

**Project Report**  
**2008 MVK160 *Heat and Mass Transport***  
**08 May 2008, Lund, Sweden**



**LUNDS UNIVERSITET**  
Lunds Tekniska Högskola

**Axial gas turbine blade cooling**  
**With**  
**Impingement/Film-Cooling**

**Adrian Dahlquist**  
**Dept. of Energy Sciences, Faculty of Engineering,**  
**Lund University, Box 118, 22100 Lund, Sweden**

## Contents:

<b>ABSTRACT:</b> .....	<b>2</b>
<b>NOMENCLATURE:</b> .....	<b>2</b>
<b>INTRODUCTION:</b> .....	<b>3</b>
<b>CFD MODELLING AND BOUNDARY CONDITIONS:</b> .....	<b>4</b>
<b>THE MODELING RESULT:</b> .....	<b>5</b>
The effect of blowing rate M .....	5
Consequence of pedestal number: .....	6
Number of impingement hole: .....	7
<b>CONCLUSIONS:</b> .....	<b>7</b>
<b>REFERENCES:</b> .....	<b>8</b>
<b>APPENDIX:</b> .....	<b>9</b>

## Abstract:

The aims with this rapport were to in a simple way describe a new and advanced cooling technology. The cooling technology described under the introduction below is a combination of traditional film-, impingement- and pin cooling.

The only parameter that has been considered is the film cooling efficiency  $\eta$ , and no concern to heat transfer between the wall and surface has been taken.

All studies have been carried out in the CFD program Fluent 6.1 by [1], and only the final results are present in this study.

## Nomenclature:

$P$	Pressure [bar]
$T$	Temp [K]
$\rho$	Density [kg/m <sup>3</sup> ]
$U$	Velocity [m/s]
$\dot{m}$	Mass flow [kg/s]
$M$	Blowing ration between two streams [-]. For example (i, $\infty$ )

$M_e$  Blowing ration at outlet of cavity, compare to free stream

$$[-] M_i = \frac{\rho_i \cdot U_i}{\rho_\infty \cdot U_\infty}$$

$D_h$   $D_h$  stands for the hydraulic diameter at the outlet from the cavity to the free stream [m]

$\frac{\partial}{\partial x_i} = 0$  Non gradient in the direction i

## Greek Symbols

$\gamma$  Ration of specific heat  $\frac{c_p}{c_v}$  [-]

$\eta$  Film cooling efficiency [-],

$$\eta = \frac{T_w - T_\infty}{T_c - T_\infty}$$

## Suffixes

0	Stagnation value
1,2,...	reference planes
$\infty$	far away, "free stream"
i	direction (x,y,z)

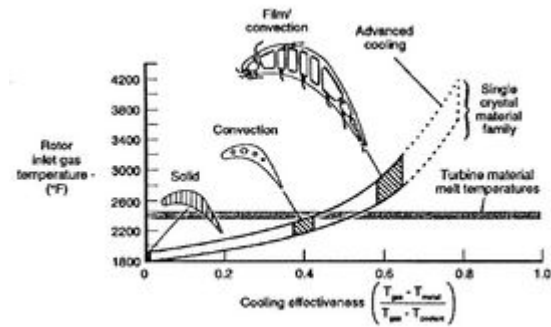
**Introduction:**

In the gas turbine hot air enters the nozzle at a high speed and temperature. The reason for the need of a high temperature is that it will increase the pressure

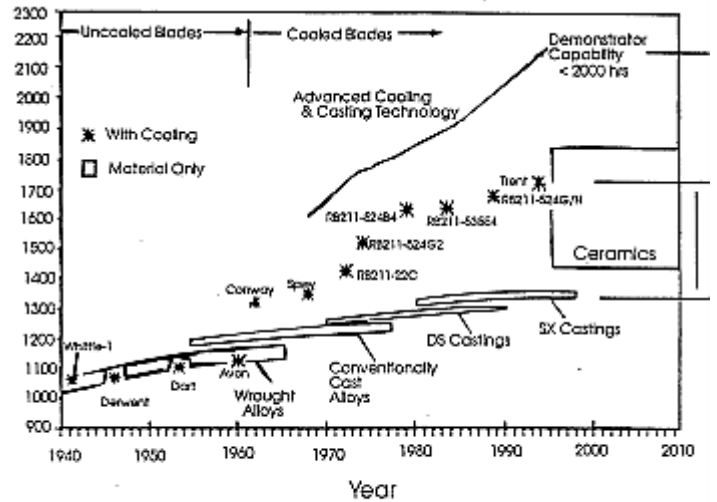
according to eqn 
$$\frac{P_{02}}{P_{01}} = \left( \frac{T_{02}}{T_{01}} \right)^{\left( \frac{\gamma}{\gamma-1} \right)}$$
 for an

adiabatic compression, and then also the opportunity to create a high specific work output, as the gas expands in the turbine. [2]

The thermal efficiency will as well increase with an increase of the inlet temperature to the rotor. This relationship between inlet temperature, specific power and efficiency, have lead to great engineering efforts in the attempt to push the temperature to higher and higher levels. To day the rotor inlet temperature "RIT", have reach a level far above the actually melting temperature of the turbine blade materials. As is shown in Figure 1a and 1b, the RIT can be about 1700K, as for the Trent airplane engine. For military aircraft the temperature can reach above 2000K when the need for long life and low specific fuel consumption aren't that important, therefore more coolant air can be bleed of from the compressor. The same figure also show that the melting temperature for the single crystal casting is just above 1300K, so the temperature difference is around 400K. [3],[4].



**Figure 1a:** Shows the RIT [K] that can be used together with different cooling techniques. Also the cooling efficiency and turbine blade melting temperature are shown. [5]



**Fig 1.1.1 - Advance of Materials & Cooling Technology**  
Source: Royal Aeronautical Society / Aerospace - 1994

**Figure 1b:** The RIT [K] vs. year. Also shown the improvements for the material melting temperature and cooling technology. [5]

To be able to decrease the temperature of the blade in an efficient way a combination of different cooling technology is employed, see below. The standard procedure are to bleed air from some stage of the compressor, bypass the combustion chamber, and inject it as cooling air in the turbine. But also steam and water are used as coolant medium in some special applications.

The cooling technology can be split into two groups, internal and external cooling. The first one includes the forced convection as the air flow in multiple passages inside the blade. Some different technologies are used here to increase the heat transfer from turbine blade to the coolant air.

- **Ribs:** The idea behind ribs is to trip the boundary layer directly after it have attached and in that way increase turbulence and then also increase heat transfer. The variation of the geometry for the ribs to achieve this is countless. They can also be in a V-format or combined in a matrix. So a lot of study is needed for achieve an optimum concerning maximum heat transfer for minimum pressure losses.
- **Pin-fin and dimples cooling:** Some other technologies are to place pins in the channel and at that way create a recirculation zone behind and a high heat transfer. The uses of dimples are to place holes in the wall and in that way create turbulence, and high heat transfer.

- Impingement cooling is when a concentrated airflow is impinging on the inner surface of the blade through some small hole.

External cooling is when the air after being used for internal cooling is ejected out through some holes or slots to the external surface to create a protecting surface “air film” acting as a barrier from the hot combustion gases.<sup>1</sup>

An interesting new technology that combines pin, impingement and film cooling in an attempt to further increasing the RIT, is what this rapport will be about.

In this text we will call it IFC, standing for “Impingement/Film Cooling”.

The idea behind IFC is that instead of as for normally film cooling, where the fluid is ejected directly out to the surface and free stream air  $U_\infty$ , instead letting it pass a sophisticated flow passage, see figure 2 and “figure 7 in appendix”. The flow through the passage can be split into a numbers of steps as below.

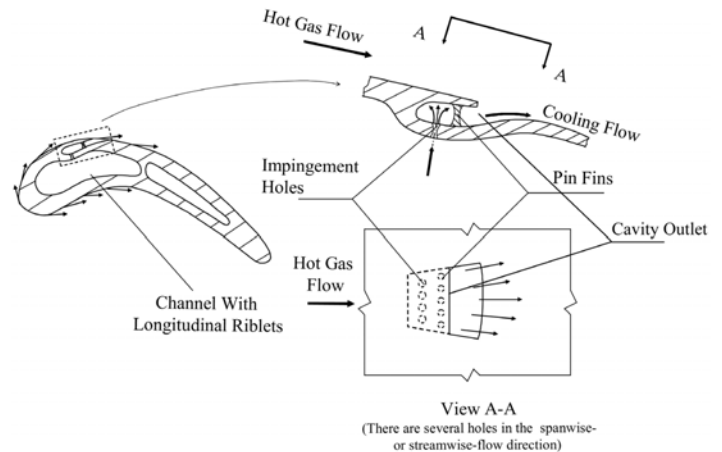
1. After that the air has been used for internal cooling it is impingement through a row of small holes into a cavity. This will give a high heat transfer from the inside surface of the outer wall, closest to the surrounding hot gas stream, to the cooling air stream inside the cavity. In that way it’s cooling a larger part of the upstream blade, compare to a normal film cooling.
2. The air is now driven by the pressure gradient out from the cavity in the direction towards the free stream. It passes a row of pins that will create a separation of the flow and a recirculation zone to increases the turbulence and heat transfer rate even more.
3. Finally the airflow is passage through a duct and out on the suction surface of the blade, to act as a protecting film barrier.

The efficiency is depending on the geometry, number of pins 1-3, the turbulent intensity “TI”, mass flow “ $\dot{m}$ ”, velocity “U” and injection angle to the free stream.

<sup>1</sup> For the hole introduction down to this point, references [2],[3],[4] has been used together to get a fundamental understanding of the problem, and will not be specified specific for every text line.

These parameters can be chosen accurate to optimize the efficiency of the cooling mechanism.

The effects for the cooling efficiency, downstream flow velocity and wall temperature when a number of parameter are altered will be studied in the following text. The studies were done by simulation in CFD by [1].



**Figure 2:** A schematically view of the IFC arrangement. From the side plane and also top view A-A. Here the impingement hole, pin-fins and filmcooling can be seen.

### CFD modelling and boundary conditions:<sup>2</sup>

The modelling was done in the CFD program Fluent 6.1. Below some of the most important assumption and geometry will be explained but for a more comprehensive description see [1]. Figure 2 and figure 7 in appendix shows the geometry. The simulations have been made for four different geometries, see table 1.

Name	Impingement holes	Pins-fins
NOV1-4p	3	4
NOV1-2p	3	2
NOV1-0p	3	0
NOV2-4p	1	4

**Table 1:** The four different geometries of the cavity. Viewing the differences in impingement holes and pin-fins

The size of the computational domain was  $21D_h$  upstream,  $32 D_h$  downstream and  $26D_h$  above the injection surface.

The free stream temperature and cooling air temperature were hold constant at 1300K respectively 750K.

The  $U_\infty$  were varied between 1-6 m/s, while the coolant inlet velocity were maintained at 3 m/s,

<sup>2</sup> The present results, figure and calculations from hair and down to the headline “Number of impingement hole” below is from Immarigeon A and Hassan I, ref [1].

corresponding to a variation of the blowing rate  $M_i$  between 0.87-5.22. At the boundary of the solution domain it's assumed that there will be no change anymore, so the outlet boundary condition is  $\frac{\partial}{\partial x_i} = 0$ , where  $x_i$  is the three dimension (x,y,z).

At the top surface a no slip condition were assumed and, and the surface downstream of the injection hole were assumed as adiabatic.

### The modeling result:

To be able to draw any conclusions from the experiments, efficiency  $\eta$  was used as a measurement. The definition is:

$$\eta = \frac{T_w - T_\infty}{T_c - T_\infty}$$

And  $T_w$  is the average temperature in z-axis close to the wall,  $y = 0^+$ ,  $T_\infty$  is the free stream (1300K),  $T_c$  is the cooling air (750K).

To have a reference to the new  $\eta$ , and also find a accurate turbulence model the CFD results for different turbulence models with a normal film cooling case was compared with measurements. The result is shown in figure 3.

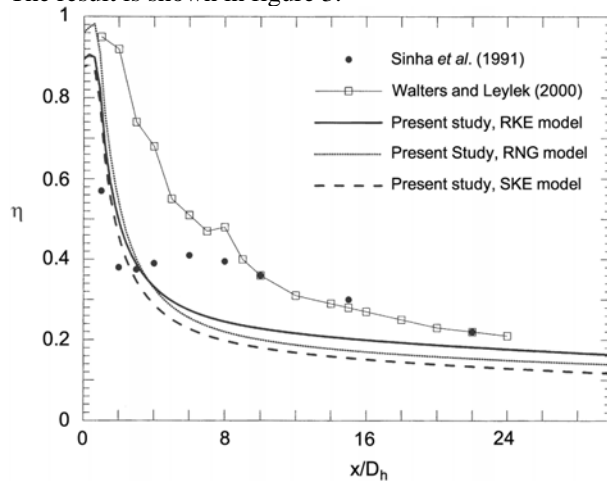


Figure 3: Compare CFD against real experimental measurements.

### The effect of blowing rate M

To study the effect of different blowing rates the efficiency had been plotted against a blowing rate variation.

Definition of blowing rate and the range:

$$M_i = \frac{\rho_i \cdot U_i}{\rho_\infty \cdot U_\infty}, \text{ cavity inlet, range (0.87-5.22)}$$

$$M_e = \frac{\rho_e \cdot U_e}{\rho_\infty \cdot U_\infty}, \text{ cavity outlet, range (0.44-1.77)}$$

A problem with film cooling is that when the cooling flow speed increases there is a great risk for liftoff for the cooling jet so the hot free air stream can breakthrough and directly heat the blade. A decrease in  $\eta$  would in that case be seen close to the jet inlet, before the jet might reattach and increase the  $\eta$  again. The reason for this liftoff is simply due to the high momentum in y-direction of the jet compare to the free stream. With the normal cylindrical injection hole this has been shown to normally occur around  $M = 0.5$ , (but will differ a bit for different shape and angle of the injection hole). But for the IFC, there were now sign for any liftoff even at  $M_i = 5.22 \Rightarrow M_e = 1.77$ , as can be seen in figure 4. The reason for this is probably that due to the shape and diffusion in the cavity the momentum is reduced.

Figure 4 also shows that  $\eta$  decreases at a slower rate with  $x/D_h$  as  $M_i$  increases.

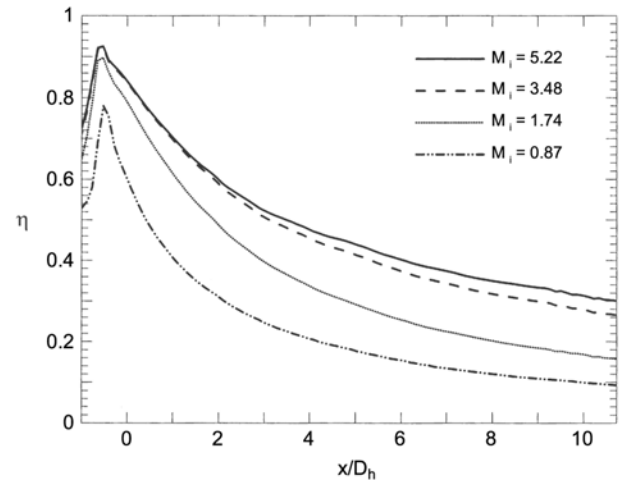


Figure 4: The film cooling efficiency at different M. There is no sign for any liftoff.

Figure 8 in appendix shows the distribution of the  $\eta$  in the z direction as a function of  $x/D_h$  and M.

The conclusions that can be drawn direct from this is that when M increases also  $\eta$  increases at all z

and  $x/D_h$ . And also that when  $x/D_h$  increases

$\eta$  at  $\left| \frac{z}{D_h} \right| > 1$  increases.

The increase at  $\left| \frac{z}{D_h} \right| > 1$  is due to the spreading

of the cooling jet in the z-direction. This in turn is due to the decrease in the jet momentum, when the energy is dissipated in the boundary layer. That's given a lower velocity and due to continuity eqn the area covered by cooling air in the z-direction is

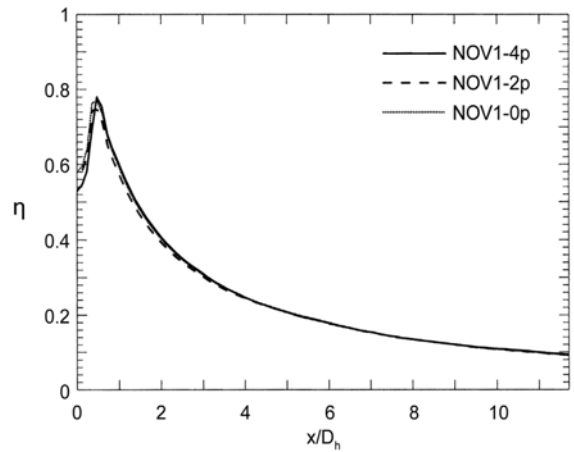
enlarged, therefore are also the cooling efficiency at bigger  $z$  improved.

**Consequence of pedestal number:**

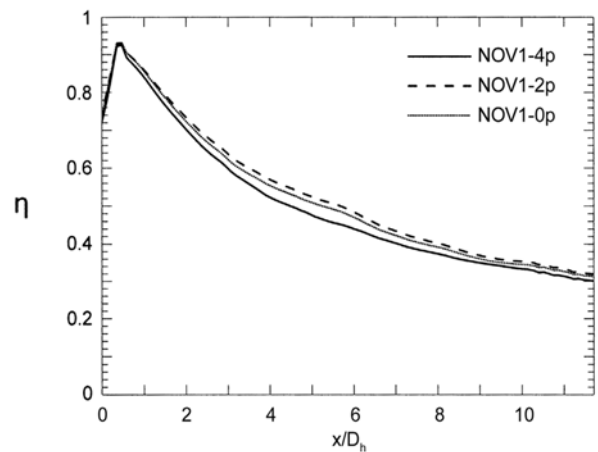
This section aim to investigate the difference of the outgoing flow, with different number of pedestals to disturb and increase the TI in the cooling flow outlet.

In figure 9 in appendix a picture of the normalized velocity at the cavity outlet shows that for example if there is only two pedestals in the center, the highest velocity is at the sides of the cavity. And if there aren't any pedestals at all the flow is more uniform.

When comparing the efficiency for the centerline, see figure 5, the configuration with two pedestals is slightly higher. That's primary due to the lower mixing as a result from a lower centerline velocity. Most turbulence and mixing in the  $z$ -direction occur for the four pedestals, as can be seen in figure 10b in appendix where the normalized velocity is shown. Figure 10a also illustrate that the penetration in the  $y$ -direction for the four pedestals are about 20% lower, compared to the other two. The consequence of this is a thinner cooling film in the center, and lower centerline cooling efficiency. But instead will the film cover a larger surface area downstream and increase the efficiency at greater  $z$ -direction as being shown in figure 5. The benefits and disadvantage for each case is close to cancel each other so that the mean cooling performance is basically equal for the three configurations.



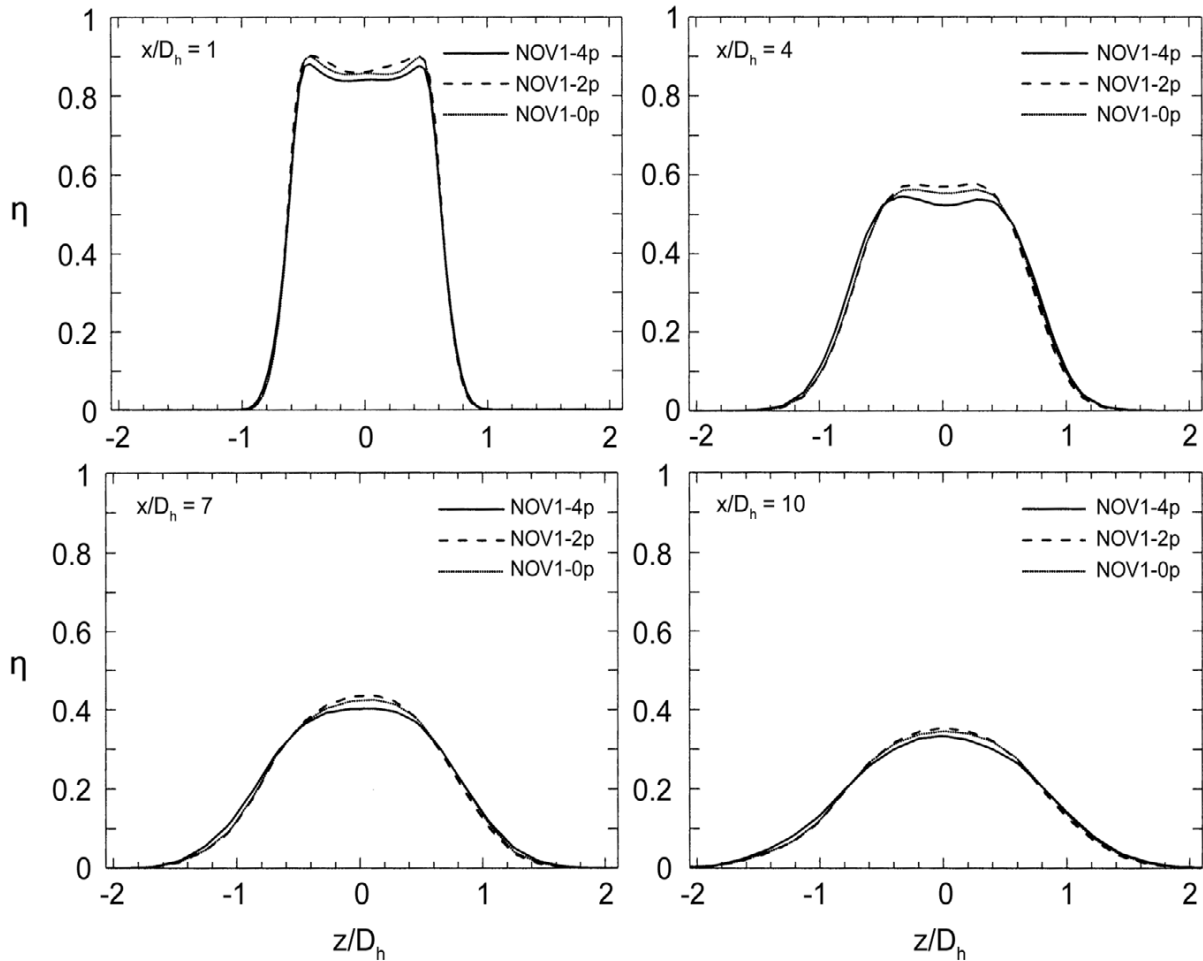
(a)  $M_i = 0.87$



(b)  $M_i = 5.22$

**Figure 5:** Centerline cooling efficiency.

At low flow rate the two pedestal configuration is slightly higher close to the injection and non significant difference further downstream. When the flow rate is increased the highest efficiency is for the two pedestals, and lowest for four pedestals.



**Figure 6:** Cooling efficiency in the “lateral” z-direction. There will be a slightly increase for NOV1-4P at large lateral position downstream.

**Number of impingement hole:**

The effects of on the down stream cooling efficiency were also studied for the NOV1-4P and NOV2-4P, (three and one impingement holes). But the effects for the inside blade surface cooling wasn’t studied.

The centerline cooling will not differ much between the two configurations, although there will be a slightly higher penetration in the vertical axis for one single impingement hole, NOV2-4P.

Meaning that for NOV1-4P the loss of laterally momentum will be less, and the film will stay closer to the surface as it enters the outside free stream,  $U_\infty$ .

One impingement hole will cause a greater loss of lateral momentum, as it expands in the z-plane to fill the whole cavity.

**Conclusions:**

An important trend was that the use of a cavity between the injection hole and free stream shown a great improvement to prevent liftoff. This means that when there is a capacity to use a higher mass flow of cooling air, it can be done to achieve a higher cooling efficiency, with the use of IFC.

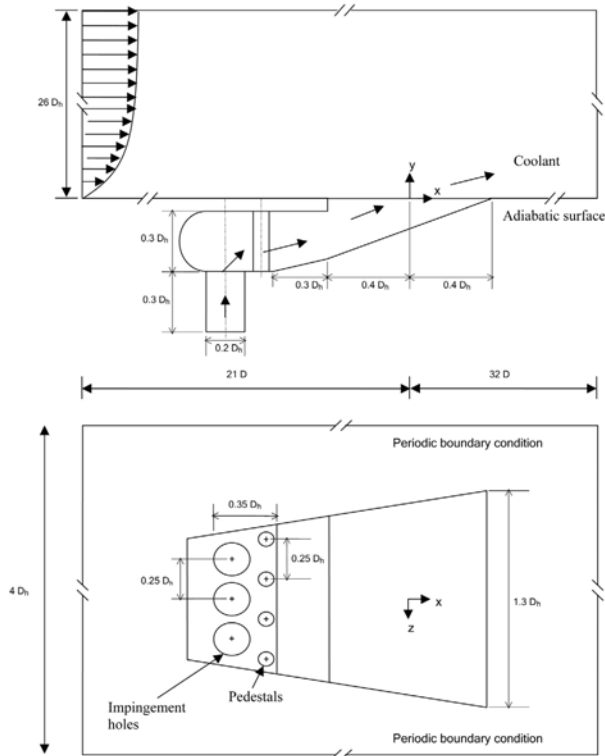
The effects of different numbers of pedestals were to effect the distribution of the cooling flow. For four pedestals the greatest lateral cooling were achieve, but with only two pedestals the highest centerline cooling efficiency were obtained, due to the lowest mixing with the mainstream.

Another very important question that hasn’t been considered by [1] is if it’s possible to mold a that highly advanced structure. Maybe it would be a better idea to skip the pin-fins, to get a simplified structure and more reliably turbine blade. Because the turbine blade is a highly stressed structure. And no internal stresses due to thermal gradient inside the blade under operation both at standard case and of design would be appreciated.

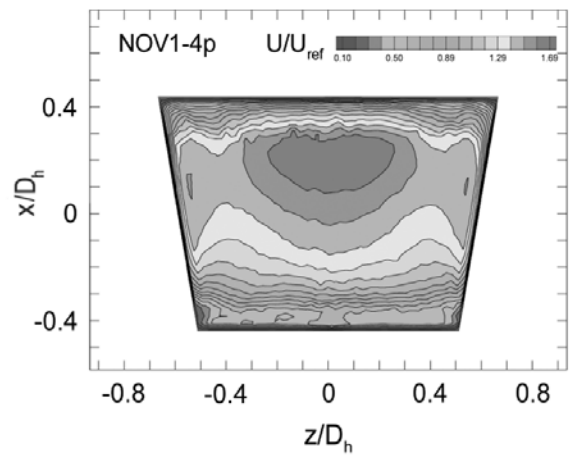
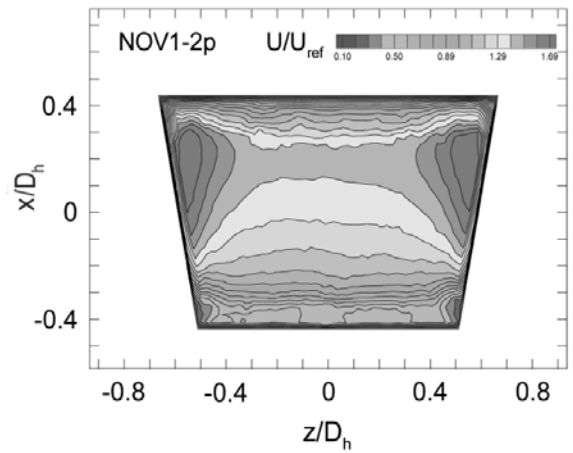
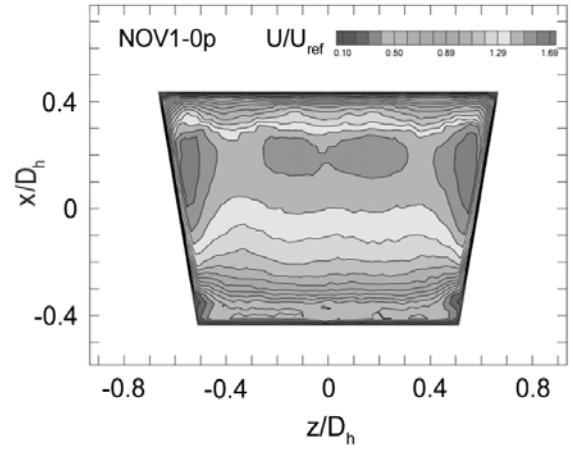
## References:

- [1] An advanced impingement/film cooling scheme for gas turbines – numerical study  
Department of Mechanical and Industrial Engineering, Concordia University, Montreal, Canada
- Authors:** Immarigeon A; Hassan I
- Journal:** International Journal of Numerical Methods for Heat & Fluid Flow  
**Year:** 2006 **Volume:** 16 **Issue:** 4  
**Pages:** 470-493 **Provider:** Proquest  
<http://www.emeraldinsight.com/Insight/viewContentItem.do;jsessionid=8399FE80ECC9FB867C07320D1C9A2986?contentType=Article&contentId=1556944>
- [2] Gas Turbine Theory  
HIH Saravanamuttoo GFC Rogers and H Cohen  
Fifth Edition
- [3] Lecture by Mats Annerfeldt at Siemens Industrial Turbomachinery AB
- [4] Gas Turbine Heat Transfer and Cooling Technology  
by Je-Chin Han  
ISBN: 156032841X
- [5] Figure 1 and 2 from MIT  
<http://web.mit.edu/16.unified/www/SPRING/propulsion/notes/node27.html>  
2008-04-28

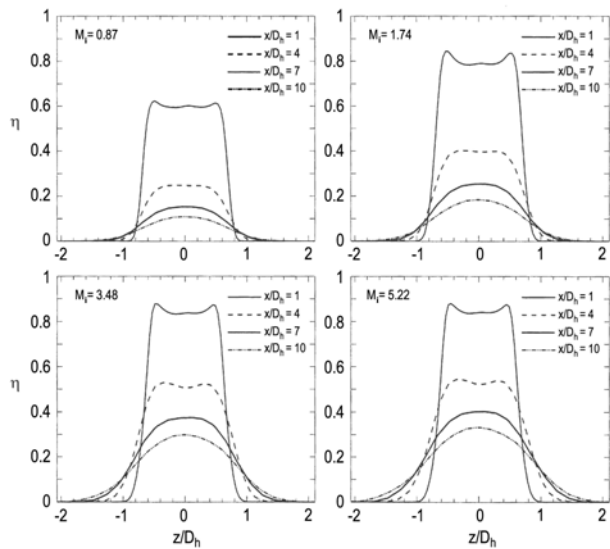
**Appendix:**



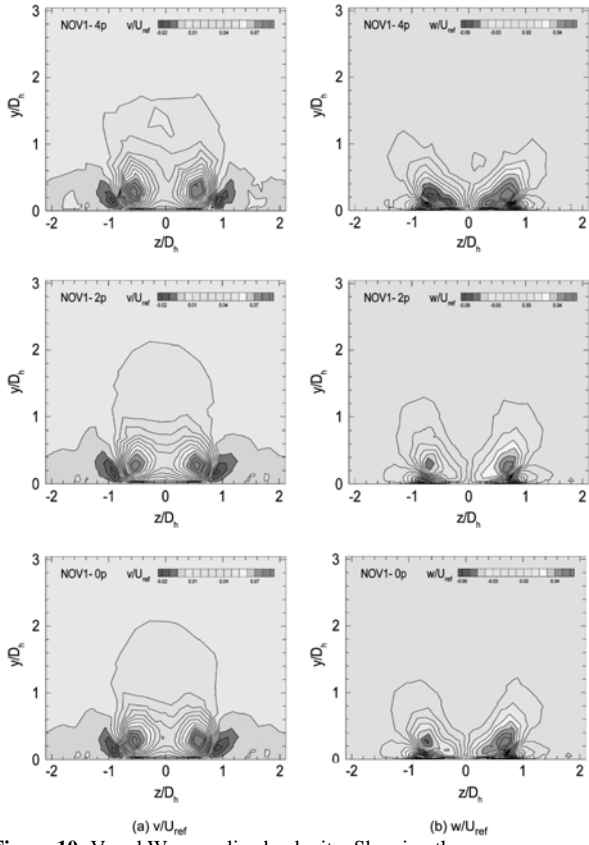
**Figure 7:** the modeling geometry.  $D_h$  stands for the hydraulic diameter of the inlet from the cavity to the free stream.



**Figure 9:** Shows contours of normalized velocity magnitude at the cavity outlet plane, for three different pedestal combinations.



**Figure 8:** Lateral cooling efficiency at different blowing rates,  $M$  and downstream position,  $x/D_h$ .



**Figure 10:** V and W normalized velocity. Showing the difference in penetration in the  $z$ - and  $y$ -direction for different pin-fins number.