EXPERIMENTAL AND THEORETICAL INVESTIGATIONS
OF HEAT TRANSFER IN CLOSED
GAS-FILLED ROTATING ANNULI II

D. Bohn, R. Emunds, V. Gorzelitz, and U. Krüger
Institute of Steam and Gas Turbines
Technical University of Aachen
Aachen, Germany

ABSTRACT

Increasing the thermal efficiency by higher turbine inlet temperatures is one of the most important aims in the area of gas turbine development. Because of the high temperatures not only the turbine vanes and blades have to be cooled, but also the knowledge of the mechanically and thermally stressed parts in the hottest zones of the rotor are of great interest. The prediction of the temperature distribution in a gas turbine rotor containing closed, gas-filled cavities, for example in between two discs, has to account for the heat transfer conditions encountered in these cavities. In an entirely closed annulus forced convection is not present, but a strong natural convection flow exists, induced by a non uniform density distribution in the centrifugal force field.

In /3/ experimental and numerical investigations on rotating cavities with pure centripetal heat flux had been carried out. The present paper deals with investigations on a pure axially directed heat flux. An experimental set-up was designed to realize a wide range of Ra-numbers ($2 \times 10^8 < Ra < 5 \times 10^{10}$) usually encountered in cavities of gas turbine rotors.

Parallel to the experiments numerical calculations have been conducted. The numerical results are compared with the experimental data. The numerical scheme is also used to account for the influence of Re-numbers on heat transfer without changing the Ra-number. This influence could not be pointed out by experiments, because a variation of the Re-Ra characteristic of the employed annuli was not possible.

It was found that the numerical and experimental data are in quite good agreement, with exception of high Ra-numbers, where the numerical scheme predicts higher heat transfer than the experiments show. One reason may be that in the experiments the inner and outer cylindrical walls were not really adiabatic, an assumption used in the numerical procedure. Moreover the assumption of a 2-D flow pattern may become invalid for high Ra-numbers. The influence of 3-D effects was studied with the 3-D-version of the numerical code.

In opposite to the radial directed heat transfer it was found that the Nu-number is much smaller and depends strongly on the Re-number whereas the radial heat transfer is only weakly influenced by the Re-number.

NOMENCLATURE

- $a$ = thermal diffusivity
- $b$ = distance between lateral side walls
- $c_p$ = specific heat at constant pressure
- $H$ = distance between outer and inner cylindrical wall
- $L$ = distance between hot and cold wall
- $p$ = pressure
- $q$ = heat flux from the hot to the cold wall
- $q_{\lambda}$ = heat transfer by conduction alone
- $r$ = radius
- $R$ = gas constant
- $T$ = temperature
- $\Delta T$ = temperature difference between hot and cold wall
- $(x, r, \varphi)$ = axial, radial, circumferential coordinate
- $(u, v, w)$ = relative velocity components in $(x, r, \varphi)$ - direction
- $\mu$ = dynamic viscosity
- $\rho$ = density
- $\omega$ = angular velocity of the cavity
- $\lambda$ = thermal conductivity
- $\alpha$ = section angle

Presented at the International Gas Turbine and Aeroengine Congress and Exposition
This paper has been accepted for publication in the Transactions of the ASME
Discussion of it will be accepted at ASME Headquarters until September 30, 1994
In the past many theoretical and experimental investigations have been carried out to study the heat transfer in rotating enclosures with a throughflow of cooling fluid, e.g. Ong et al. /12/, Farthing et al. /6/, Owen et al. /13/. For sealed cavities with a purely free convection flow the known theoretical and experimental investigations pertain mainly to constant temperature walls, and are limited to qualitative descriptions of the convective processes. These investigations differ with respect to the direction of the heat flux in the cavity (Fig. 2).

**Subscripts:**
- c = cold
- h = hot
- i = inner
- o = outer
- m = arithmetical mean
- max = maximum
- min = minimum
- SB = solid body
- w = water
- 0 = reference
- red = reduced

\[
Gr = \frac{r_m \cdot \omega^2 \cdot \Delta T \cdot L^3 \cdot \rho^2}{T_m \cdot \mu^2}
\]
Grashof number

\[
Pr = \frac{\mu \cdot c_p}{\lambda} = \frac{\mu}{\rho \cdot a}
\]
Prandtl number

\[
Ra = Gr \cdot Pr
\]
Rayleigh number

\[
Re = \frac{\rho \cdot \omega \cdot r_m \cdot L}{\mu}
\]
Reynolds number

\[
Nu = \frac{q}{\dot{q}_h}
\]
Nusselt number

\[
Ec = \frac{(\omega \cdot r_m)^2}{2 \cdot c_p \cdot \Delta T}
\]
Eckert number

**INTRODUCTION**

The development of gas turbines towards higher gas temperatures at the turbine inlet with a simultaneous increase of the compressor pressure ratio is a continuing trend to increase their thermal efficiency. In connection with this trend attention is being paid to the mechanically and thermally stressed parts of the gas turbine. To estimate these stresses a proper evaluation of temperature distributions in units and components operating in the hottest zones is required. In such a zone temperature nonuniformities may lead to considerable supplementary stresses, the permissible value of which is also determined by the temperature level.

At present only an approximate estimation of the temperature distribution in a gas turbine rotor containing gas-filled enclosures (Fig. 1) is possible. In those cavities a strong, free convective flow is induced. This convection is caused by the buoyancy force corresponding to centrifugal acceleration and temperature differences of the cavity walls. Such a flow increases the heat transfer throughout the cavities considerably.

Most of the investigations have been performed for an axially directed heat flux applied to a cylindrical rotating enclosure shown in figure 2(I). Kapinos et al. /9/ performed experimental investigations on heat transfer in an enclosure as described above. They pointed out the influence of Coriolis forces on the fluid motion and compared their experimental results with numerical investigations given by...
Harada and Ozaki /7/. Abell and Hudson /1/ conducted experiments on an oil-filled rotating cylinder. They deduced a correlation between the Nusselt number and the temperature difference between the hot and cold wall of the cylindrical cavity and the rotational Reynolds number. Chew /5/ also did numerical investigations on heat transfer in these enclosures, producing computations consistent with the experimental results achieved by Abell and Hudson /1/.

Investigations on rotating annular cavities, as in figure 2(II), were conducted by only a few authors. Most of these studies were not even performed under conditions as encountered in turbomachinery. Müller and Burch /11/ obtained measurements of the transient natural convection in an axially heated rotating annular enclosure simulating geophysical conditions. Similar experimental studies were made by Hignett, Ibbetson and Killworth /8/.

Bohn, Dibelius, Deuker and Emunds /2/ made a first numerical study on the flow and the heat transfer in rotating cavities with a pure axial heat flux. It was pointed out that the employed numerical scheme seems to be well suited for the prediction of the physics occurring in such cavities.

Considering the heat transfer in a cavity as shown in figure 2(III), Lin and Preckshot /10/ calculated the temperature, velocity and streamline distribution. Zysina-Molozhen and Salov /15/ analyzed experimentally the influence of rotational speed and various thermal boundary conditions on heat transfer in a rotating annular enclosure.

The experimental investigations of heat transfer in a sealed rotating cavity with axially directed heat flux (case II in Fig. 2) were performed on one fixed geometric configuration. The dimensions of this enclosure are given in Fig. 3.

EXPERIMENTS

Apparatus

The experimental investigations of heat transfer in a sealed rotating cavity with axially directed heat flux were carried out in a previous paper by Bohn, Deuker, Emunds and Gorzelitz /3/. There, the possibility of numerical prediction on the flow and the heat transfer was shown by comparison of the numerical results with the experimental data of Bohn and Gorzelitz /4/. It was found too, that the flow inside the cavity might be unstable.

However, there is still a lack of knowledge on rotating sealed cavities, bounded by an outer and an inner cylindrical wall, operating under conditions valid for gas turbines.

At the Institute of Steam and Gas Turbines at the Technical University Aachen experimental and theoretical investigations have been carried out studying the influence of heat flux direction and geometry on the convective heat transfer inside such enclosures. In this paper further experimental and numerical results of convective heat transfer in a rotating closed annulus with pure axial heat flux (Fig. 2(II)) are presented for conditions very close to turbomachinery operation.

Analyzing the basic conservation equations of mass, momentum and energy (see below) it can be demonstrated that

\[ \text{Nu} = f(\text{Ra}, \text{Re}, \text{Pr}, \text{H/r}_m, \text{b/r}_m) \]  

The Nusselt number (\( \text{Nu} \)) is defined as the ratio of the heat flux throughout the cavity to that flux which would occur in solid-body rotation without any motion relative to a co-rotating frame of reference; Thus, \( \text{Nu} \) is equal to unity for no convection and is greater than unity when convection takes place. The rotational Reynolds number (\( \text{Re} \)) has its origin in the Coriolis force terms in the momentum equations. The rotational Rayleigh number (\( \text{Ra} \)) is the product of the Grashof number and the Prandtl number and is related to the buoyancy term in the radial momentum equation. The Prandtl number (\( \text{Pr} \)) is a combination of fluid properties and does not change significantly due to temperature variations.

To get well defined conditions the side walls have to be isothermal. Therefore the electric resistance wire of the heater disc and the cooling channels of the cooling disc were designed ring-shaped (Fig. 6). Additionally the heater disc is insulated against the rotor disc to minimize the influence of the rotor temperature distribution.
Figure 4: Cross section of the experimental apparatus

Figure 5: Heating and cooling of the cavity

Another important condition is that the cylindrical walls should be adiabatic. So the radial cylindrical walls had also been heat insulated. The cavity can be pressurized while running the rotor by using a labyrinth housing. The rotor shaft is driven by a DC motor using a belt drive. Measurements of the rotational speed are accomplished by mounting a perforated disc on the shaft and using a coil to produce a voltage spike when one of the perforations passes it. Double bearings are installed at the ends of the rotor shaft enabling steady rotation.

The determination of the heat flux from the hot wall to the working fluid and from the fluid to the cold wall is realized by measuring the temperature differences across a thermal resistance (Fig. 5) and using the law of thermal conduction:

\[ \dot{q} = -\lambda \frac{dT}{dx} \bigg|_{x=b} \]  

This thermal resistance is constructed as a disc which is made of Polymethacrylimid (PMI). Its conductivity and thickness are chosen...
in such a way that a sufficient temperature drop can be obtained due to the expected heat fluxes. Temperatures are measured using thin-film resistance thermometers of platinum. Seven thermoelements were radially distributed on each side of the thermal resistance, seven thermometers were inside the heater disc and three on each side of the cylindrical walls. A telemetry unit, fixed on one end of the rotor shaft, registers the analogue signals of the resistance thermometers and converts them into digital. The digital data are transmitted out of the telemetry unit to an interface of a personal computer (RS 232 C) by a slip ring assembly.

**Procedure**

The procedure to get experimental results of the heat transfer in the closed rotating annulus with axial heat transfer is the same as described in the last paper /3/ for the heat flux directed radially. Only some technical data have been changed because of the constructional differences between the two test rigs. The maximum rotor speed was set up to 2500 rev/min. The maximum pressure was 3.5 bar and the temperature of the hot left side of the cavity was varied up to 62 °C. The minimum temperature of the cooled side wall was 8°C. The temperature of the inner side of the thermal resistance depends on the absolute heat flux.

During the first experiments it was detected that the temperature on the hot side wall was not isothermal in spite of the ring-shaped designed electrical heater. The difference between the local and the average temperature was so large that the measurements could not be evaluated in the right way. Especially the Nu-number, which is defined with the assumption of an isothermal wall, becomes wrong. The greatest difference occurs at the outer radius on the hot side wall (Fig. 7) because of heat losses to the rotor disc. To solve this problem additional air heaters were applied on the outer surface of the rotor disc carrying the electrical heater.

**Experimental Results**

As shown above the heat transfer can be expressed by the Nu-number, which is given as a function of Ra, Re, Pr, H/r_m and b/r_m. The geometry of the annulus was fixed - compare with figure 3 - and also the Pr-number was not varied in the present study. From eqn. 1 it can be obtained consequently that:

\[ Nu = f(Ra, Re) \]  \hfill (2)

There is only a great influence of the Re-number on the Nu-number if the temperature ratio \( AT/T_m \) varies over a large range. As the temperature ratio has been varied only over a small range during the tests, the Nu-number depends only on the Ra-number. The influence of the Re-number on the heat transfer, which is predicted by the numerical methods, is discussed later in this paper.

From the experiments a heat transfer law was evaluated:

\[ Nu = 0.346 \, Ra^{0.124} \]  \hfill (3)

The value of the exponent (0.124) shows that the Nu-number depends only weakly on the Ra-number.

Because of the very low convective heat flux - the Nu-number has only values between four and eight - the heat transfer through the cylindrical radial walls has to be taken into account. It takes values up to 30 % of the convective heat flux through the axial walls. The influence of the radiative heat transfer on the heat flux was calculated. For Nu-numbers greater than five this amounts less than 10% of the total heat flux.

In figure 9 the pure axial heat flux is compared to the radial one.
There are only those measurements plotted which have the same ratio of $\Delta T/T_m$ and the same Re-number. It is clearly shown that over the whole range of Ra-numbers the heat transport in the case of pure radial heat flux is much greater than the one with pure axial heat flux. This dominance becomes stronger with increasing values of the Ra-number. For Ra-numbers of about $2 \times 10^8$ the radial heat transfer is about a factor 2.5 larger than the axial heat transfer, for Ra=10$^{11}$ this factor increases to 6. This comparison shows that the radial heat transfer is the important one for cavities with a mixture of axial and radial temperature distributions as occurring in gas-turbine rotors.

Therefore, the heat loss over the radial walls have a great influence on the buoyancy driven flow in a closed gas-filled cavity.

As explained before only certain temperature ratios $\Delta T/T_m$ could be adjusted. The influence of the temperature ratio on the Ra-Re characteristic is shown in figure 10. The min./max. values of $\Delta T/T_m$ are typical boundary conditions for gas turbine rotors.

To study the influence of the Re-number on the heat transfer mechanism, the pure axial heat transfer and the radial heat transfer have been investigated with a 2-D and 3-D numerical code.

The results of these calculations are shown in figure 11, in which the Nu-number is plotted as a function of Ra for different Re-numbers.

**NUMERICAL INVESTIGATION**

Although the geometry of the cavities under consideration is quite simple, the flow is characterized by a complex interaction of convection, viscous forces, pressure forces, buoyancy effects and Coriolis forces. In /2/ flow structure and heat transfer were analyzed for a test rig with a pure co-axial heat flux situation, and special attention was paid to the buoyancy and Coriolis forces, which were found to be the most important terms to determine the heat transfer. In /3/ experimental and theoretical investigations were carried out for a pure centripetally directed heat flux situation for Ra-numbers usually encountered in cavities of gas-turbine rotors.

In the present case we started with experimental and theoretical examinations on the flow and the heat transfer for a pure co-axial heat flux situation for Ra-numbers $2 \times 10^8 < Ra < 5 \times 10^{10}$ usually encountered in cavities of gas-turbine rotors.
From our experiments, no information about the flow structure and only little information about the thermal conditions at the side walls can be obtained. Therefore we restrict our numerical analysis to the idealized case of isothermal side walls and adiabatic cylindrical walls. Although this is not a common investigation of the flow pattern inside the cavity, it may highlight some basic features of this type of flow and can be considered as a basic case that is independent of special thermal conditions at the side walls.

Basic Modelling Assumptions

The co-axial heat flux was modelled assuming that the temperatures at the side walls were different but uniformly distributed, while all other walls were assumed to be adiabatic.

The computer code solves the conservation equations for mass, momentum and energy. All computations were carried out for air, the density was calculated by the ideal gas law, and all other properties were treated as functions of temperature. Some common assumptions for natural convection flows are made: in the viscous terms of the momentum equations the compressibility is neglected because the velocities at this type of flow are very low. In the energy equation the influence of the dissipation and pressure changes are assumed to be negligible too, due to very small Ec-numbers (Ec < 0.1). The flow is assumed to be laminar in the range of Gr-numbers considered here. These assumptions for the axial heat flux mechanism in a rotating closed annulus with square cross section could be verified by theoretical studies presented in /2/.

The Governing Equations

The steady state governing equations are derived and declared comprehensively by D. Bohn et al. /2/. The dimensionless form of the equations reads:

\[
\frac{\partial \bar{p}}{\partial t} + \frac{1}{r} \left( \frac{\partial \bar{p} \bar{u}}{\partial x} + \frac{\partial \bar{p} \bar{v}}{\partial y} + \frac{\partial \bar{p} \bar{w}}{\partial z} \right) = 0
\]

\[
\frac{\partial \bar{p} \bar{u}}{\partial t} + \frac{1}{r} \left( \frac{\partial \bar{p} \bar{u} \bar{u}}{\partial x} + \frac{\partial \bar{p} \bar{v} \bar{u}}{\partial y} + \frac{\partial \bar{p} \bar{w} \bar{u}}{\partial z} \right) - \frac{\bar{p} \bar{u}}{\bar{r}} \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} \right) = -\frac{\partial \bar{p}_{\text{red}}}{\partial x}
\]

\[
\frac{\partial \bar{p} \bar{v}}{\partial t} + \frac{1}{r} \left( \frac{\partial \bar{p} \bar{u} \bar{v}}{\partial x} + \frac{\partial \bar{p} \bar{v} \bar{v}}{\partial y} + \frac{\partial \bar{p} \bar{w} \bar{v}}{\partial z} \right) - \frac{\bar{p} \bar{v}}{\bar{r}} \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} \right) = -\frac{\partial \bar{p}_{\text{red}}}{\partial y}
\]

\[
\frac{\partial \bar{p} \bar{w}}{\partial t} + \frac{1}{r} \left( \frac{\partial \bar{p} \bar{u} \bar{w}}{\partial x} + \frac{\partial \bar{p} \bar{v} \bar{w}}{\partial y} + \frac{\partial \bar{p} \bar{w} \bar{w}}{\partial z} \right) - \frac{\bar{p} \bar{w}}{\bar{r}} \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} \right) = -\frac{\partial \bar{p}_{\text{red}}}{\partial z}
\]

The system of coupled differential equations is solved numerically with a finite volume scheme. A non-uniform staggered grid is defined, with T and \( p \) being calculated at the main grid points and u,v,w being calculated at locations which are midway between the main grid points. In the employed "hybrid" approximation scheme upwind differencing is used for the convective terms when the cell-Peclet-number is greater than 2, otherwise central differencing is used for these terms /14/.
Theoretical Results For The Closed Rotating Cavity

In figure 12 the temperature distribution and the flow pattern of a 2-D calculation for the closed rotating annulus with a pure co-axial heat flux are shown. The computation was carried out for air at $T_m=317$ K, resulting in a Pr-number of 0.6925 with $Ra=2.261\cdot10^8$ and $Re=6.57\cdot10^4$. As described in /2/ for a rotating cavity with $r_m/h=4$, $Ra=6.6\cdot10^6$ and $Re=10^4$ an analogous flow situation occurs here. The fluid circulates around the walls in boundary layers, and nearly no motion occurs in the core region (see fig. 12b). Due to the buoyancy force the cold and heavy fluid at the cold side wall flows to the outer cylindrical wall. Passing this wall, the fluid is warmed up. When the fluid reaches the hot side wall a large temperature gradient in the corner appears. Opposite temperature gradients are localized in that corner where the inner cylindrical wall meets the cold side wall. The physical mechanism is discussed more in detail in /2/.

Due to the theoretical investigations on the heat transfer in closed rotating annuli with pure co-axial heat flux done in the past we proceeded with further computations using the 2-D version of the program code.

In figure 13 the comparison of the numerical results with experimental data is shown for a range of $Ra$-numbers $2\cdot10^8<Ra<10^{10}$. The result for the Nu-number taken from the measurement is plotted with the variation given by the experimental apparatus.

![Figure 12a: Temperature distribution of a 2-D calculation, $Re=6.57\cdot10^4$, $Ra=2.261\cdot10^8$](image)

![Figure 12b: Flow pattern of a 2-D calculation, $Re=6.57\cdot10^4$, $Ra=2.261\cdot10^8$](image)

![Figure 13: Comparison of the Nu-number obtained from experiment and 2-D calculation](image)

It can be taken from this figure, that in case of $Ra<2\cdot10^9$ the theoretical prediction of the Nu-number matches well the Nu-number taken from experiment. Increasing the $Ra$-number, the numerically predicted Nu-number differs more and more from the Nu-number determined on the basis of experimental data. The reason for this is on one hand the supposition of a 2-D flow in the cavity and on the other hand the assumption of ideal adiabatic cylindrical side walls, which could not be realized in the experiment. Due to this it is obvious that in the experiments a mixture of an axial and a radially directed heat transfer occurs. Therefore, it is evident that the calculated Nu-number - based on a pure co-axial heat flux direction - must be greater than the Nu-number taken from experiment containing the heat losses throughout the cylindrical walls. The influence of these non-adiabatic cylindrical walls is shown in figure 14, where the temperature distributions along these walls are plotted.
At the inner cylindrical wall, where the fluid flows from the hot to the cold side wall the numerically predicted temperatures are somewhat greater than the measured ones. At the outer cylindrical wall, where the fluid flows from the cold to the hot wall an analogous but opposite situation occurs. Due to the relative small heat transfer throughout the cavity, the assumption of ideal adiabatic cylindrical side walls becomes significant on the overall Nu-number. As well known for other bouyancy driven flows - see the Bénard-Problem - a critical Ra-number exists, at which the flow structure changes from a 2-D one to a 3-D one. To investigate such a complex flow phenomena we used the 3-D code for the calculations. The result obtained for the Nu-number, \( \text{Nu}_{3D}=5.475 \), calculated with \( \text{Ra}=2.261\times10^8 \) and \( \text{Re}=6.57\times10^4 \) differs strongly from the one taken from the 2-D calculation or experimental data respectively. In figure 15 results of the 3-D calculation for the temperature distribution is shown (for comparison see fig. 12a for the 2-D calculation). It can be seen that the temperature distributions of both cases are similar to each other, but in the region where the cold fluid reaches the hot wall (analogous to the region where the hot fluid reaches the cold wall) a significantly different temperature gradient occurs.

In figure 16 the temperature distribution is plotted in an \( r-\phi \) plane for a medium axial position.

In the regions near to the outer and inner cylindrical wall no circumferential temperature gradient exists. Between these regions an area

---

**Figure 14a:** Experimental and theoretical temperature distribution at the inner cylindrical wall

**Figure 14b:** Experimental and theoretical temperature distribution at the outer cylindrical wall

**Figure 15:** Temperature distribution of a 3-D calculation, \( \text{Re}=6.57\times10^4, \text{Ra}=2.261\times10^8 \)

**Figure 16:** Temperature Distribution of a 3-D calculation, \( \text{Re}=6.57\times10^4, \text{Ra}=2.261\times10^8 \)
The predicted Nu-number of the 2-D and 3-D calculations are close together for a constant Re-number of Re=1.457\times10^4. Due to the damping influence of the Re-number an enlarged flow circulation between the cold and the hot wall is induced by decreasing the Re-number more and more. At this situation the Nu-number is increased. Increasing the Re-number up to Re=6.57\times10^4 an increasing fluid motion in circumferential direction occurs, which can not be calculated by a 2-D algorithm.

### SUMMARY AND CONCLUSIONS

Investigations have been carried out on convective heat transfer in a closed annulus rotating around his horizontal axis. A pure axial heat flux throughout the cavity was applied by heating the axial side walls. All other walls of the annular cavities were thermally insulated.

Measurements have been performed varying the Ra-number in a range usually encountered in the gas-filled cavities of gas turbine rotors (2.10^8 < Ra < 5.10^{10}). It can be concluded, that in the case of an axially directed heat flux the heat transfer depends strongly on the Re-number. For the Re-Ra characteristics from the obtained experimental apparatus only a weak influence of the Ra-number on the heat transfer occurs. With radially directed heat flux throughout the cavity the heat transfer depends strongly on the Ra-number but only weakly on the Re-number.

### ACKNOWLEDGEMENT

The authors thank the Arbeitsgemeinschaft Hochtemperatur-Gasturbine (AG Turbo) for sponsoring the experimental research work. The AG Turbo is funded by German Gas Turbine Industry Companies and Ministry of Research and Technology (BMFT).

### REFERENCES


