Difference Between Ribbed or Dimpled Systems

Angel J. Ortiz Andujar
Mechanical Engineering
Bernardo Restrepo, Ph.D.
Mechanical Engineering Department
Polytechnic University of Puerto Rico

Abstract *— The purpose of this design project is to compare the heat transfer and pressure drop phenomena in different cases. The project was carried out with two different models, that is, a channel with ribs and another with dimples with numerical data at different Reynolds Numbers. The numerical study was carried out with the Flow simulation method. The result shows that for both channels with ribs and dimples there were no significant changes due to the geometry selected in this project. However, the friction factor is comparatively higher for both geometries with the lower Reynol numbers used.*

Key Terms *— Dimples, Gas turbine, Internal cooling, Ribs,.*

INTRODUCTION

A gas turbine is a combustion engine that produces a large amount of energy, depending on its size and weight. They are used for power generation, aviation, and ship propulsion, among others. The gas turbine is composed of the following parts: inlet, compressor, combustion chamber, turbine and exhaust as shown in Figure 1 [1].

The component called the turbine is placed after the combustion chamber and is subjected to high temperatures, resulting in higher cycle efficiencies. Over the years, turbine efficiency has increased, designs, airflow methods and materials have been modified. Different manufacturing companies fight to the end to find the ideal inlet temperature and efficiency of the turbines. Due to the increased power production and thermal efficiency requirements for gas turbine engines have required significant advances in turbine blade cooling technology. These advances allow the turbine blade to withstand higher temperatures for prolonged periods while reducing the amount of bleed air through the compressor, which is necessary to reduce the temperature of the blades [2].

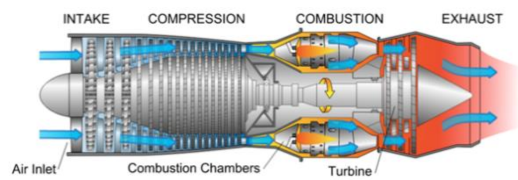

Figure 1
Gas Turbine Engine

In recent years, the cooling of gas turbine blades has been investigated, redesigned, and analyzed. There are many cooling techniques used in gas turbine blades, as seen in Figure 2, consisting of internal convection cooling, film cooling and external cooling. Project are continually working to improve cooling technology to increase gas turbine efficiency [3]. As one of the essential ways to ensure the safety and longevity of turbine blades, internal cooling has been extensively investigated in recent decades [4].

To improve the thermal efficiency of a gas turbine engine, the turbine inlet temperature is constantly increased until it produces high thermal loads on the turbine blades. Therefore, the new designs that are worked on must be able to improve the performance and protection of the gas turbine engine.

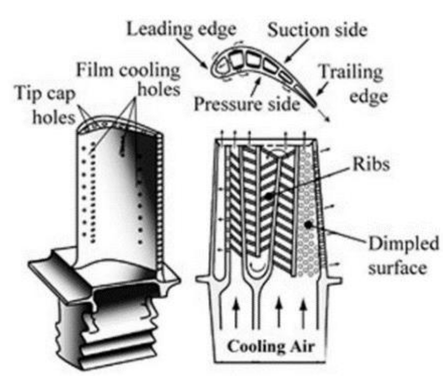

Figure 2
Gas Turbine Internal Cooling Passage

DESCRIPTION OF THE PROBLEM

The proposed project will focus on studying the comparison of effects on internal heat transfer under different conditions because a lot of research has been done to optimize the geometry in terms of maximizing the increase in heat transfer and maintaining the increase in factor friction as low as possible.

Therefore, this project will focus on comparing two models, one with ribbing and one with dimples. Specifically, turbine blades, the ribs are mainly used in the internal cooling channels in the middle of the component [5]. It mainly consists of using convective heat transfer in the internal channels, extracting heat from the metal wall of the blade to cool it, and this depends on the size of the ribs, distribution, flow, angle of attack and Reynolds number. The ribs are located on opposite walls, almost always towards the pressure side and the suction side, see Figure 3 [1]. There are cases where sometimes only one side has ribs, because the cooling h Property of Air coincides with the external load of the Blade [5]. The ribs cause the flow to separate at the top of the ribs and reconnect with the flow.

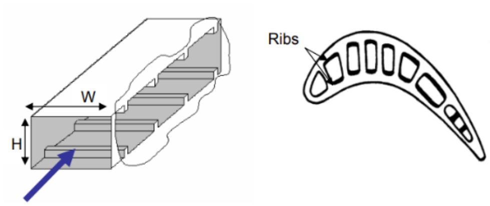

Figure 3
Ribbed Channel

The other part of the project will be carried out with linear dimples that produce exceptional heat, transfer performance and power management. In addition, it has a lower weight, penalty due to low pressure drop and simple manufacturing [6]. Figure 4 depicts the flow mechanism responsible for dimple-enhancing heat transfer. Therefore, the flow in the system separates at the dimple inlet and recirculation forms in the upstream half of the dimples, where heat transfer is very low. The flow rejoins a little before exiting the dimples. The air experiences flow circulation in its wide section [6]. For studies with dimple systems the flow mechanism is the same.

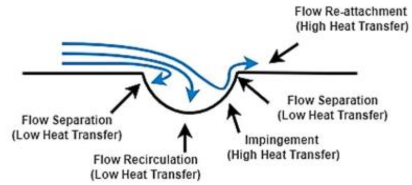

Figure 4
Flow Mechanism Responsible for Heat Transfer Improvement by Dimples

In order to improve the thermal efficiency of a gas turbine engine, the turbine inlet temperature is constantly increased until high thermal loads occur on the turbine blades. Therefore, the designs being worked on must be capable of improving the performance and protection of the gas turbine engine. The addition of rib turbulators increases the overall internal convection heat transfer coefficient, causing a corresponding drop in the component metal temperature [7].

THE PURPOSE OF THE STUDY

For many decades, numerous experiments have been carried out on turbulators on blades and blades in stationary channels. The project will be carried out with two geometric models designed in SolidWorks to compare the results of numerical simulations using the CFD system with Flow Simulation in SolidWorks. The main data will be worked in conjunction with references from previous projects such as [8] that have experimented with real-world temperature distribution along the channel and airflow behavior, see Table 1. The purpose is to be able to analyze and compare which of the two methods seen in Figure 5, Case A and Case B, are ideal when creating a new design.

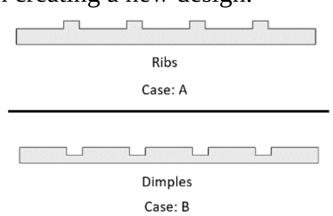

Figure 5
Rib and Dimple Configurations

Table 1
Property of Air

Table A-15 Properties of air at a pressure of 1atm.		
P=	0.199	kg/m^3
K=	0.09599	w/m*c
dt=	0.000391	m^2/s
μ=	5.82E-05	kg/m*s
v=	0.000292	m^2/s
Pr	0.7478	

The material used in the channels is 6061 Aluminum with a rectangular geometry as seen in Figure 6, and dimensions of 0.52 meters long, 0.15 meters wide and 0.05 meters high.

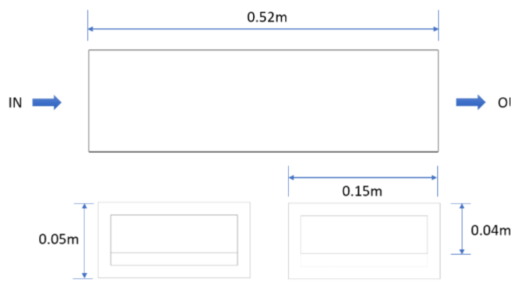

Figure 6
Dimensions of the Cooling Channel (Dimensions are in Meters)

The two stages of the Blades Case A and Case B as seen in Figure 7, will have the same parameters using the Reynold number. The idea is to develop a method to predict heat transfer. The main requirement is the analysis and simulation of continuous processes with rotation and rib and dimple turbulators. Boundary conditions are assumed. All walls of the two-pass square channel are assumed to be constant heat flux. Non-slip speed conditions apply to all walls. The inlet velocity is based on the Reynolds number, a uniform velocity and the air temperature is set at the inlet and a pressure condition is chosen at the outlet. In Table 2, below we can see the Numbers according to the parameters that we will be using.

The increase in heat transfer depends mainly on the aspect ratio of the duct, the flow Reynolds Number and configuration of the ribs or dimple. Figure 8 shows the configuration of a rib duct and the important parameters of the rib, such as the height of the rib - e, the height of the rib, angle, and the length from rib to rib, see Table 3.

Table 2
Assumption Parameter

Re"1"=	5000	
Re"2"=	10000	
T "Solid:	1500	k
T "Inlet"=	295	K
Q=	2000	W/m^2

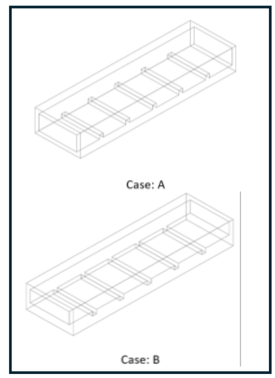

Figure 7
Guide vane details: (Case A) Ribs, (Case B) Dimples

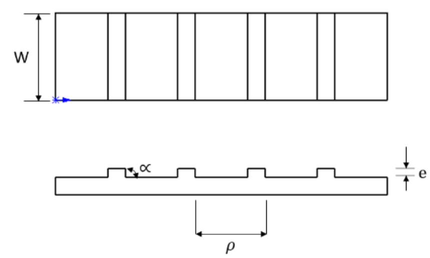

Figure 8
Coolant Channel in Turbine Airfoil and Internal Rib Arrangement

Table 3
Numerical Volume Parameter

Rib to rib pitch (P)	0.09	m
Channel Width (W)	0.13	m
Rib Height (e)	0.50	m
Channel Orientation	90˚	

RESEARCH METHODOLOGY

The Reynolds number is defined as:

$$Re = \frac{\rho V D_h}{\mu} = \frac{4\dot{m}}{\mu P e} \qquad (1)$$

where Dh is the hydraulic diameter of the channel and μ is the dynamic viscosity of the coolant.

The heat input to the system is electrical energy. It can be defined by (2). The flux, Q, was calculated using the energy input, Q", and the heat transfer area, A.

$$Q = R \times I^2 \qquad (2)$$
$$Q^" = \frac{Q}{A}$$

The local heat transfer coefficient (h) is defined as:

$$h = \frac{Q}{T_S - T_\infty} \qquad (3)$$

The local Nusselt number (Nu) is used to normalize the local heat transfer coefficient (h) and can be written as:

$$Nu = \frac{h\, D_h}{k} \qquad (4)$$

The Nusselt number (Nu_o) for fully developed turbulent flow in a smooth tube is defined as:

$$Nu_o = 0.023\, Re^{0.8}\, Pr^{0.4} \qquad (5)$$

The Fanning friction factor in the ribbed canals is defined as:

$$f = \frac{\Delta PD}{(\frac{L}{Dh})(\frac{1}{2pu^2})} \qquad (6)$$

The friction factor obtained for a fully developed flow in a smooth tube is expressed as:

$$f_o = 0.316Re^{-0.25} \qquad (7)$$

The thermal performance factor (g) is defined as:

$$\eta = (\frac{N_U}{N_{U_O}})/(\frac{f}{f_o})^{1/3} \qquad (8)$$

Numerical Simulation Configuration

The same number parameters were implemented for the different studies. In this numerical project, we worked with the Flow Simulation software using SolidWorks to obtain the necessary data for this study considering turbulent flows with a Heat generation rate in the channel. The inlet temperature is room temperature. Air was considered as the working fluid with segregated flow conditions. A constant heat flux of 2000 W/m2 was applied to the lower surface of the cooling channel and a solid temperature of 1500k. A turbulent velocity distribution was set for the inlet at 1000 rpm and a pressure outlet was set at the outlet. The study was carried out with two Reynolds numbers, Re=5000 and Re=10000.

RESULTS AND DISCUSSION

The results of this project were discussed with numerical work. The numerical results presented in Table 4 indicate the temperature, flow, and thermal performance of the cooling channel. In the different simulations we can observe a good representation in the different cooling channels.

Table 4
Data Obtained

Hydraulic Diameter "Dh"=	0.08	m
Channel length "L"=	0.52	m
Area "A":	0.30	m^2
Gravity "G"	9.80	m/s^2
V Int "1"=	1.62	m/s
V Int"2"=	3.24	m/s
Tout "1"=	1684.73	k
Tout "2"=	1716.76	k

Heat transfer phenomena along the cooling channel in Figure 9, indicate the temperature distribution along the cooling channel under steady conditions. For both stationary and rotational cases, the temperature is lower at the inlet since ambient air barely enters the cooling channel. As the air passes through the cooling channel, it extracts heat and the outlet temperature increases.

Figure 10 represents the different velocity trajectories in the cooling system with different types of Re=5,000 and Re=10,000. Figure 9 shows the cooling flow characteristics with the same inlet temperature and Heat Generation number. In the simulations it could be seen that the Ribbed stationary cooling ducts with a Re=5,000 had more turbulence than the other cases. The vortex rolls and movement were generated in both cases of Ribs and Dimples due to the curvature between each one, but in the channels the air flow was maintained throughout the curvature region as seen in Figure 10.

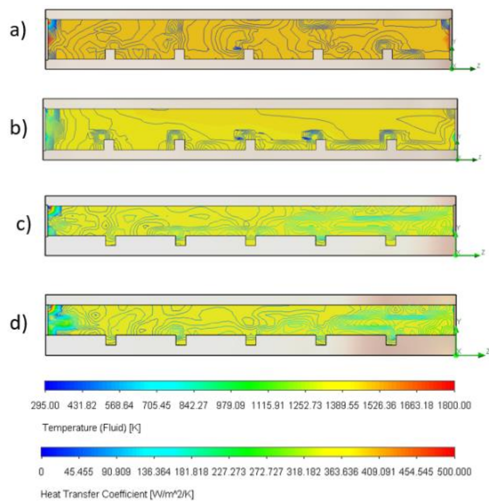

Figure 9
Temperature and Heat Transfer Coefficient Distribution along the Cooling Channels at Re=5,000 for Case A), Case B) and Re=10,000 at Case C), Case D)

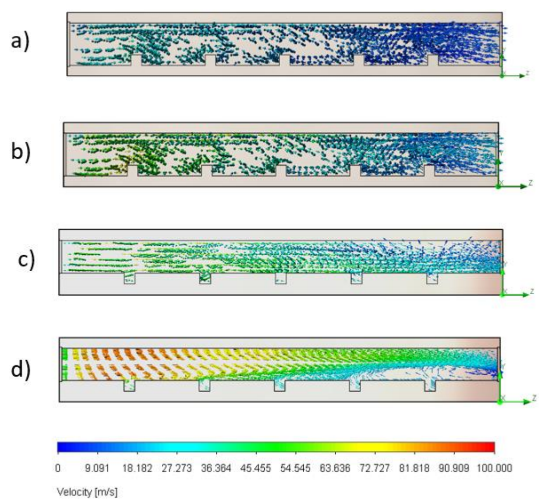

Figure 10
Velocity Distribution along the Cooling Channels at Re=5,000 for Case A), Case B) and Re=10,000 at Case C), Case D)

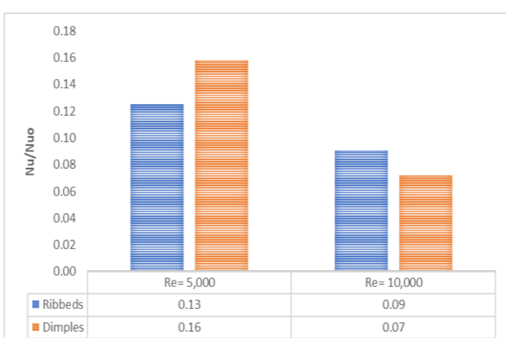

Figure 11
Normalized Nusselt Number

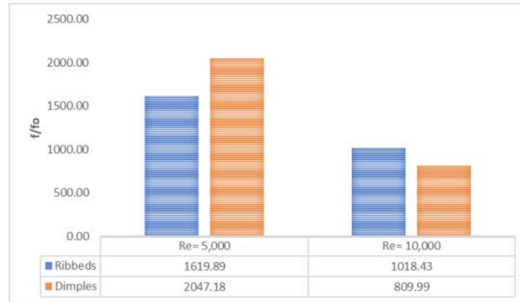

Figure 12
Normalized Fiction Factor

Figure 13
Different Temperatures

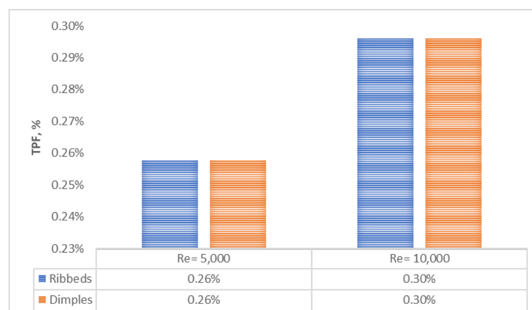

Figure 14
Thermal Performance Factor

COMPARISON OF COOLING DUCTS

Figure 11 and Figure 13 reflect the maximum heat transfers of the cooling channels with Ribs and Dimples with Re=5,000 and Re=10,000. In the different cases the difference in heat transfer was not very marked because a greater heat transfer was obtained in the channels with the Reynold Number of 5,000. Channels for Re=5,000 indicated a Nu/Nuo (0.13) for Rib and (0.16) for Dimple. In comparison with those with lower heat transfer, which in this case were those of Reynold Number of 10,000 with (0.09) for Rib and (0.07) for Dimple. We can say that the average heat transfer between the channels studied was (0.115). On the other hand, the pressure drop significantly affects the thermal performance of the cooling channel. In this project, as seen in Figure 12, the average friction factor of the cooling channels could be studied. The ribs and dimples for a Reynold Number of 10,000 could see the largest pressure drop among the 4 cases. The pressure drop phenomenon affects the thermal performance of the cooling channel, which is reflected in Figure 14. Whereas for the different Reynold Numbers the thermal performance of the channel is the same.

CONCLUSIONS

In this project we analyze the comparison of cooling channels with ribs and dimples. However, in the project it was possible to observe and analyze that there is not much difference in the channels of the different cases. In the project, two different models with two different orientations were demonstrated. The goal was to compare two channels with different internal models to analyze heat transfer and see which is more efficient when creating a design from scratch.

In the present study, the following conclusions were reached:

- With the ribbed cooling channel, we were able to see more changes in the system because it is more efficient when creating a design from scratch.
- It can be concluded that the increase in heat transfer and decrease in pressure drop depend on the geometry and aerodynamic design.
- In general, for our study, no significant change in heat transfer was observed because the geometry dimensions were very small, so the efficiency was almost the same for the different cases.

RECOMMENDATION

After the investigation, it was possible to observe and analyze the different results in the two models that we used. Due to this, the recommendations to follow are:

- It is recommended to study different models with larger geometry.
- Analyze the system with higher temperatures and be able to analyze if there are changes in drop pressure and efficiency.

REFERENCE

[1] Tamunobere, O. (2015). *Cooling of Gas Turbine Blade and Heat Transfer and Film Cooling on a Gas Turbine Blade and Shroud*. *1*.

[2] Saravani, M. S., Amano, R. S., DiPasquale, N. J., & Halmo, J. W. (2020). Turning guide vane effect on internal cooling of two-passage channel with parallel ribs. *Journal of Energy Resources Technology*, *142*(9). [Online] Available: https://doi.org/10.1115/1.4046731

[3] Nourin, F. N., & Amano, R. S. (2021). Review of gas turbine internal cooling improvement technology. *Journal of Energy Resources Technology*, *143*(8), 1–22. [Online] Available: https://doi.org/10.1115/1.4048865

[4] Jiaxu Yao, Ke Zhang, Wenbin He, Jiang Lei1, Jin Xu. (2023). *Investigation on Heat Transfer Distribution of Rotating Two-Pass Smooth and Ribbed Channels Under Aero-Engine Simulated Conditions* [Xi'an Jiaotong University]. [Online] Available: https://ezproxy.pupr.edu:2253/turbo_145_12_121002.pdf

[5] Sundberg, J. (2006). Heat Transfer Correlations for Gas Turbine Cooling. *asme*. http://liu.diva-portal.org/smash/get/diva2:21321/FULLTEXT01

[6] Nourin, F. N., Blum, B., & Amano, R. S. (2022). Evaluation of heat transfer enhancement on rotational gas turbine blade internal cooling channel with dimpled surface. *Journal of Energy Resources Technology-transactions of The Asme*, *144*(11). [Online] Available: https://doi.org/10.1115/1.4054288

[7] Dees, J. E., Bogard, D. G., Ledezma, G. A., Laskowski, G. M., & Tolpadi, A. K. (2012). Experimental measurements and computational predictions for an internally cooled simulated turbine vane with 90 degree rib turbulators. *Journal of turbomachinery*, *134*(6). https://doi.org/10.1115/1.4006282

[8] Rao, Y., Xu, Y., & Wan, C. (2012). An experimental and numerical study of flow and heat transfer in channels with PIN Fin-dimple and PIN FIN arrays. *Experimental Thermal and Fluid Science*, *38*, 237-247. [Online] Available: https://doi.org/10.1016/j.expthermflusci.2011.12.012