Advanced Internal Cooling Geometries - Double Wall Schemes With and Without Effect of Rotation

Srinath V. Ekkad

Associate Professor

Mechanical Engineering

Virginia Tech

sekkad@vt.edu

Collaborators: Dr. Diganta Narzary, Justin Lamont, Preston Stoakes, and Dr. Mary Anne Alvin (NETL)

Virginia Tech

Heat, Energy and Fluids Transport (HEFT) Lab

Programmatic Relevance

- IGCC plants play a key role in the future of DOE's Clean coal initiative by facilitating pre-combustion CO_2 capture from syngas
- Gas turbines used in IGCC are subjected to high thermal loads and high temperatures along with residual particulate and vapor contaminants which could potentially alter the life of precision engineered vanes and blades in the hot gas path
- The proposed research aims to develop physics based modeling tools to develop and predict new cooling strategies for hot components and provide effective cooling schemes with low coolant usage with direct impact on overall efficiency
- Work should directly impact materials development and coatings also

Virginia Tech

Heat, Energy and Fluids Transport (HEFT) Lab

Why Double Wall Cooling?

- Double wall cooling uses a thin gap between two walls to enhance heat transfer from the surface of turbine blades
- Double wall cooling increases area for heat transfer between cooling fluid and metal
- Impingement jets and modified surfaces can be used to increase heat transfer on the outer wall
- Nothing new about this concept – has been around for several decades

VirginiaTech

Heat, Energy and Fluids Transport (HEFT) Lab

Patents on Double Wall cooling

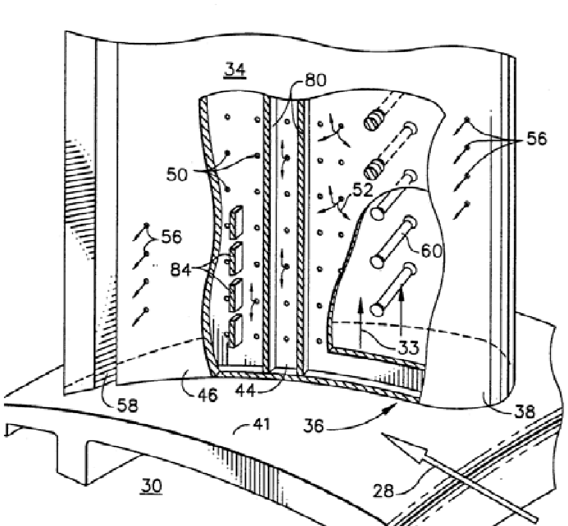
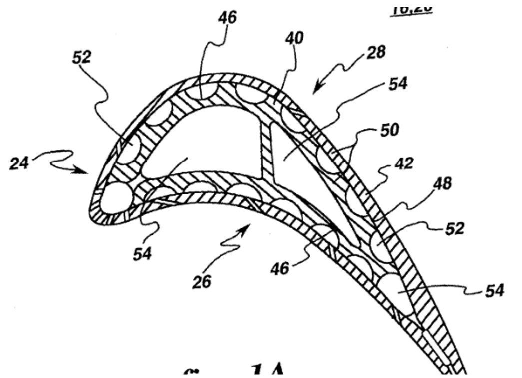
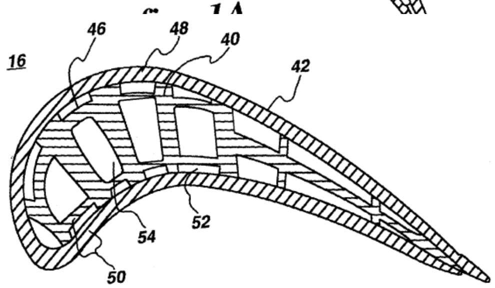


FIG. 6





ig. 2

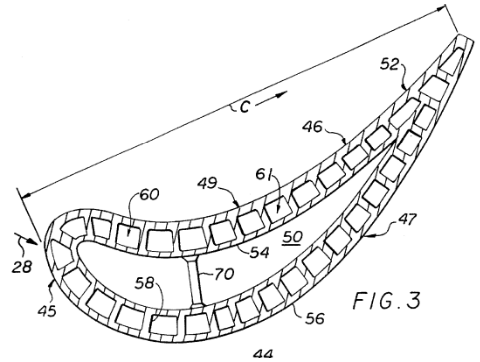


FIG. 3

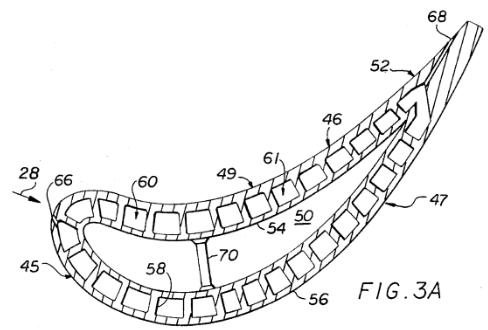
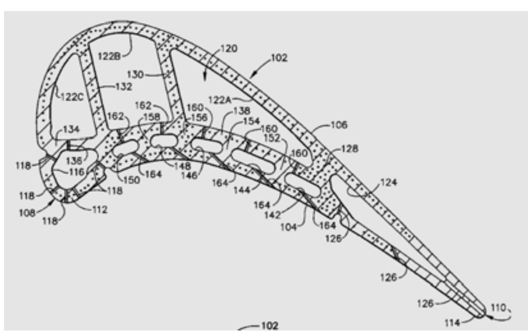
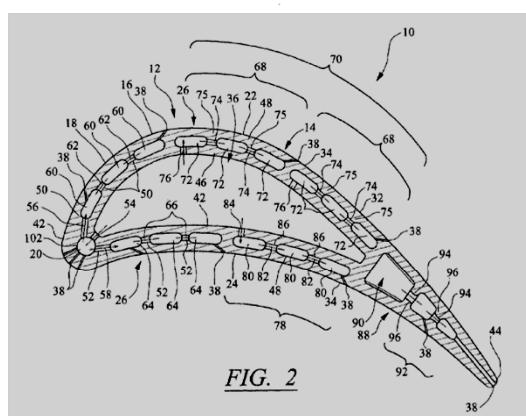
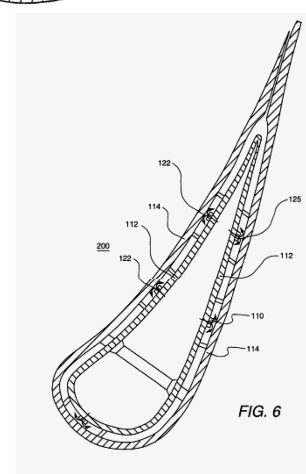


FIG. 3A



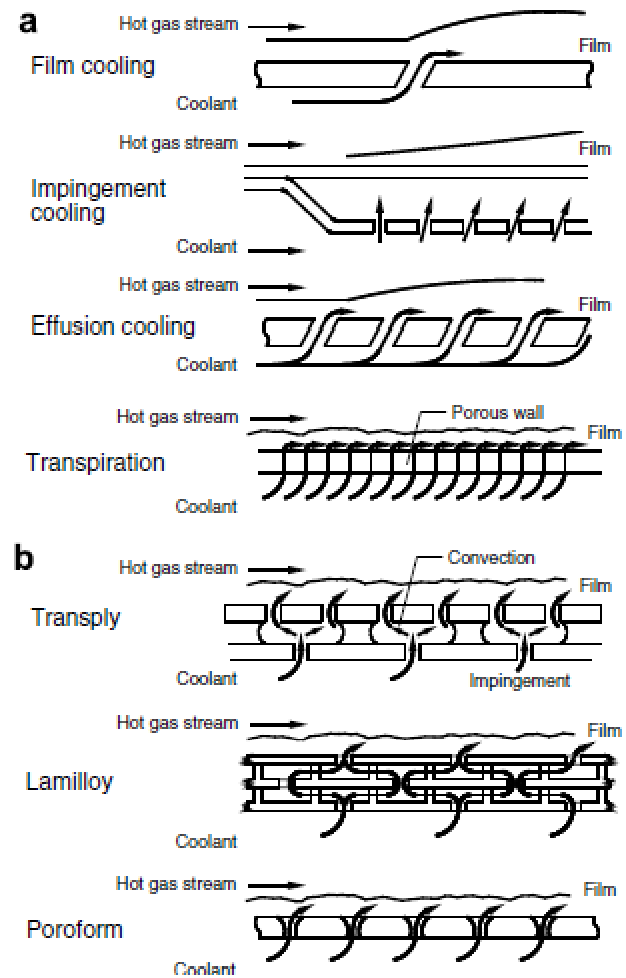




Patents by Bunker et al., Ishburg and Lee, Jackson et al., Liang, Melvin et al.

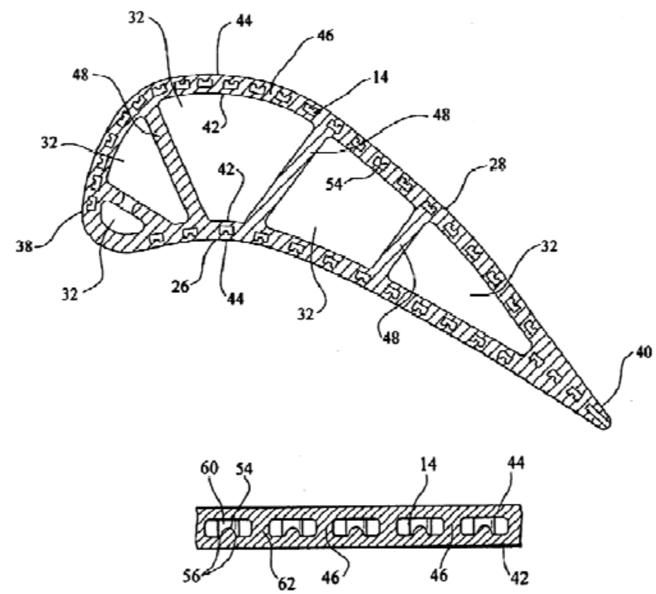
Other cooling options including double wall

- Network of orifices connected by small passages to create impingement areas and outward film cooling
- Cool air is force into double wall area through small passages and impinges on the outer wall
- Lamilloy© is a standard practice used By RR in North America (mature technology)



Another Existing Design

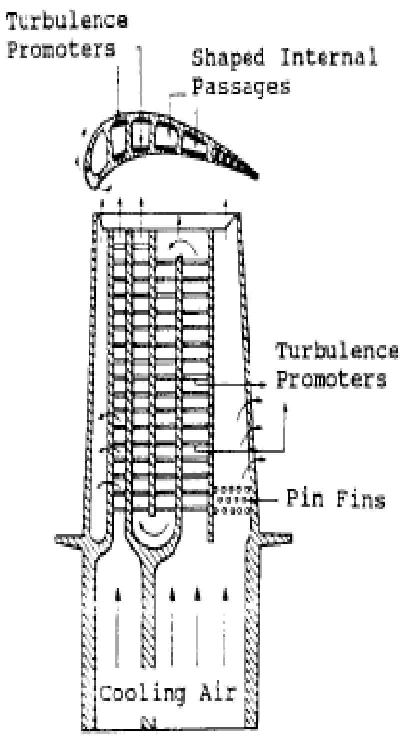
- System of pedestals and ribs used to increase turbulence and heat transfer
- Cooling gas exhausted through trailing edge of blade
- Design does not use impingement cooling



Liang G, inventor; 2004 Oct. 26. Cooling system for a turbine blade having a double outer wall. United States Patent US 6,808,367.

Motivation & Objectives

- Gas turbine blades need to be effectively cooled to increase component life and reduce maintenance costs. The level of cooling is always off-set by the amount of coolant used. Increased amount of coolant usage directly impacts the overall efficiency of the engine.
- Usage of double wall cooling schemes can reduce overall coolant usage by pushing coolant closer to the inside of the wall exposed to hot gas path.
- The enhanced heat transfer to the coolant through the thin wall and also due to high performance schemes such as impingement will greatly benefit overall and thermal efficiency of the system.
- The focus is:
  - to develop an optimization methodology to determine the most effective double wall/near wall scheme for turbine airfoil cooling. These cooling geometries will be optimized for the highest heat transfer enhancement with low pressure drop with optimization software and CFD.
  - to study the effect of rotation on double wall cooling geometries to ensure applicability to rotating blades
  - to study the performance of the optimized geometry working with OEMs to compare cooling effectiveness of double wall geometries compared to current cooling schemes



Typical turbine blade internal convection cooling configuration (Han et. al. (1986).

Project Summary

- Objective:
  - Explore the use of double wall cooling in turbine blades, using impingement cooling and combining with other standard heat transfer enhancement techniques
  - Develop design methodologies for optimized double wall cooling designs for industry usage
- Procedure:
  - Use CFD to explore the effectiveness of impingement cooling in channel flow
  - Use CFD to optimize design pattern of impingement holes in channel flow
  - Experimentally determine performance of optimized configuration in comparison to current standard designs
  - Also experimentally study effect of rotation on double wall cooling designs with intention to optimize with rotational effects

Stationary Optimization Study

Virginia Tech

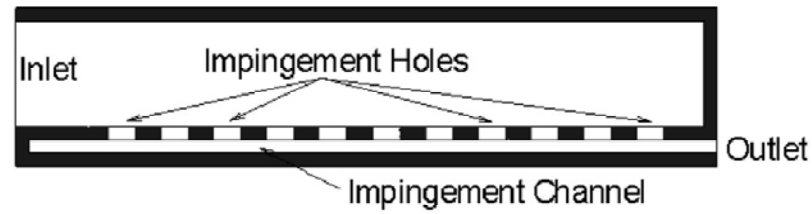
Heat, Energy and Fluids Transport (HEFT) Lab

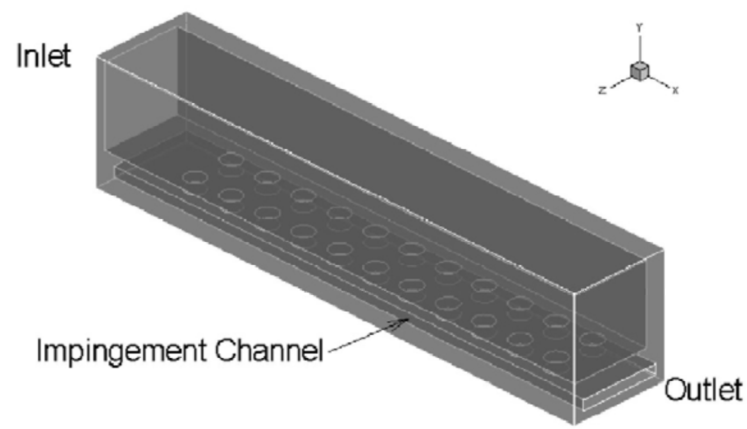
Start with a Simplified Geometry

- The simplified geometry consists of main channel with an impingement channel connected by an array of impingement jets
- Main channel measures 1” x 1” for all test sections
- Impingement channel dimensions and length of test section varies between the cases

Geometry

- Impingement jet array is 2 x N, N varies in the optimization study
- Impingement channel confines the spent coolant to exit opening opposite inlet





Procedure

- Optimize geometry using Computational Fluid Dynamics (CFD)
  - Independent parameters chosen based on studies found in literature
- Top designs from CFD optimization will be built for experiments
  - Test sections constructed from aluminum to include conduction effects

Optimization

- Four parameters varied to determine effect of each on heat transfer and pumping power
  - Hole Diameter (D)
  - Jet-to-Jet Spacing Ratio (L/D)
  - Jet-to-Wall Spacing Ratio (H/D)
  - Number of rows of holes (N)
- Air flow rate varied with N and D to maintain a jet Reynolds Number of 10,000

Optimization

- A total of 256 test section designs were considered by choosing four design points for each parameter
- Total length, L_T, held constant when N is varied

Parameter	Range	
	Min	Max
D	1/32"	1/4"
	(0.794 mm)	(6.35 mm)
L/D	2	5
H/D	0.5	4
N	5	11

$$L_T = 11 \cdot D \cdot L/D + 1\,\text{in}$$

Optimization

- Test section designs ranked based on heat transfer coefficient along the impingement surface, *h*, and pumping power for fluid, P_P
- The two parameters are combined to form a performance, *Per*, parameter to easily compare the test section designs

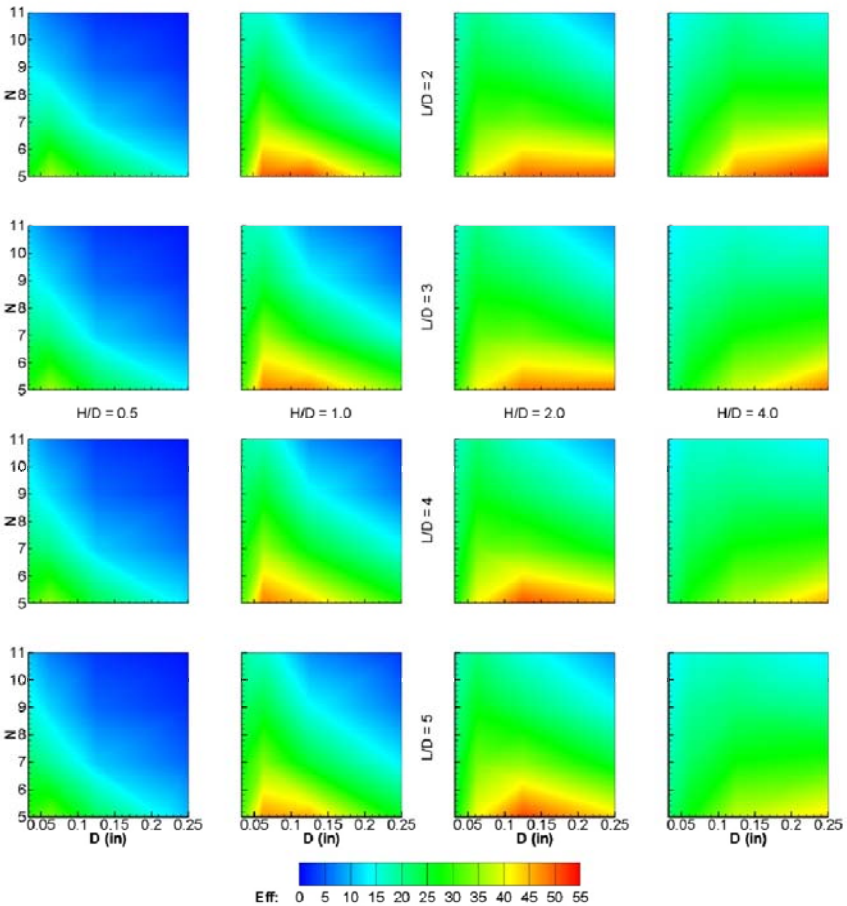
$$Per = \frac{\bar{h}/\bar{h}_0}{P_P/P_{P,0}}$$

- All test sections compared to baseline design

Results

- Number of rows of holes, *N*, and Jet-to-Wall spacing ratio, *H/D*, appear to have the largest affect on the performance parameter
- Jet-to-Jet spacing ratio, *L/D*, appears to have very little affect on the performance parameter
- The top 10 performing designs were built to validate CFD study

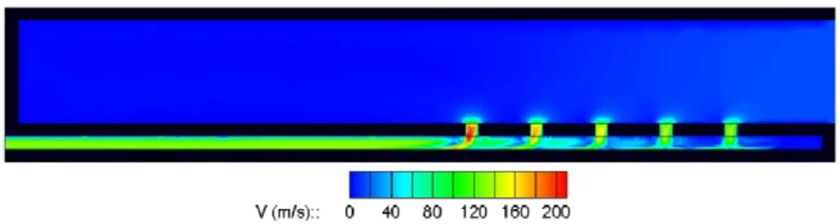
Results



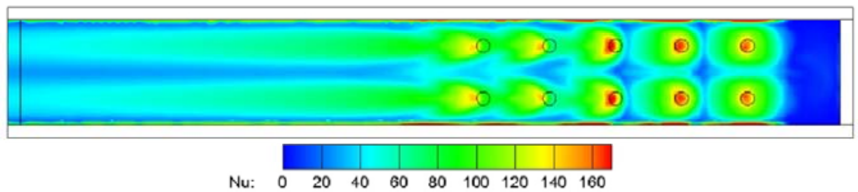
Rank	D (in)	L/D	H/D	N	Per
1	0.125	5	2	5	51.7
2	0.125	2	1	5	51.6
3	0.125	4	2	5	51.4
4	0.0625	2	1	5	50.2
5	0.25	2	2	5	50.2
6	0.125	2	2	5	49.9
7	0.25	3	2	5	49.3
8	0.125	3	2	5	48.9
9	0.0625	4	1	5	48.9
10	0.0625	3	1	5	48.8

Results for best design

- Velocity contours show higher velocity through last row of holes
  - Final jet is deflected by cross flow of exhaust gas
- Highest Nusselt number values occur under impingement jets
  - Exhaust gas appears to exit in two streams inline with impingement jets



Contours of velocity at plane intersecting impingement jets



Contours of Nusselt Number on impingement surface

Future Work

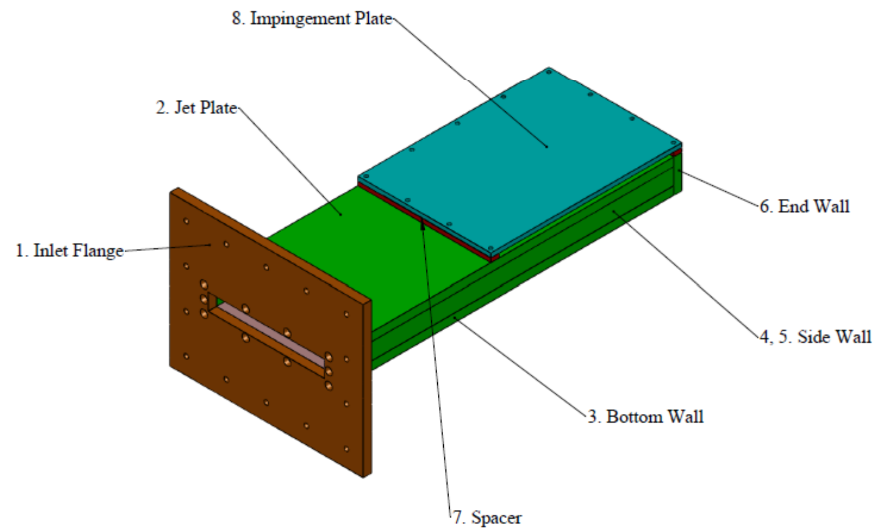
- Experiments are being conducted to validate CFD simulations on initial test section
- Turbulators can be added to the impingement channel to disrupt flow of exhaust gases
  - The flow disruption should allow for a more even heat transfer distribution, as well as reduce jet deflection due to cross flow
  - Pin fin turbulators will increase amount of conduction into the main channel and further increase area for convective heat transfer

VirginiaTech

Heat, Energy and Fluids Transport (HEFT) Lab

Multiple Array Impingement for Low H/d double wall cases

- Study effect of exhaust gas cross flow when jet-to-wall spacing ratio is below 1
- Initial CFD study base on jet-to-jet spacing ratio
- Compared inline and staggered arrangement of jets

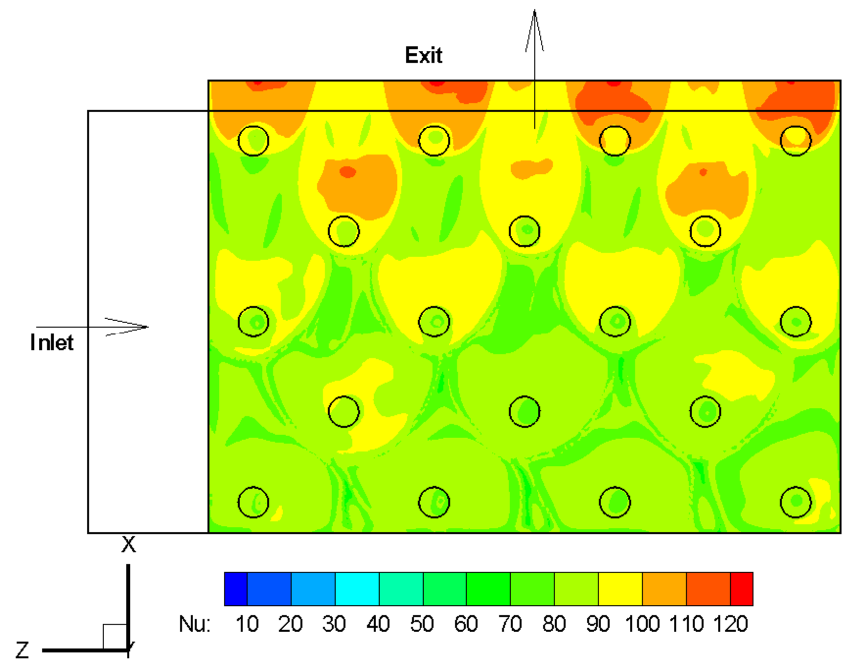


Virginia Tech

Heat, Energy and Fluids Transport (HEFT) Lab

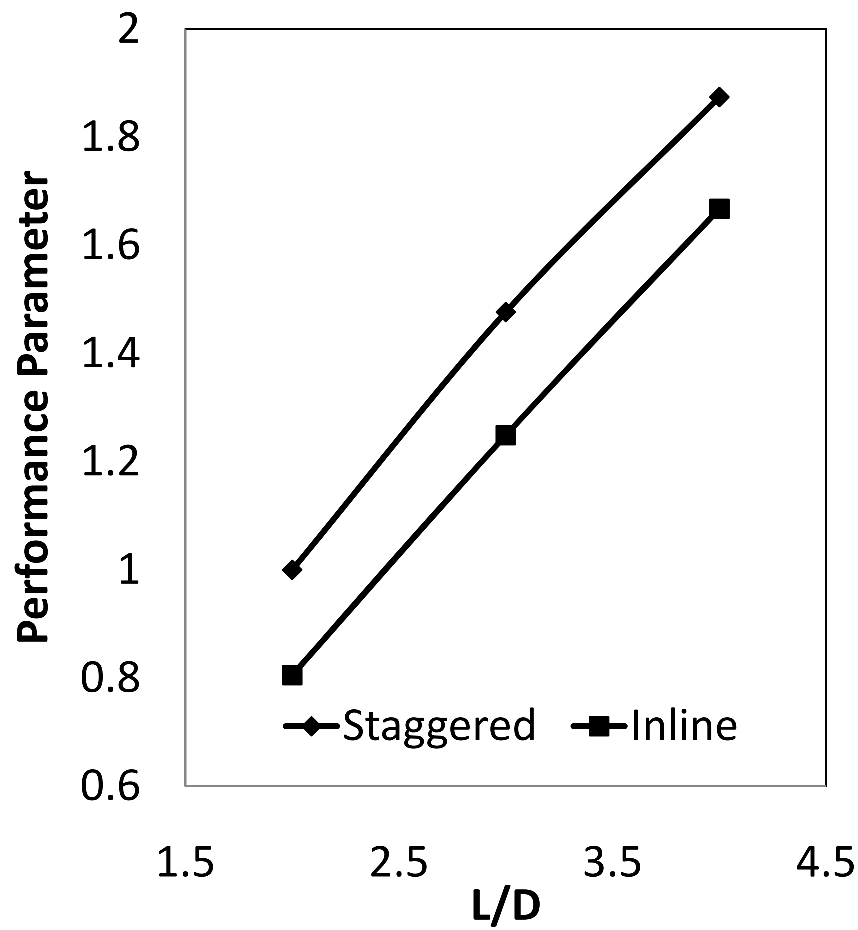
Multiple Array Impingement for Low H/d double wall cases

- Heat transfer appears to be increased by the wall jet downstream of the impingement jet and not by the impingement jet
- Undeveloped core jet does not appear to increase heat transfer



Multiple Array Impingement for Low H/d double wall cases

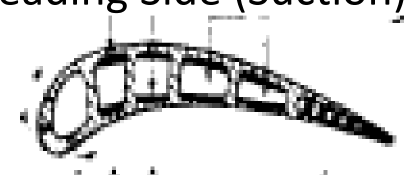
- Staggered arrangement of jets performed better at all jet-to-jet spacing ratios
- Higher jet-to-jet spacing ratio appears to increase performance of design
- Future study will explore larger jet-to-jet spacing ratios



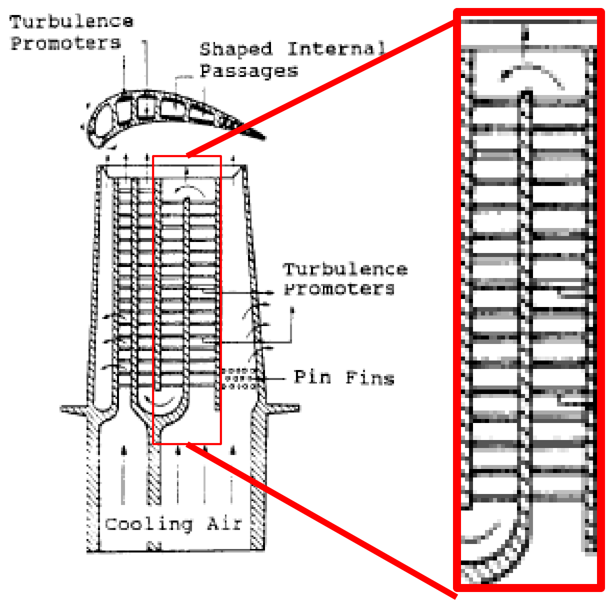
Rotational Effects

A simplified geometry was tested to determine the effects of rotation on blade internal coolant channel flow

Leading Side (Suction)

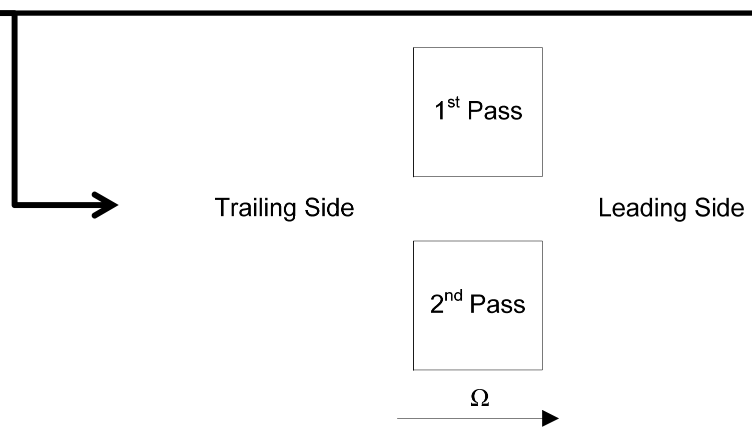


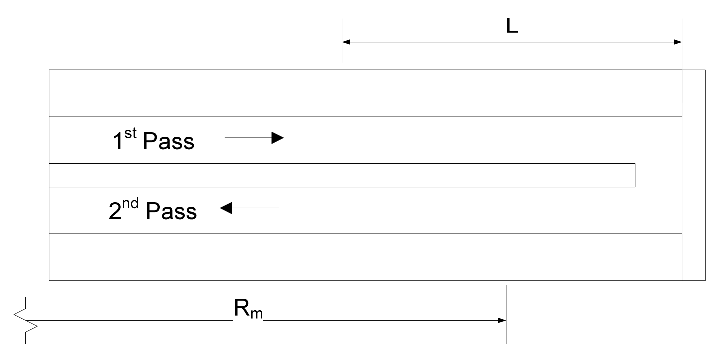
Trailing Side (Pressure)



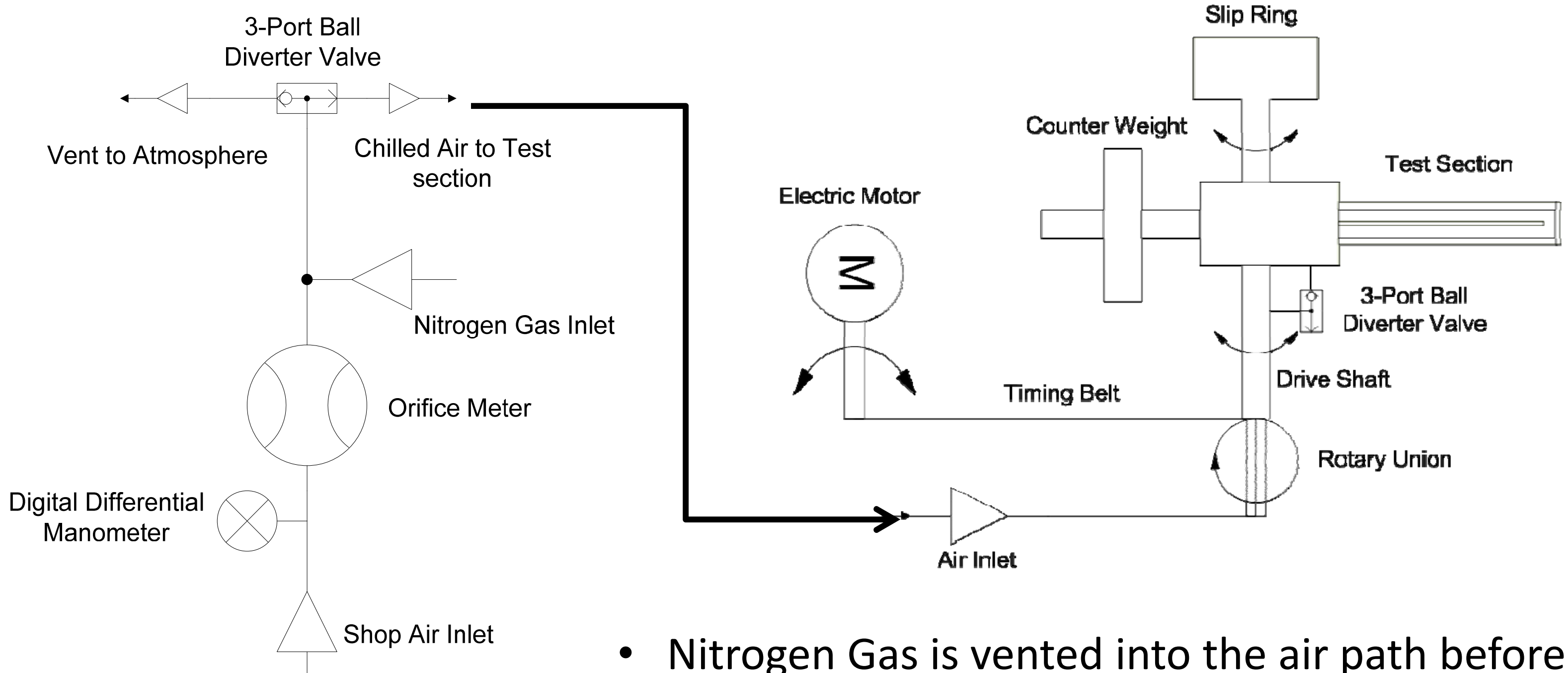
Typical turbine blade internal convection cooling configuration (Han et. al. (1986).

A typical coolant passage from turbine blade is modeled in this experiment with simple square geometry.



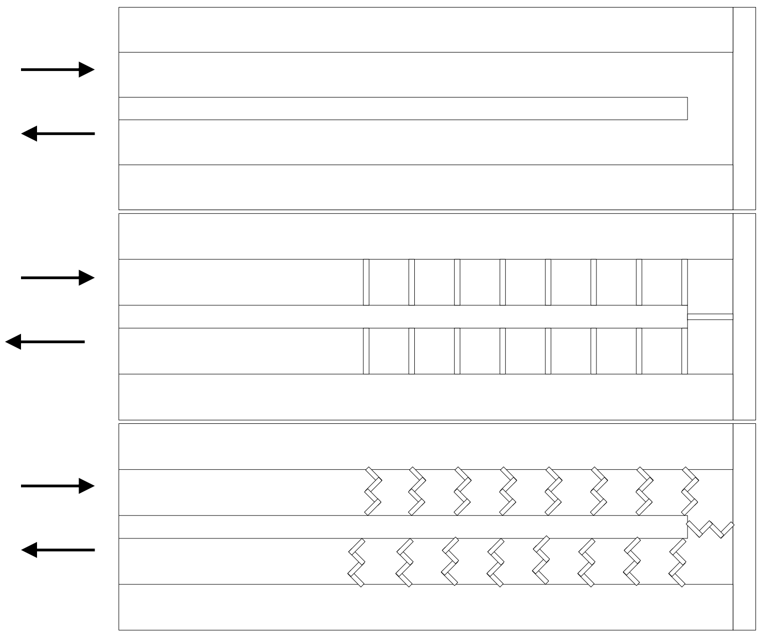


The rotating rig spins interchangeable test sections at desired speeds while low temperature air is injected



- Nitrogen Gas is vented into the air path before testing to chill components. This drops the temperature of the air before reaching the test section during a test.
- A camera is mounted to the test section to view the color change of the liquid crystals. One side is filmed at a time, so the motor rotation direction is reversed when filming either the leading or trailing side.

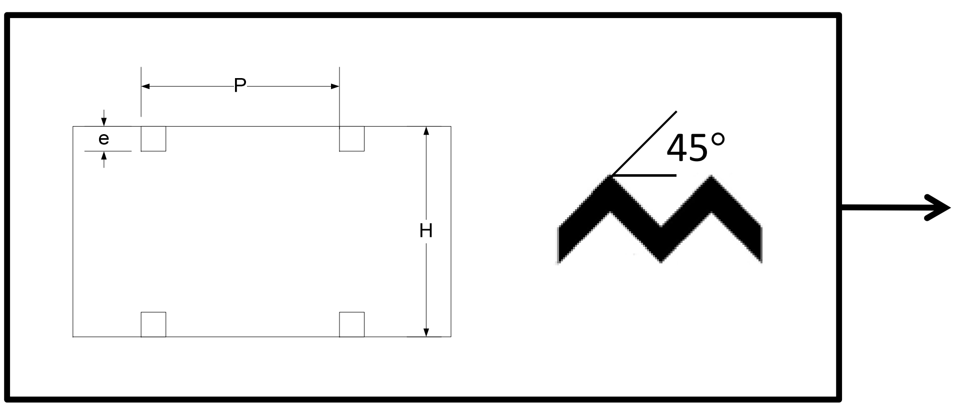
The effect of rib type in rotational two-pass channels was explored using a transient liquid crystal technique



Smooth Wall

90° Ribs

W-shape Ribs



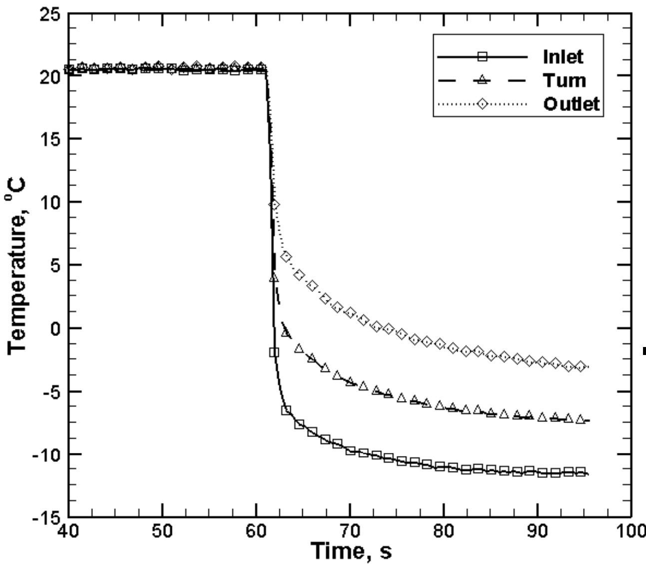
Pitch-to-rib height ratio (P/e)=8
Blockage ratio (H/e) = 8

- Smooth wall, 90° Ribbed, and W-shape ribbed walls were explored in the study.

- Each case was held at a
- Re = 16,000 and a rotational speed of 250 rpm
- (Rotation number=0.08).
- Inlet density ratio = 0.10

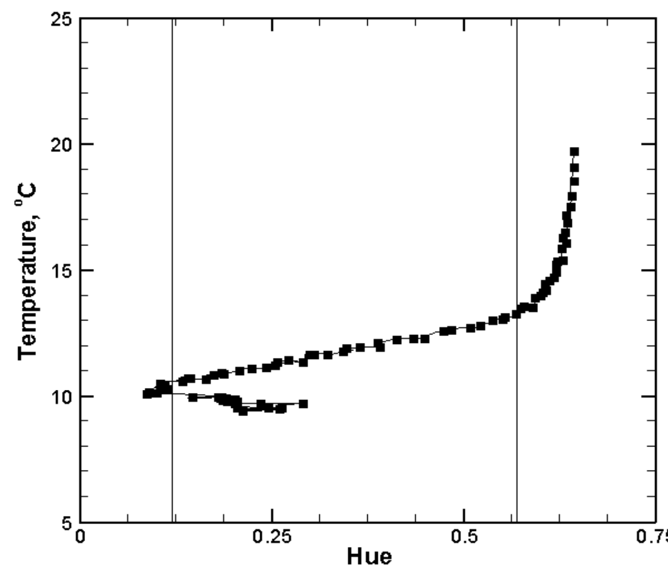
- Results are reported for the trailing side, leading Side, and stationary. The sides are compared to the stationary case.

The hue from the liquid crystal is calibrated with temperature to calculate the heat transfer on the surfaces



Typical Temperature response

Video



Calibration Curve

A curve fit generates an algebraic expression for temperature with respect to Hue which is applied to the rest of the surface

A semi-infinite model is used to determine the local convective heat transfer on the channel walls

- A transient liquid crystal technique is used to determine the heat transfer.
- For the given test section, the semi-infinite solid model is valid if the experiment times does not exceed 25 minutes.
- Average test time is less than a minute.
- After calibrating the liquid crystal hue with temperature, wall temperatures for each pixel at each frame in the video is calculated.

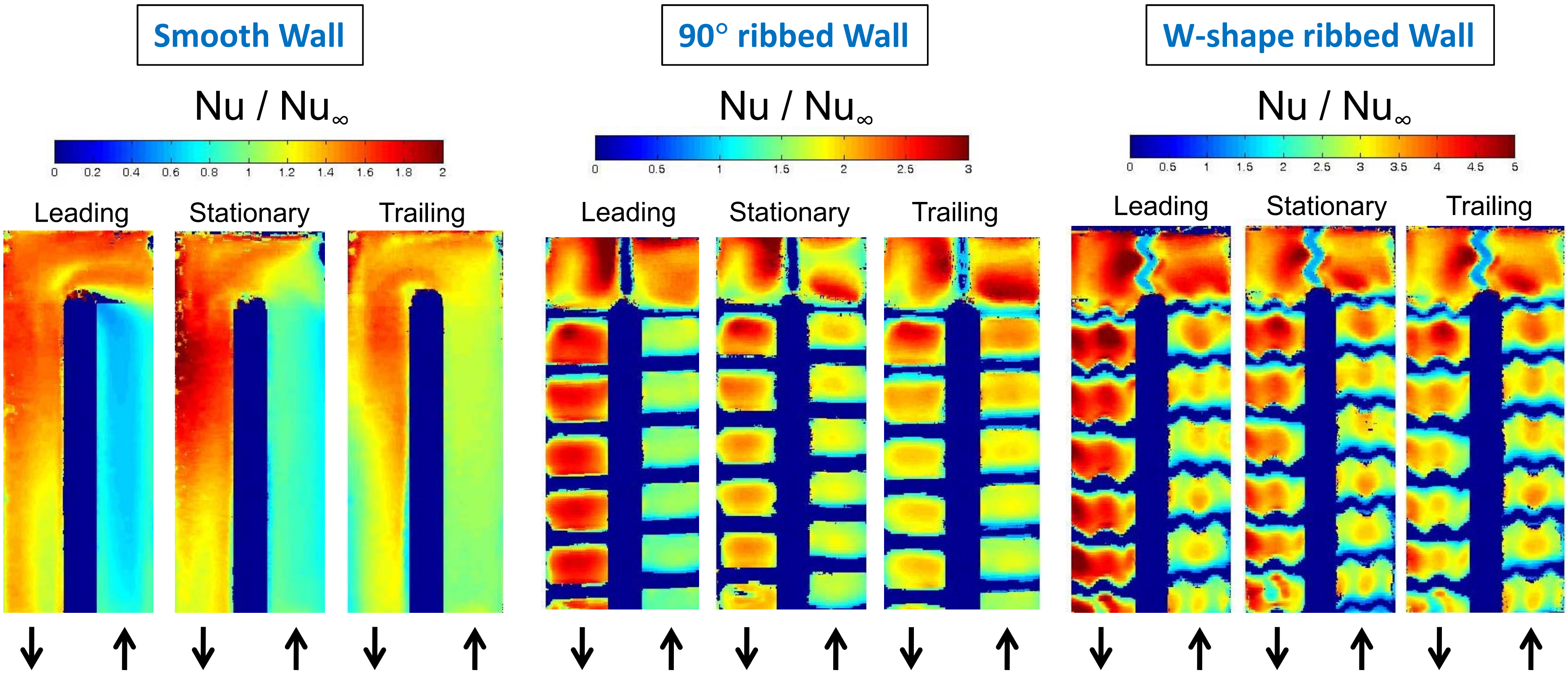
Temperature is calculated at the surface (x=0) so the mathematical model used reduces to:

$$\frac{T(0,t)-T_i}{T_\infty - T_i} = 1 - \left[\exp\left(\frac{h^2 \alpha t}{k^2}\right)\right]\left[erfc\left(\frac{h\sqrt{\alpha t}}{k}\right)\right]$$

T_∞ : is the averaged bulk temperature from the inlet and outlet
T_i : is the initial wall temperature
T(0,t): is the wall temperature
α: is the thermal diffusivity of the acrylic
t: is the time

We know all parameters, so we are able to solve for h numerically

The liquid crystal creates a continuous plot of heat transfer on the surface of the coolant channel

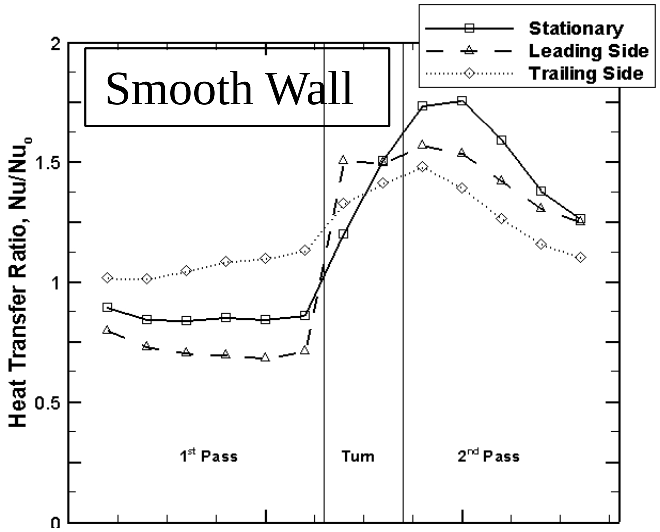


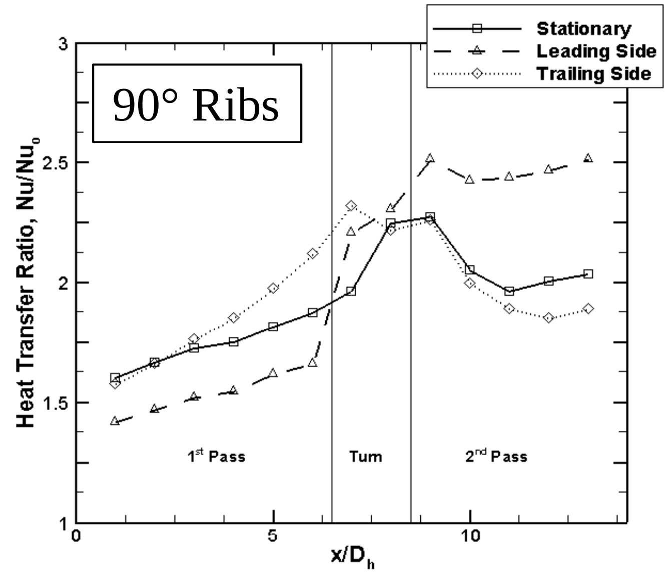
The W-ribs create the highest heat transfer in the channel. The smooth wall channel has the lowest heat transfer.

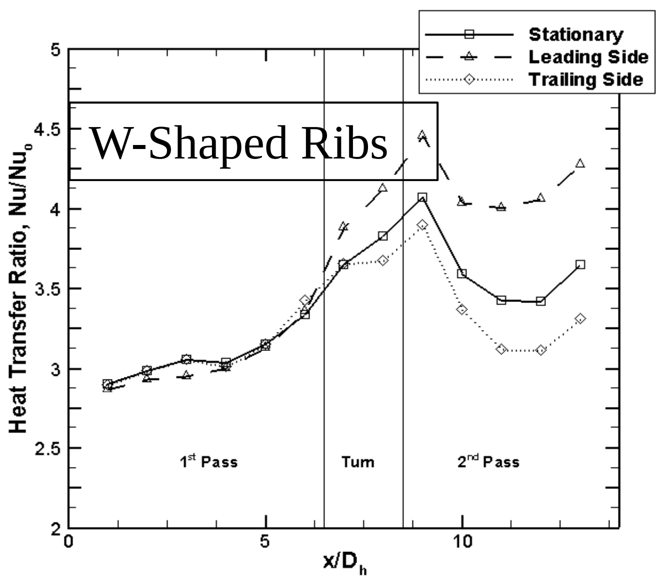
VirginiaTech

Heat, Energy and Fluids Transport (HEFT) Lab

Area averages in heat transfer along the length of the channel give estimates to the percent differences







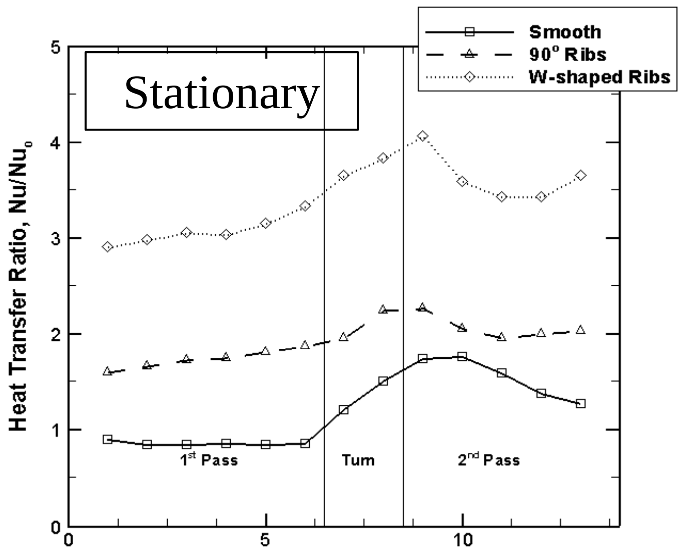
% Increases

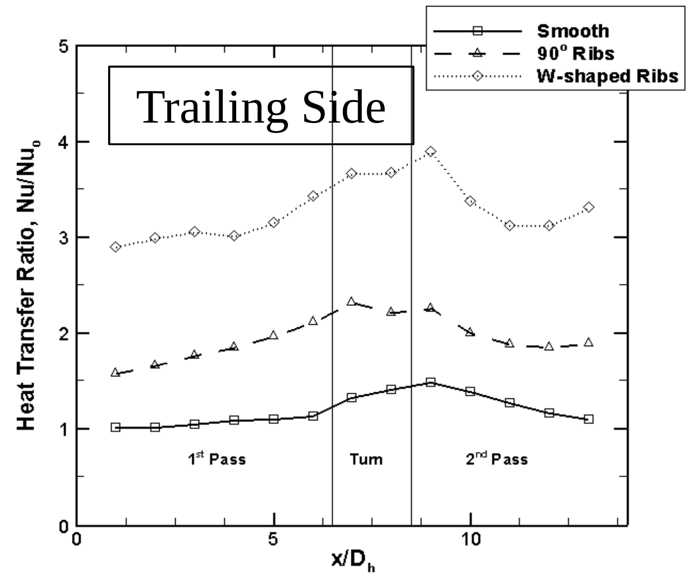
		Smooth Wall	90° Ribs	W Ribs
Trailing Side	1st Pass	32	13	0.1
	2nd Pass	-21	-7	-9.3
Leading Side	1st Pass	-19	-13	-3.3
	2nd Pass	3	24	17.2

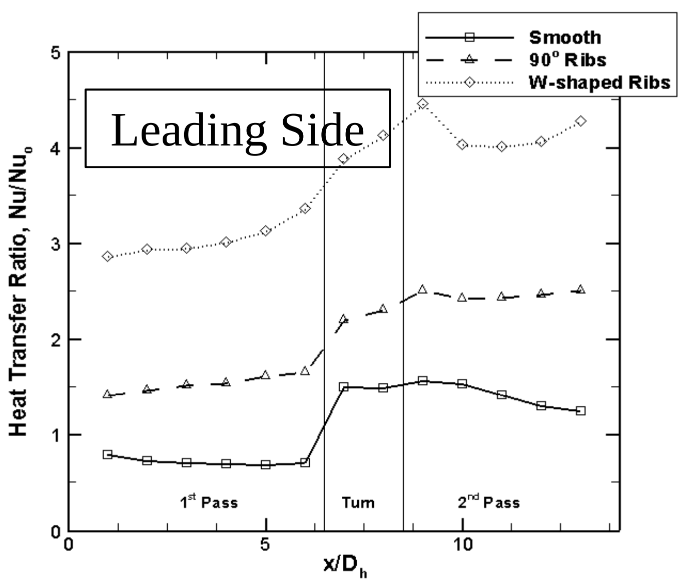
VirginiaTech

Heat, Energy and Fluids Transport (HEFT) Lab

The rib types are directly compared for the stationary, trailing side, and leading side



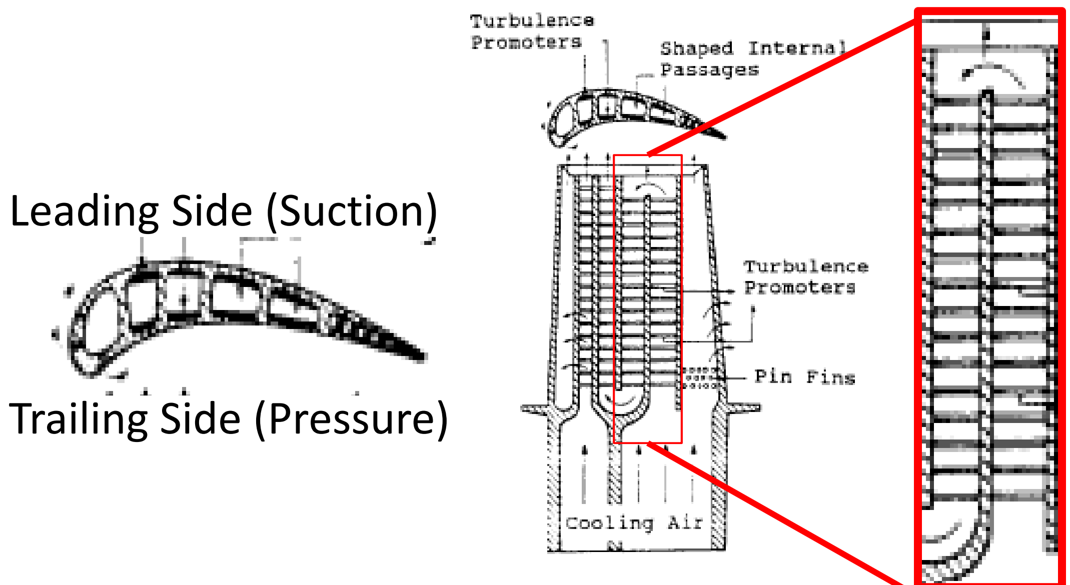




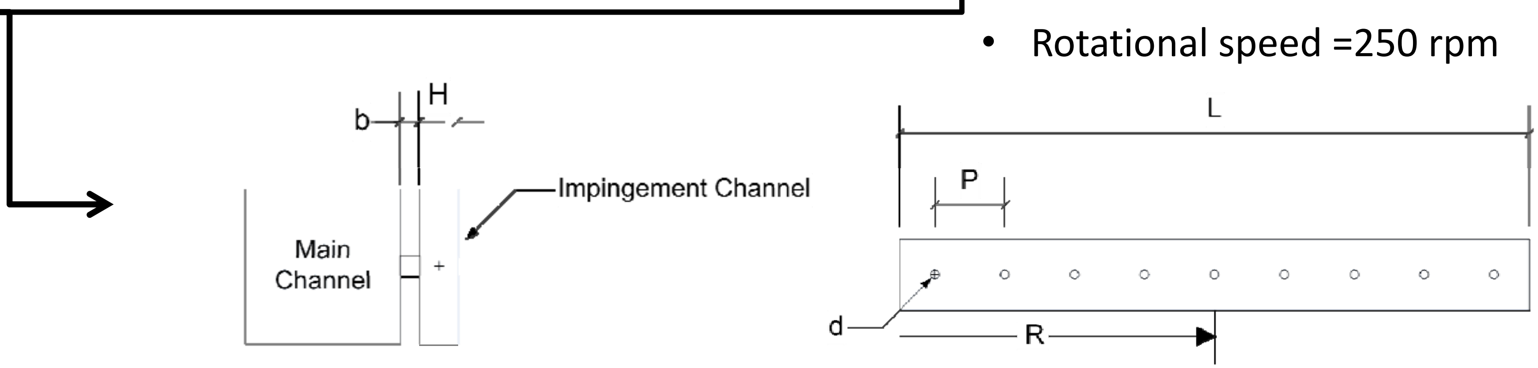
% Increases

		Stationary	Trailing Side	Leading Side
1st Pass	90° Ribs	104	71	116
	W-Ribs	260	190	325
2nd Pass	90° Ribs	35	55	76
	W-Ribs	138	164	197

Jet Impingement cooling is an alternative to rib roughened walls to create high heat transfer

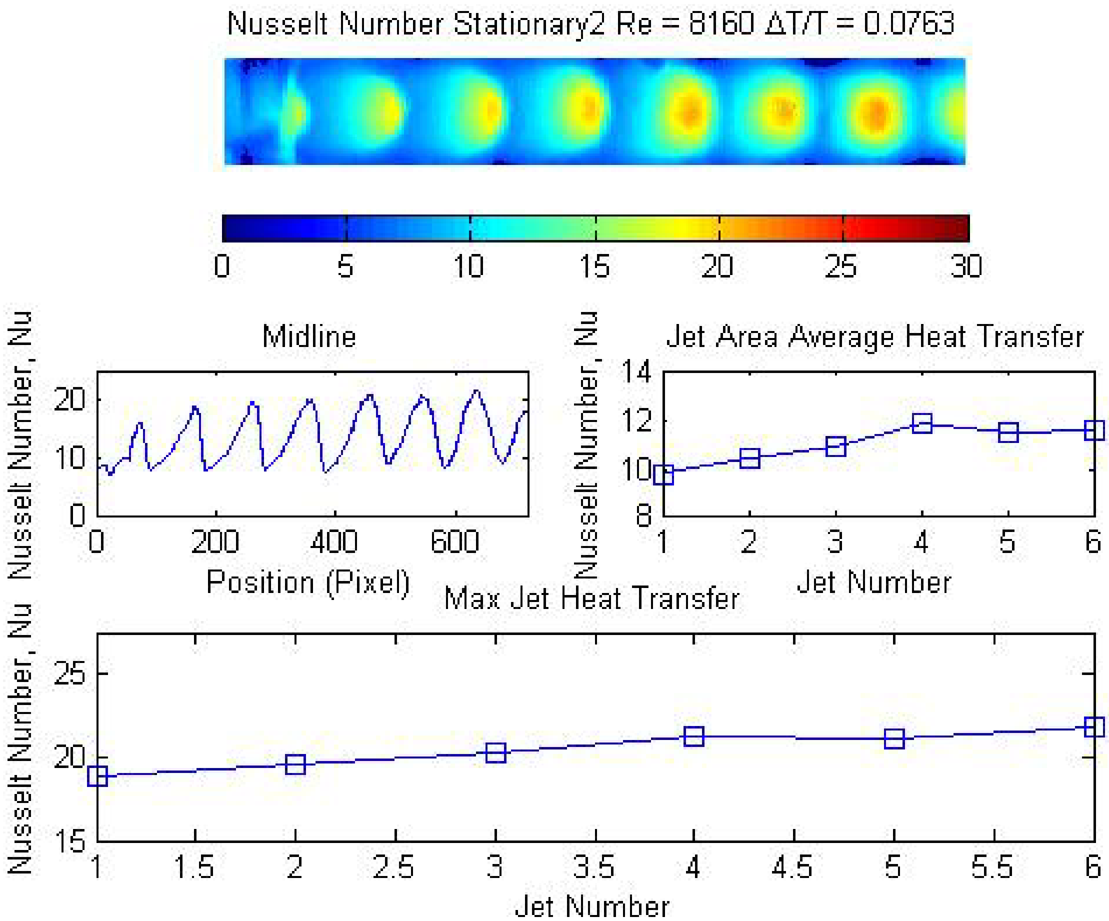


Typical turbine blade internal convection cooling configuration (Han et. al. (1986).



- Instead of a two pass channel, a radially outward channel with impingement is studied under rotation
- Current results show impingement channel height-to-jet diameter ratio (H/d) = 2
- Pitch-to-jet diameter ratio
- (P/d) = 8
- Jet length-to-jet diameter ratio
- (b/d)= 1
- Rotational speed =250 rpm

Preliminary stationary results for jet impingement cooling with crossflow effect

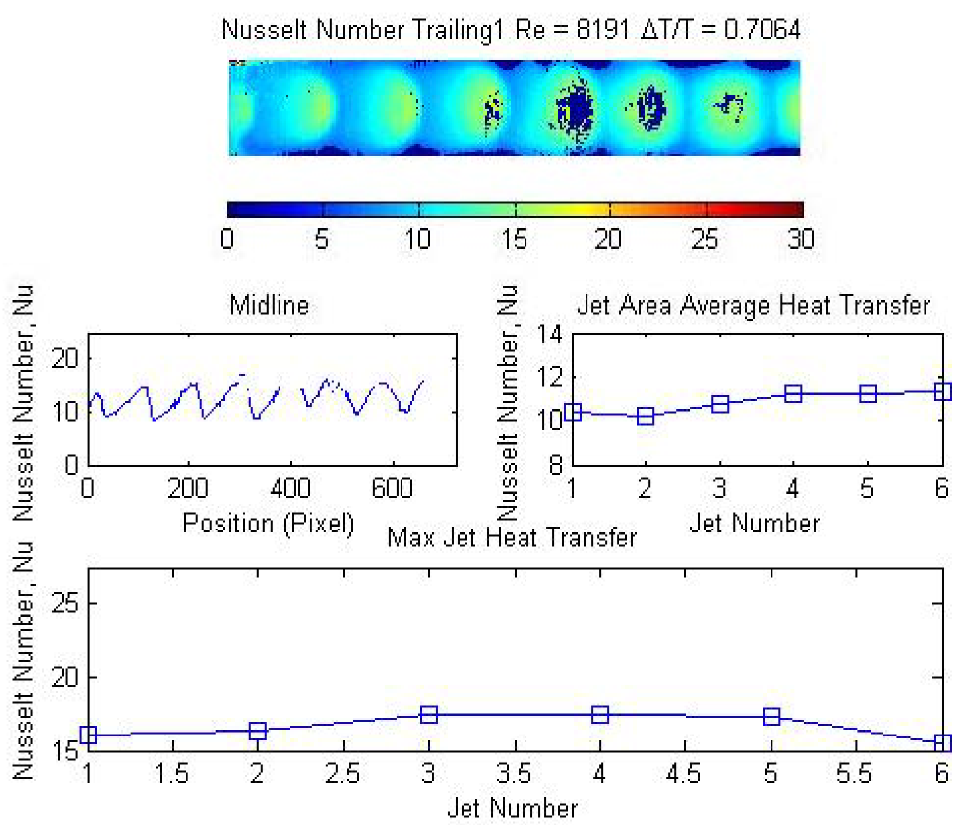


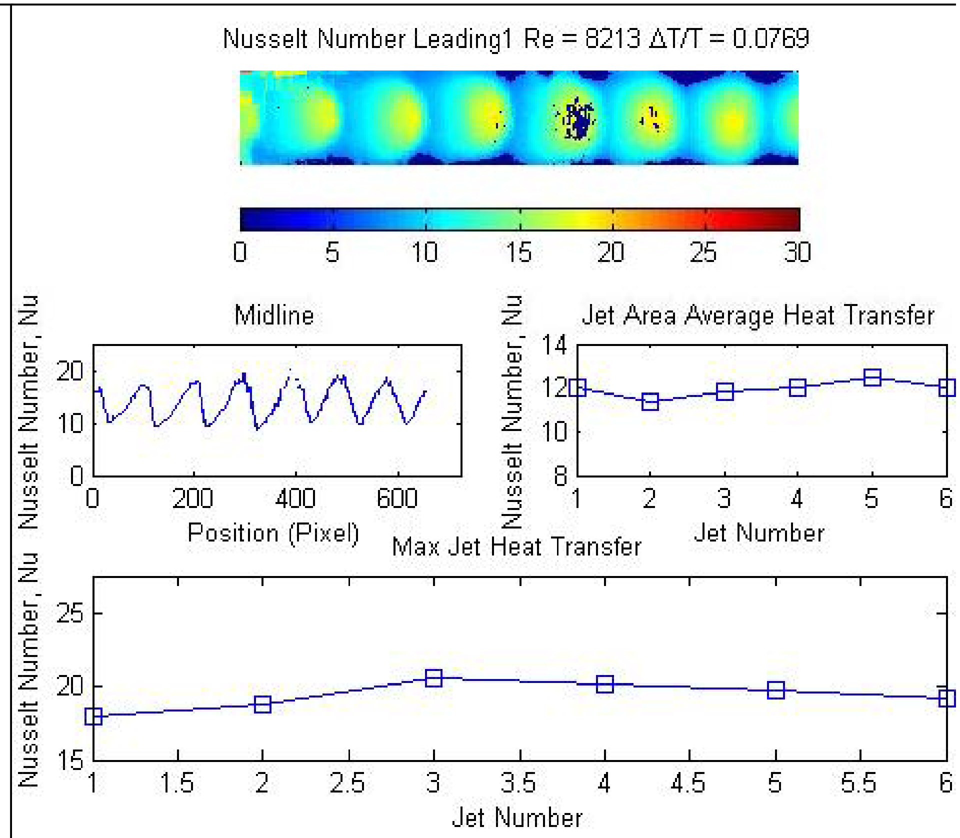
- Flow moves from right to left. There is one outlet for the air, so the later jets feel the effects of crossflow.
- Crossflow bends the jets away from the wall, reducing the effectiveness.
- The effects of crossflow are present in the results, as the average heat transfer reduces and the maximum heat transfer for each jet reduces as we get closer to the exit.

VirginiaTech

Heat, Energy and Fluids Transport (HEFT) Lab

Preliminary rotational results for trailing and leading sides with crossflow effect



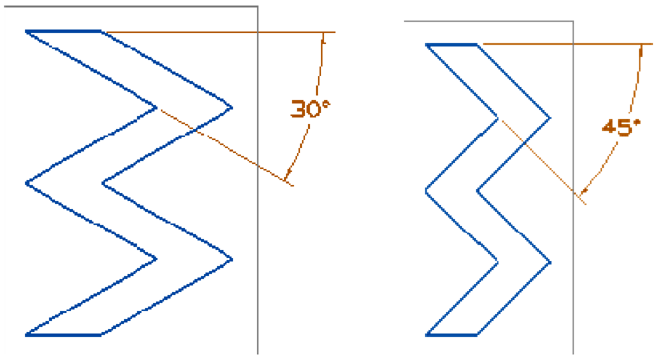


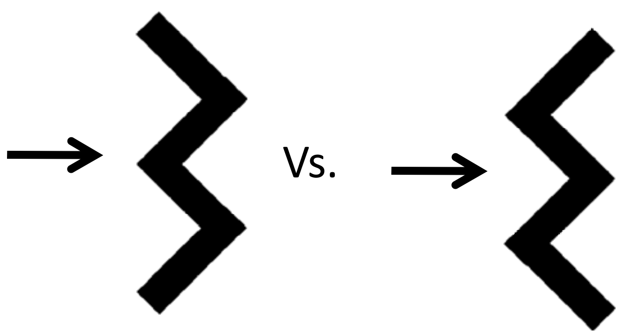
- In rotation, both the leading and trailing side results are less than the stationary results (this is expected due to previous studies by Parsons and Han (1998).
- Also, the trailing side reduced more than the leading side. This is counter intuitive because radially outward flow for two-pass channels show an increase in heat transfer for the trailing side due to the favorable effects of the Coriolis force.

Future studies on rib roughened and jet impingement cooling schemes under rotation

Rib Roughened Walls

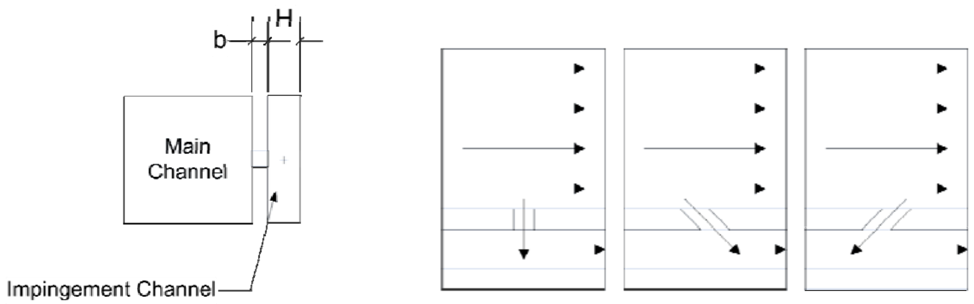
- Further explore the W-shaped Ribs with variations on flow orientation of ribs and angle of ribs





Jet Impingement

- Explore rotational effects when the impingement height is varied (H/d), when pitch is varied (P/d), effect of film coolant extraction vs. crossflow exit conditions, and Jet angle with respect to the impingement plate is varied.



- Capabilities of the Rotating Rig are in the process of being increased. Higher flow rates, lower temperatures and higher rotational speeds will be achievable.

Conclusions

- First year of project
- Understanding various parametric effects on double wall cooling schemes
- Developed a new test rig for detailed measurements in rotating frame for internal heat transfer
- Fundamental methodology of optimization achieved
- Focusing on more realistic geometries from here on
- Working with industry to determine factors of evaluation and design methodology for new cooling geometries