Internal Cooling Concepts For an Additively Manufactured Gas Turbine Combustor Liner.

Thomas Corbett

Pennsylvania State University

Mentor: Sudhakar Bollapragada

Supervisor: Erik Lopez-Partida

Department Manager: Douglas Rawlins

University Turbine Systems Research

2020 Industrial Fellowship

South West Research Institute ®

Abstract

Gas turbine combustor airflow must be carefully managed to achieve good combustor performance. Reducing the air required to cool the metal structure of the combustor provides more air to lower NOx emissions or to adjust the exit temperature profile. This work was initiated to investigate novel cooling concepts that minimize and more effectively use the cooling flow to cool the combustion liner walls. Leveraging the ability of Additive Manufacturing (AM), conceptual internal cooling configurations were developed. These proposed designs were analyzed by a series of increasingly complex analyses ending in 3D Conjugate Heat Transfer (CHT) simulations. It was found through this analysis that implementing such AM internal cooling has great potential to improve and minimize the liner cooling flow usage.

Nomenclature

A	Cross-Sectional Flow Area
D_h	Hydraulic Diameter
f	Darcy Friction Factor
k_s	Sand Grain Roughness
L	Channel Length
Nu	Nusselt Number
P	Perimeter
Pr	Prandtl Number
ΔP	Pressure Drop
Re	Reynolds Number
R_a	Arithmetic Mean Roughness
v	Velocity
ρ	Density

Background

This project investigated the potential to use AM capabilities to improve the performance of the combustor liner components of a popular Solar Turbines engine. An example of a combustor liner can be seen in Figure 1. The components that make up the combustor liner are located in the combustion section of the engine, and commonly have several functions. These components are primarily structural in nature and are commonly exposed to high temperatures or heat fluxes from the combustion. There are many different types of external cooling technologies available to these components, such as impingement, ribs, effusion, or channels. One issue that becomes apparent when producing these components however is the more sophisticated the designs become, the more difficult it will be to manufacture. Producing test parts to develop new designs can also be costly and take a significant amount of time. This is where leveraging AM can offer many new possibilities to engineers.

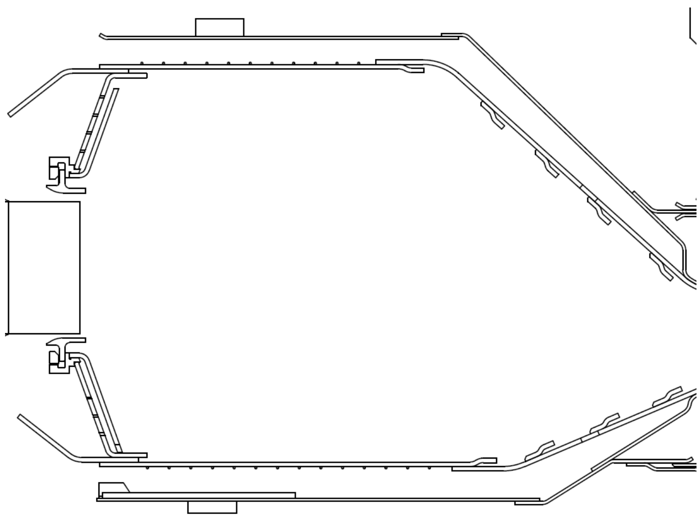

Figure 1: Combustor liner used in popular Solar Turbines engine

Additive Manufacturing (AM) is an increasingly popular means to produce components. There are many different types of AM processes, but generally speaking they all follow a similar layer-by-layer deposition approach. Specifically, for this project the AM process that was kept in mind was Powder Bed Fusion (PBF). To print components using this process, CAD files are first imported into a slicing program, where they are virtually placed inside the build volume, cut into a series of cross sections, or slices, and then sent to a printer. Once the files are ready, the printer can start the build process by first laying down a thin layer of metal powder on a large metal substrate, and then fuse powder to the substrate and to surrounding particles using a laser that will raster a single slice of a given build. Once the layer is complete, the machine will lay down a new layer of powder atop the old and repeat the process, joining the new layer to the previous. This can be repeated many times over and allows for the creation of unique components that can feature internal features that were previously very difficult or impossible to create. Additionally, AM is unique in that part complexity does not scale with cost or manufacture time nearly as much as traditional methods, allowing for greater creativity and faster design iterations.

An issue that is a result of the powder-based process however is surfaces can have high roughness values, due to partially sintered powder or variations between fused layers. This can cause problems with internal flow passages, but also creates opportunities to improve the performance of internal geometries by leveraging this roughness to improve heat transfer.

To best use the AM capabilities, this project focused on incorporating internal cooling schemes into a combustor liner component that is currently produced as individual parts. This would allow for joining of multiple components together into a single part, reducing overall part count, in addition to potentially improving part performance through new cooling schemes. These schemes can take the form of microchannels, pin fin arrays, internal and external ribs, and more. For this preliminary study, the specific geometry of interest for this project were microchannels and pin fin arrays.

Method

To accomplish better understand representative performance of an AM combustor liner component, an analysis of increasing complexity was completed. First, a 1D analysis was completed that identified ideal sizing for various components, taking the roughness characteristics into considerations. The sizing that was found was used to develop several 2D concepts that fit within a representative component. The concepts were then the subject of CHT simulations using Star CCM+, where their boundaries were modeled based on conditions inside an industrial turbine engine. Unfortunately, the 2D simulations did not capture the heat transfer in the system well, so the 2D concepts were packaged into a 3D component of a representative thickness. The updated concepts then underwent a more refined CHT analysis that captured a more representative performance.

1D Analysis

The first step in the investigation was determining a characteristic size for the internal cooling schemes. The difficulty in this step was including the roughness characteristics into the analysis, as it has a significant impact on the overall performance of the channels. This roughness was accounted for by using several correlations created by researchers at Penn State [1]. The first correlation allows for the substitution of an arithmetic mean roughness into the Colebrook equation, and the second correlation allows for a relatively accurate prediction of the Nusselt number. The two correlations can be seen below:

$$\mathrm{D_h} = \frac{4\mathrm{A}}{\mathrm{P}} \qquad \text{Eqn. 1}$$

$$\frac{\mathrm{k_s}}{\mathrm{D_h}} = 18\frac{\mathrm{R_a}}{\mathrm{D_h}} - 0.05 \qquad \text{Eqn. 2}$$

$$\frac{1}{\sqrt{\mathrm{f}}} = -2.0\log\left(\frac{\mathrm{k_s}/\mathrm{D_h}}{3.7} + \frac{2.51}{\mathrm{Re}\sqrt{\mathrm{f}}}\right) \qquad \text{Eqn. 3}$$

$$\mathrm{Nu} = \left(\frac{(\mathrm{Re}^{0.5} - 29)\mathrm{Pr}\sqrt{\mathrm{f}/8}}{0.6(1 - \mathrm{Pr}^{2/3})}\right) \qquad \text{Eqn. 4}$$

After obtaining the friction factor from Equation 3, the pressure drop could be calculated using Darcy-Wiesbach equation, seen as Equation 5.

$$\frac{\Delta P}{L} = f\frac{\rho}{2}\frac{v^2}{D_h} \qquad \text{Eqn. 5}$$

A roughness value of 15 microns was selected based on sample roughness values found in work from Penn State [2].

Figure 2: Diagram for the 1D analysis.

Using these equations and roughness, a representative size was determined by balancing the expected pressure drop versus the expected augmentation to the heat transfer.

2D Analysis

With a characteristic size defined, the next step was working towards a simplistic 2D analysis. This process began by defining a generic domain to fit the varied cooling schemes into. The domain was defined by mimicking the size of a component in the liner but was simplified to be only a 2D section that represented the outer perimeter of the part.

Five 2D geometries were created by using variations on the pin fin and microchannel cooling geometries. Their key characteristics (channel diameter, pin size) were defined using the characteristic size determined by the 1D analysis. The goal for the concepts was to create a general idea of what the cooling structure would be, while maintaining generally similar geometric characteristics between the designs. All were to serve as a starting point for a more in-depth optimization at a later date.

Due to its simplicity, setting up the boundary conditions for the 2D analysis proved difficult. After a several attempts at simulating the designs using CHT, it was found the designs were returning unreal results due largely to a lack of conduction, and not much valuable information was able to be determined from these results. This was especially obvious in configurations such as the pin fin arrays, where the lack of conduction between a substrate and a pin meant that their effectiveness was incredibly low. Per the guidance of other members of the combustion team, the analysis was then moved to 3D.

3D Analysis

When transitioning the 2D designs to 3D, they were kept largely the same. The channels were defined as circular channels, and the pin fin arrays were simply extruded one characteristic length. To create a complete plate, a wall of solid material was extruded on either side of each of the designs, that was also one characteristic length thick, resulting in a total thickness three characteristic lengths. Inlet and exit holes were placed on the bodies of the designs to enable flow to pass through each of the interior sections. The conjugate system was created by extruding a volume of air in front of the domain, and then creating an inverted version of the interior model using a Boolean operation inside PTC Creo. The air domain also extended out outlet locations for 10 characteristic lengths to ensure that the flow was uniform before meeting the outlet boundary.

The boundary conditions for the design were based on the physical environment that a combustor liner component experiences in an industrial gas turbine engine. Air entered the domain at a cooler temperature and higher pressure through a stagnation inlet, representing cooling air, and left the domain out the opposite side to a much hotter temperature, but lower pressure through a pressure outlet, representing the inside of the combustion chamber. The set pressures on either side of the plate represented the pressure ratio that the current production configuration is subject to. The combustion facing side was exposed to a convective boundary condition that was defined by calculating an effective convective coefficient from previous CFD of the component. The interface between the solid and fluid domain was mapped to allow for heat transfer, and there was a uniform roughness applied to these surfaces of 15 microns.

The designs were setup using a segregated flow and energy solver, with the spatial equations discretized to the second order. The realizable k-epsilon turbulence model was used due to its convergence reliability. The density of the air was calculated using the ideal gas law due to the high variation in density between the inlet and outlet conditions. The solid domain was defined as an additive metal material and had its conduction modeled as a function of temperature based on values experimentally found by Solar.

The base size of each of the model's mesh was 2.5mm, which was decided based on an initial sensitivity study. The interface between the solid and fluid components was meshed with inflation layers to ensure heat transfer was captured in the boundary layers. This resulted in each mesh having about 20 million polyhedral elements. The models were solved on a HPC system and took approximately two hours to solve each.

Conclusion

Based on this initial study, it appears there is significant thermo-fluid potential for developing an internally cooled combustion liner component. Several conceptual designs that were created for this project were found to achieve similar thermal performance to the current production design, while significantly reducing the amount of coolant mass flow required to do so.

If further investigation is to be done, it is recommended that the best preforming concepts are further explored, focusing on modifying some of their key geometric characteristics (channel size, spacing, pin shape, pin size, and pin spacing) to continue to improve the temperature uniformity and minimize maximum temperatures. While this is being done, packaging these designs into a more representative component should be done so that they can be simulated in a mid-frame CHT model to more accurately capture the thermal and fluid conditions that the component would be exposed to.

If results continue to be promising, a follow up analysis should be performed evaluating the thermal stress that these components are undergoing, especially due to the large thermal gradient between the inlet and outlet conditions. Additionally, a modal analysis should be completed to identify if there are any issues with the component given its location in the engine. Pending the design passes all the previous steps, follow up testing should be conducted where a representative component is printed and evaluated for its geometric accuracy and flow testing. Then the component could move forward through later testing phases, such as single injector rig testing, atmospheric rig testing, and full-scale engine testing.

References

[1] Stimpson, C. K., Snyder, J. C., Thole, K. A., and Mongillo, D., "Scaling Roughness Effects on Pressure Loss and Heat Transfer of Additively Manufactured Channels," J. Turbomach, vol 139(2), pp. 021003, 2016.

[2] Stimpson, C. K., Snyder, J. C., Thole, K. A., and Mongillo, D., "Roughness Effects on Flow and Heat Transfer for Additively Manufactured Channels," J. Turbomach. 2016;138(5):051008-051008-10. doi:10.1115/1.4032167.