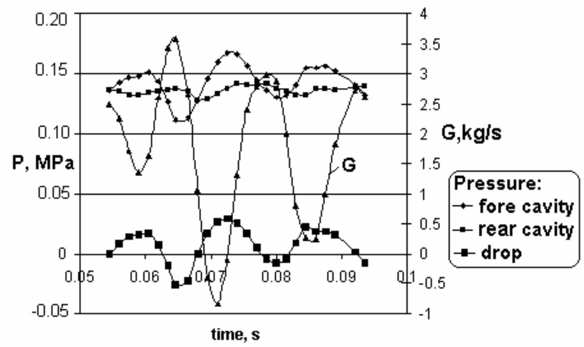

Figure 5. Static pressure difference in fore and rear cavities along surface of annulus enveloping balance holes. G - Steam flow rate through balance holes.

Character of the flow through balance hole is very different from that can be obtained with 1D model. Flow in the balance hole often runs against pressure gradient, i.e. there are two flows running simultaneously -- following gradient and against it, Fig. 5.

At the point of minimum pressure drop averaged along annulus surface, zone of pressure maximum is placed right over the hole and steam is directed to the hole (Fig.6). In the case of maximal pressure difference (Fig.7), a vortex zone of reduced pressure is located over the hole that causes even inverse rear-to-fore cavity steam movement.

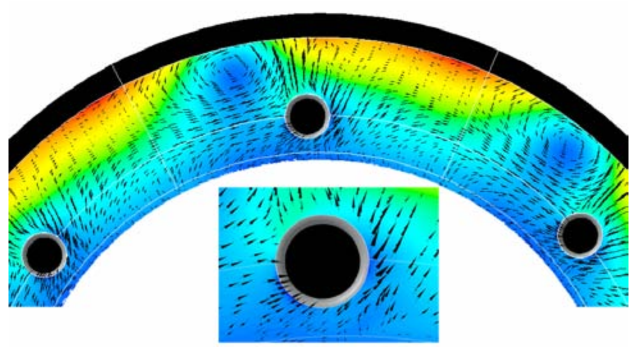

Figure 6. Flow near fore surface of the disk at maximal pressure drop and respective minimum of through holes flow rate

Reported analysis is valid for selected operation points. Computations carried out under other operational conditions showed that transient nature of that phenomenon becomes apparent in different ways. For example, as the overall pressure decreases (that, in fact, means reduced steam density) the fluctuation amplitude also decreases.

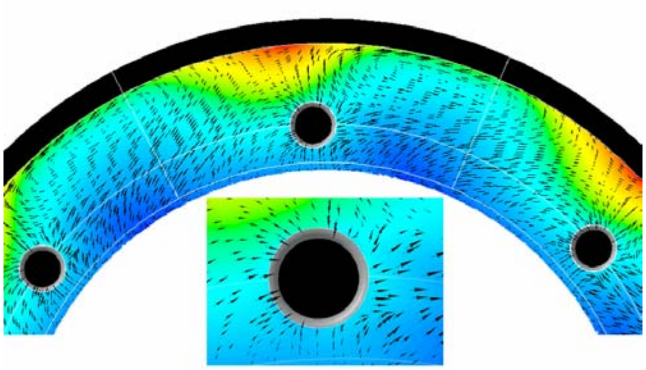

Figure 7. Flow near fore surface of disk at minimal pressure drop and respective maximum of through holes flow rate

Flow in the balance hole can be characterized as extremely random (Fig.8). Steam moves to the hole (Fig.8, flow 1) and gets twisted under impact of centrifugal forces. As result, a powerful vortex structure is generated inside the hole that overlaps most of effective cross-section. It prevents steam proceeding from one cavity to another. Then, steam flows out to the rear cavity in the direction opposite to disk rotation (flow 3). A reverse steam flow from rear to fore cavity also exists. Thus, two opposite flows run in opposite directions through the balance hole at the same time.

2D MODEL

3D simulation demonstrated quite complex periodic flow nature in two adjacent near disk cavities channeled with balance holes. Meanwhile, existing methods for flow rate prediction rely on some integral flow characteristics that in fact are quite sensitive to the method of averaging. Pressure in the disk cavity can vary even at the same radius around the seal circumference, as well as with distance along width of a seal between disk and diaphragm. Especially, it refers to the fore cavity that features transient periodic flow nature.

In spite of substantially 3D character of flow there is a desire to treat it as 2D axis-symmetric one, in order to evaluate possibilities of more simple and rapid solutions. For this purpose each cavity was studied separately (Fig.1, green zone) and mass exchange through balance holes from fore cavity to rear one was modeled by a steam flow through equivalent virtual annular channel. In fact, case when the steam that leaves the fore cavity through the annulus boundary comes to rear cavity through similar boundary (Fig.4) was considered. The size of boundary varied to obtain more accurate coincidence with 3D computations.

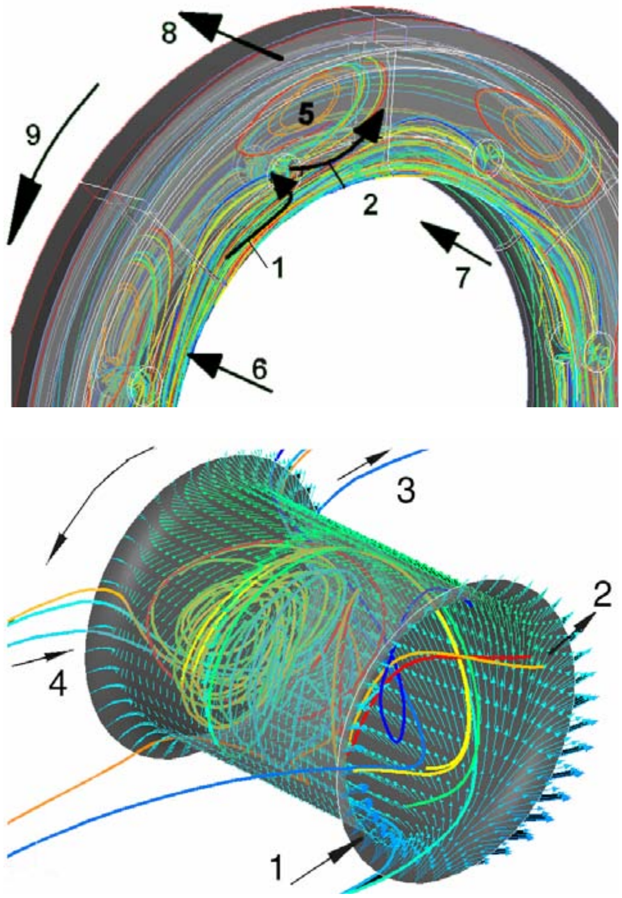

Figure 8. Steam flow in fore cavity (above) and inside balance hole (below)

Numerous axi-symmetric computations showed that flow obtained via 2D and 3D solutions are quite different at high pressures, i.e. when flow is substantially transient. Various boundary sizes were considered. For example, annulus channel width was assumed to be equal to the balance hole's diameter. In another approach it was computed from assumption that total cross section of balance holes is equal to cross section of annulus channel. In all cases, in order to preserve the same through-holes flow rate that was computed in 3D formulation, a greater pressure drop was needed in 2D modeling.

Thus, in 3D case there is better pressure equalization between two cavities than in 2D model at the account of steam migration through balance hole.

Correlations of pressure variation in 2D formulation with 3D instantaneous pressure values along selected radii are shown in Fig. 9 and 10. As diagrams indicate, when pressure is low, both types of simulation provide with almost similar pressure values, excluding radii that cross the balance hole axis. In high pressure case, an essential difference of the curves is obvious and this is an evidence of inadequacy of 2D computations at these conditions.

Comparison of 2D and 3D simulations

Type of simulation	Relative pressure at balance hole entry (kPa)	Relative pressure at balance hole exit (kPa)
3D – pressure is averaged in time and along area of annulus that envelopes balance holes	109.2	103.2
2D – width of annulus equals balance hole diameter	192.9	139.7
2D – annulus area equals to total area of the balance holes	248.5	117.9

1D MODEL

Flow through balance hole

Engineering approach to mass flow rate prediction through balance holes is based on equations

$$G = \varphi A \sqrt{2\rho(p_1 - p_2)}\,, \qquad (1)$$

$$\Delta p = p_2 - p_1 = \zeta \frac{G^2}{2\rho A^2}\,. \qquad (2)$$

where φ - mass flow rate coefficient, $A = z\frac{\pi d^2}{4}$ - balance hole area, d – hole diameter, z – holes number, ρ - fluid density at hole inlet, p_1, p_2 – pressure at the radius of holes arrangement in fore and rear cavities, ζ - resistance coefficient related to flow rate coefficient as

$$\zeta = \frac{1}{\varphi^2}\,. \qquad (3)$$

In order to find solution of (1) and (2) one needs to know values of φ or ζ, and either value of pressure drop at the balance holes radius, or mass flow rate.

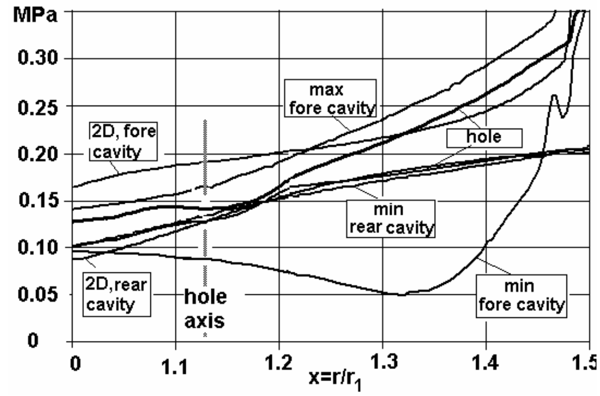

Figure 9. Distribution of relative static pressure along radii crossing balance hole axis (hole), vortex center (min) and zone of maximal pressure (max) in 3D model and along radius in 2D model for fore and rear cavities at reference pressure 16.3 MPa

Mass flow rate or resistance coefficient is determined usually by the experimental correlations that incorporate the following parameters:

- Disk/stator seal width;
- Relative pitch of holes arrangement along disk;
- Ratio of hole's leading edge radius to its diameter;
- Ratio of disk rotation velocity at radius of the holes location to velocity axial component in the hole.

All following analysis has been carried out as comparison with 3D solution.

Numerous axi-symmetric computations showed that flow obtained via 2D and 3D solutions are quite different at high pressures, i.e. when flow is substantially transient. Various boundary sizes were considered. For example, annulus channel width was assumed to be equal to the balance hole's diameter. In another approach it was computed from assumption that total cross section of balance holes is equal to cross section of annulus channel. In all cases, in order to preserve the same through-holes flow rate that was computed in 3D formulation, a greater pressure drop was needed in 2D modeling.

Thus, in 3D case there is better pressure equalization between two cavities than in 2D model at the account of steam migration through balance hole.

Correlations of pressure variation in 2D formulation with 3D instantaneous pressure values along selected radii are shown in Fig. 9 and 10. As diagrams indicate, when pressure is low, both types of simulation provide with almost similar pressure values, excluding radii that cross the balance hole axis. In high pressure case, an essential difference of the curves is obvious and this is an evidence of inadequacy of 2D computations at these conditions.

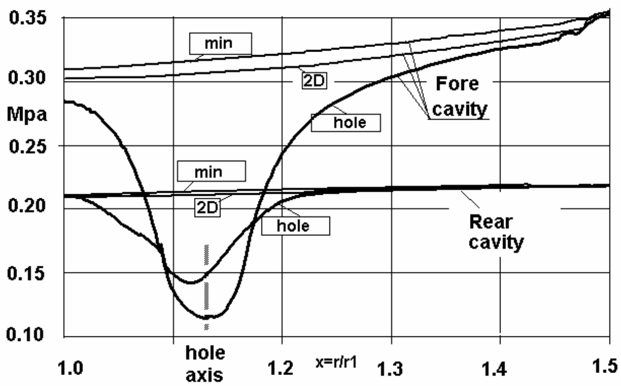

Figure 10. Distribution of relative static pressure along radii crossing balance hole axis (hole), vortex center (min) in 3D model and along radius in 2D model for fore and rear cavities at reference pressure 1.0 MPa

3D analysis reported 1.74 kg/s mass flow rate through balance holes that took place under small pressure drop equal to 0.007MPa. Using (1), (2) we arrive on the following values for the coefficients: $\varphi = 0.32$, $\zeta = 9.8$.

An attempt to estimate mass flow rate coefficient was undertaken using one of standard methods [8]. However, it precise value couldn't be found because a set of the complexes composed from above mentioned parameters dropped out of the confidence interval, although in the considered example near disk cavities geometrical characteristics and operational conditions were assigned with respect to real design of high pressure cylinder and parameters of operation. Nevertheless, obtained data for flow rate coefficient allows to estimate its value as less than 0.08, i.e. its computed value appeared 4 times smaller than actual one.

Shvetz, and Duban [5] proposed to define ζ for balance holes by taking into account rotation influence by the following equation (Eq. 4):

$$\zeta_{rotation} = \zeta_{stationary}\left(1 + 0.6K + 0.082K^2 - 0.024K^3\right)$$

where $\zeta_{rotation}$, $\zeta_{stationary}$ - resistance coefficient of rotating and stationery disk; $K = v/u$ - parameter of rotation, v – tangential velocity of disk rotation at the hole's axis radius; u – fluid average velocity.

Using Eq. (2 - 4) it has been found that $\zeta_{rotation}$ =26 that exceeded the actual value by 2.6 times.

Comparison of both methods showed their drawbacks for reliable computations of secondary flow path under high pressure conditions. However, these conditions are inherent to operation of steam turbine high pressure cylinders.

Pressure variation along radius in the cavities

As a rule, 1D computations of disk-stator system assume usage of equations based on the law of angular momentum conservation. This approach was first offered in [9] and then developed in [5, 10] in a form shown below:

$$\frac{d}{dx}\left(\beta \cdot x^2\right) = Ax^{3.6}\left[(1-\beta)^{1.2} - 1.715\beta^{1.8}\right], \quad (5)$$

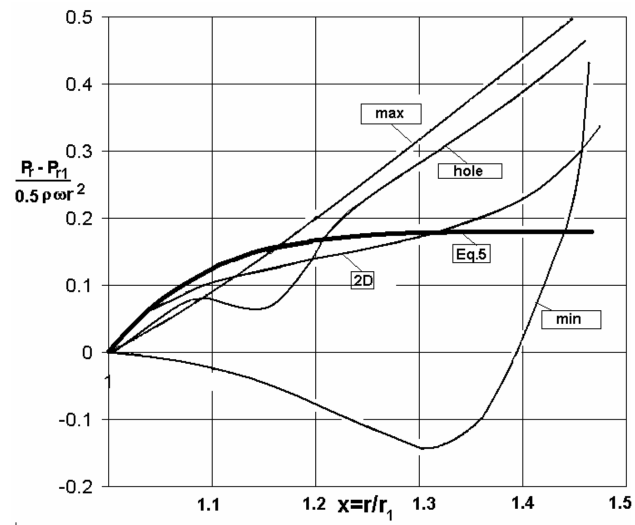

Figure 11. Pressure factor distribution in the fore cavity at reference pressure 16.3 MPa.
3D– instantaneous values along radiuses crossing balance hole axis (hole), vortex center (min), and maximal pressure zone (max); 2D – along radius;
Eq.5 - 1D analysis based on equations (5-8).

$$A = 0.0274K_v\left(\frac{r_1}{r_2}\right)^{2.6}\left(\frac{r_2}{s}\right)\mathrm{Re}_{\varpi}^{-0.2}, \quad (6)$$

$$K_v = \frac{2\pi r_2^2 s\rho\varpi}{G_r}, \quad (7)$$

$$\mathrm{Re}_{\varpi} = \frac{\omega \cdot r_2^2}{\nu}. \quad (8)$$

Here s – disk-diaphragm gap; r_1, r_2 – radiuses of diaphragm seal and axial seal at disk rim (peripheral radius); $x=r/r_1$ – dimensionless current radius.

Fig.11 presents a comparison of Eq. (5 - 8) solution with pressure factor distribution in the fore cavity along selected radiuses for one of moments in 3D (transient solution) and 2D (steady state solution) formulations. The diagram shows that 1D model is close enough to 2D case and dramatically differs from instantaneous distributions in 3D model. The latter evidently demonstrates that 1D and 2D models can not describe correctly the intricate nature of periodic flow in the fore cavity.

CONCLUSIONS

- Flow pattern in disk-rotor cavities is highly depended upon operational pressure, and it becomes transient at typical for HPC pressures.
- Amplitude of static pressure variation at the disk fore side at the radius defining balance hole location is commensurable with pressure drop in the balance holes.
- Frequency of characteristic flow variation is close to disk rotation frequency.
- Through-holes steam flow rate and pressure difference vary in time in a counter-phase manner.
- Amplitude of fluctuations is decreased with steam pressure decreasing, i.e. with density reduction.
- 2D axi-symmetric analysis doesn't allow flow modeling in near disk cavities with reasonable accuracy.
- 1D methods of computations are not as correct as it is required for high pressure flow modeling and need further improvement.

REFERENCES

1. Moroz, L, Tarasov, A, 2003, "Coupled CFD and thermal steady state analysis of steam turbine secondary flow path", International Joint Power Generation Conference, June 16-19, 2003, Atlanta, Georgia, USA, IJPGC2003-40058

2. Wilson, M., Pilbrow, R., Owen, J.M., 1997, "Flow and heat transfer in a preswirl rotor-stator system," ASME Journal of Turbomachinery, Vol. 119, pp.364-373

3. Cao, C, Chew, J.W., Millington, P.R., Hogg, S.I. "Interaction of rim seal and annulus flows in an axial flow turbine", Proceedings of ASME Turbo Expo 2003, Power for Land, Sea, and Air, June 16–19, 2003, Atlanta, Georgia, USA, GT2003-38368

4. Pilbrow, R., Karabay, H., Wilson, M., Owen, J.M., 1999, "Heat transfer in a "cover-plate" preswirl rotating-disk system," ASME Journal of Turbomachinery, Vol. 121, pp.249-256

5. Shvetz, I.T., Dyban, E.P., 1974, "Air cooling of gas turbine parts", Kiev, 488 p. (in Russian)

6. Matveev, Yu. Ya., Pustovalov, V.N., 1982, "Laminar viscous flow calculation between rotated disks", Fluid and Gas Mechanics, Proceeding of the Academic Science of USSR, 1, pp. 76-81 (in Russian)

7. Kapinos, V. M., 1966, "Convective heat transfer of turbulent flow between rotated disks in closed cavity", Proceeding of Higher School "Aviation technique", 1, pp. 132-129 (in Russian)

8. Kapinos, V. M., Matveev, Yu. Ya., Pustovalov, V.N., 1983, "Natural convection of unventilated cavities of steam turbines rotors", Moscow, Heat-and-Power Engineering , 8, pp. 36-39 (in Russian)

9. Lomakin, A.A., "Axial pressure in the centrifugal pumps with take into account a clearance", 1940, Moscow, "Soviet boiler fabrication", 12, pp.431-437 (in Russian)

10. RTM 108.020.113-77 "Fixed steam and gas turbines", 1979, Leningrad, CKTI, 103 p. (in Russian)

