ABSTRACT

Additive manufacturing (AM) techniques are expanding what is possible for designing and constructing gas turbine components that can handle increasingly harsh operating conditions. However, AM techniques can also introduce surface roughness that is more prominent than conventional manufacturing methods. The influence of this roughness could have an important effect on the performance of film cooling holes. However, little is known experimentally on what happens inside of a film cooling hole since this region is challenging to access with traditional measurement techniques. This study uses magnetic resonance velocimetry (MRV) to observe the in-hole flow structure of a scaled version of a baseline diffuser-shaped hole configuration with and without surface roughness at two blowing ratio conditions. The roughness geometry is derived from computed tomography (CT) scans of a metal-AM diffuser hole. The three-component, three-dimensional, time-averaged velocity field is measured by MRV and includes the flow from the plenum, within the hole, and in the vicinity of the hole exit. The momentum distribution within the diffuser differs between the smooth and rough holes, with peak velocities and flow asymmetry influenced by the surface roughness. Flow along the leeward wall of the diffuser is nearly separated, and the size of this separated flow region is smaller in the roughened cases. These effects are accentuated at larger blowing ratio. These data provide explanatory evidence of how the momentum distribution within smooth and rough holes may impact surface effectiveness results downstream of the shaped holes. CT scans of the surface roughness coupled with the MRV measurements provide non-optical means to characterize the hole geometry and as-built performance of additively manufactured test coupons with important implications for the field.
NOMENCLATURE

- $A_c$: cross-sectional area
- $AM$: additive manufacturing
- $A_{min}$: minimum cross-sectional area of cooling hole metering section
- $BR$: blowing ratio
- $CT$: computed tomography
- $D$: metering hole diameter
- $D_M$: momentum distortion parameter
- $H$: main channel height
- $L_d$: diffuser section length
- $L_m$: metering section length
- $Ma$: Mach number
- $MRC$: magnetic resonance concentration
- $MRI$: magnetic resonance imaging
- $MRV$: magnetic resonance velocimetry
- $N$: number of elements
- $\hat{n}$: unit normal vector
- $PIV$: particle image velocimetry
- $R_a$: arithmetic mean roughness
- $Re$: Reynolds number
- $U$, $V$, $W$: streamwise, wall-normal, and spanwise velocity components
- $U'$, $V'$, $W'$: velocity components in the reference frame aligned with the metering hole axis
- $U_a$: cross-sectionally averaged velocity
- $U_b$: main channel bulk velocity
- $x$, $y$, $z$: streamwise, wall-normal, and spanwise direction
- $x'$, $y'$, $z'$: coordinate system rotated about the $z$-direction to align with the metering hole axis
- $\delta_{99}$: boundary layer thickness
- $\theta$: momentum thickness
- $\nu$: kinematic viscosity

INTRODUCTION

Additive manufacturing (AM) has become an increasingly important solution to develop next-generation components for gas turbine engines. Unlike conventional manufacturing techniques, AM enables the ability to create complex geometries that were previously unattainable. In recent years, advanced AM has enabled the creation of complex components with high-performance materials (e.g., high-temperature nickel alloys, titanium, and ceramic matrix composites (CMCs)) that are relevant for enduring the harsh environment of a high temperature gas turbine.

Despite the apparent benefits, there are several issues associated with AM that must be explored. One of those issues is to assess the influence of imperfections in AM components. The difference in the intended design (i.e., as-designed) versus the resulting imperfect geometry of the AM process (i.e., as-built) is a critical topic for study. Imperfections, such as surface roughness, and the intentional weave pattern of CMCs, have been shown to have important implications for aerodynamic performance [7–10]. AM-induced flow effects are particularly relevant when coupled with film cooling.

Film cooling is a technique to route relatively cold air to the inside of turbine components that are subject to some of the highest heating loads inside the engine. The air is first used to cool the inside of the blade or vane before being bled out onto the surface. Emitting the air onto the surface of the component produces a coolant film that acts to thermally insulate it from the harsh environment of the hot gas downstream of the combustor. Small changes in the geometry of film cooling holes can have large changes in film cooling behavior, thereby impacting the performance of a gas turbine engine. Significant effort has been put forth to improve film cooling geometries [11].

AM methods present the opportunity to further improve film cooling geometries by allowing for increasingly optimized designs which conventional subtractive techniques could not achieve. However, the subsequent roughness and surface patterns of AM components might alter how engineers must design next-generation components. The emerging nature of this topic means that gaps remain in the open literature for fully describing how AM techniques might influence film cooling performance. The majority of relevant studies that are in the open literature assess film cooling performance based on the use of IR thermography for optically accessible surfaces [7–10]. Schroeder and Thole [12] examined the influence of in-hole roughness by measuring adiabatic effectiveness downstream of shaped holes with varying degrees of roughness inside the metering hole. The results from Schroeder and Thole provide foundational information to compare to for the purpose of this study. What is not fully resolved in the literature is the precise behavior of the coolant flow in regions that are not optically accessible. For example, what are the mechanisms for how in-hole surface roughness alters flow behavior within the hole and how do the in-hole flow characteristics impact external cooling performance downstream of the hole?

Magnetic resonance imaging (MRI) techniques have been used for several years in gas turbine applications to measure three-dimensional (3D) velocity and concentration fields [13–19]. Specifically, magnetic resonance velocimetry (MRV) [20] and magnetic resonance concentration (MRC) [21, 22] provide the time-averaged three-component (3C) velocity and concentration fields at millions of points within an imaging region, respectively. The techniques do not require optical access and are therefore well-suited to characterize complex internal flows with strong secondary motions and flow separation. At present, most MR-based techniques use aqueous solutions as the working fluid.
to improve signal-to-noise ratio due to their high proton density. Therefore, the flows considered are in the incompressible limit and at moderate density ratios. Nevertheless, comparisons to studies in air at subsonic Mach numbers (Ma < 0.7, c.f. [23]) showed good agreement provided that the flows were fully turbulent and appropriate corrections for thermal effects were applied. Therefore, MRI techniques are especially relevant for understanding complex flow patterns within cooling holes.

This study employs MRV to collect high resolution time-averaged velocity data of flow inside of a film cooling hole with and without surface roughness and at two blowing ratios. Baseline smooth and rough holes are manufactured based on the 777 geometry [24], with internal roughness derived from the computed tomography (CT) scan of a metal-AM hole. The analysis focuses on modification to the mean momentum distribution within the hole as a result of the roughness. This work is motivated by the previous study of Schroeder and Thole [12] that investigated the influence of in-hole roughness on the external flow field using particle image velocimetry (PIV). Despite the fact that Schroeder and Thole observed the influence of in-hole roughness by investigating turbulent mixing downstream of the holes, that study did not make direct measurement of the fluid flow inside the hole due to the lack of optical accessibility. As such, the current study is meant as an extension of Schroeder and Thole [12] to help fill the knowledge gap regarding the influence of in-hole roughness with AM components. This work is also a first of its kind to present experimental data for in-hole flow behavior with and without roughness on an AM-relevant component. It demonstrates the ability to conduct rapid and systematic studies to understand the impact of as-built features on fluid flow in complex geometries.

**EXPERIMENTAL METHODS**

**Overview and Flow Conditions**

Experiments are performed on scale models of smooth and rough film cooling holes based on the 777 diffuser geometry of Schroeder and Thole [24] and motivated by the rough hole study of Schroeder and Thole [12]. This section describes the baseline smooth hole, as shown in Figure 1, and the rough hole is characterized in the following section. The metering hole diameter is $D = 5.8\, \text{mm}$ and the hole axis is inclined by $30^\circ$ relative to the downstream direction. The metering hole length is $L_m = 2.5D$, after which it expands in each of the forward and lateral directions by $7^\circ$ to produce the laidback fan-shaped diffuser. The diffuser section has a length of $L_d = 3.5D$ measured from the end of the metering section to the breakout surface. The origin of the coordinate system is located where the metering hole axis intersects the breakout surface, with $x$, $y$, and $z$ denoting the streamwise, wall-normal, and spanwise coordinates, respectively.

The smooth and rough holes are integrated into the water channel shown in Figure 2. All parts were 3D printed by the W.M. Keck Center located at the University of Texas El Paso using stereolithography. A flow conditioning section consisting of a gridded diffuser, a settling chamber with a honeycomb and grid, and a 4:1 area ratio contraction produces a nearly uniform velocity profile at the inlet to the test section. The test section has a $50\, \text{mm}$ by $50\, \text{mm}$ square cross-section. A $1\, \text{mm}$ boundary layer trip along all four walls initiates a new turbulent boundary layer which develops over $210\, \text{mm}$ ($36D$) before reaching the location of the cooling holes. The cooling holes are placed on opposite walls to minimize hydrodynamic interactions, and each are fed by a plenum. A $307\, \text{mm}$ straight outlet section is placed downstream of the holes. The entire apparatus is part of a closed flow loop. A reservoir supplies the main flow and film cooling holes using centrifugal pumps and flexible, reinforced plastic tubing. The main flow is metered using a diaphragm valve and ultrasonic flow meter (Transonic Systems 20PXL flow probe). The cooling holes are operated simultaneously and independently metered using needle valves and ultrasonic flow meters (Transonic Systems 6PXL flow probes). The outlet section is connected back to the reservoir by tubing.

Water with a 0.06 molar solution of copper sulfate is used as the working fluid. This dilute solution of copper sulfate does not appreciably change the properties of water [22]. The temperature of the fluid was maintained at approximately $20^\circ\text{C}$ using a chilled water heat exchanger placed in the main reservoir. The bulk velocity of the main flow was maintained at $U_b = 0.5\, \text{m/s}$ throughout the experiments, giving a channel Reynolds number of $Re_H = 25,000$ based on the test section height, $H = 50\, \text{mm}$, and the kinematic viscosity of water. Previous experiments using the same flow conditioning sections and development length characterized the boundary layer profile at the injection location in the absence of the hole using hot-wire anemometry [17]. Specifi-
cally, the boundary layer thickness is \( \delta_{99} = 1.5D \), the momentum thickness is \( \theta = 0.15D \), and the momentum thickness Reynolds number is \( \text{Re}_\theta = 435 \). The freestream turbulence intensity is approximately 1%.

The holes are operated at blowing ratios \( BR = 1.0 \) (BR1) and \( BR = 1.5 \) (BR1.5), where blowing ratio is defined as the ratio for the bulk velocity in the metering section to the bulk velocity of the main flow since the density ratio is equal to one. This gives hole Reynolds numbers defined using the metering hole diameter of \( \text{Re}_D = 2,900 \) and 4,350, respectively. Milani et al. [25] showed that the metering hole flow is fully turbulent due to a separated shear layer at the inlet for these Reynolds numbers, and Gunday et al. [18] demonstrated that adiabatic effectiveness results for the BR1 case agree well with the thermal measurements of Schroeder and Thole [24] taken at moderately higher Reynolds number and a density ratio of 1.2. For each blowing ratio, the rough and smooth holes on opposite walls are operated simultaneously. Flow visualization using dye and an abbreviated set of MRV measurements showed that the jets emanating from each hole remain close to the downstream surfaces and do not interact. However, the additional flow rate added to the main channel by the holes is expected to create a slight acceleration of the freestream. The additional flow rate downstream of injection, accounting for both holes, is 2.1% at BR1 and 3.2% at BR1.5. Note that although the free-stream acceleration increases slightly between blowing ratios, both the smooth and rough holes at a fixed blowing ratio experience identical conditions.

**Rough Hole Geometry and Characterization**

To understand how in-hole roughness affects a film-cooling flow field, two 777 holes developed by Schroeder and Thole [24] were printed using stereolithography (SLA). The original engine-scale holes had a nominal metering diameter of \( D = 0.763 \text{ mm} \). This is termed the 1X scale or small-scale hole. The SLA designs were increased in size so that roughness could be controlled. Specifically, the nominal metering hole diameter of the smooth and rough holes was increased to \( D = 5.8 \text{ mm} \). These are termed the 7X scale or large-scale holes.

The geometry of the 7X smooth hole was previously described. The 7X rough hole was designed by scaling the surface of an AM engine-scale (1X) 777 hole as determined by an industrial CT scan. The 1X rough hole was manufactured using laser powder bed fusion (L-PBF) on an EOS M280 in Inconel 718 with a 40 \( \mu \text{m} \) layer thickness [8]. To minimize the deviation from the design intent (i.e., 1X smooth hole) during the L-PBF build, the hole was oriented such that the metering axis was perpendicular to the substrate. The objective of this study is to isolate the effect of in-hole roughness and mitigate differences due to the inlet geometry, as well as external roughness which could affect the developing boundary layer. Therefore, the inlet of the 7X rough hole was replaced with a smooth inlet and the test section surface external to the hole was made smooth. Roughness towards the end of the metering section and in the diffuser was scaled from the metal-AM surface data. Finally, the L-PBF hole prints slightly oversized as will be described below. To remove this effect and again isolate the impact of realistic in-hole roughness, the large-scale rough hole was scaled so that the mean surface height matched the idealized 777 hole as closely as possible.

CT scans were used to compare the as-built 7X SLA holes (smooth and rough) and the 1X metal-AM hole to their design intents. Note that there are different design intents for each hole. The 7X smooth hole and the 1X metal-AM hole were designed to match the idealized 777 geometry. The 7X rough hole was designed to match the metal-AM hole roughness but with a smooth metering hole inlet. Deviations from design intent for the 1X hole are due to the L-PBF process and understanding their impact on the flow field is the objective of this work. Deviations from design intent for the 7X holes are due to the SLA process and must be quantified. Both 7X holes (rough and smooth) were CT scanned with a voxel size of 50 \( \mu \text{m} \) while the as-built 1X hole was scanned with a voxel size of 35 \( \mu \text{m} \). The software used to post-process the CT scans determines the surface to within 1/10\(^{th}\) the voxel size [26]. From the CT scan surface the as-built metering diameter (\( A_{\text{min}} \)), the arithmetic mean roughness (R\(_a\)), and the deviation from the design intent could be determined.
The deviation of the small-scale hole from the design intent is shown in Figure 3 while the deviation between the small- and large-scale holes is shown in Figure 4. Positive deviation values indicate that the surface was over-built and negative values indicate it was under-built. Overall, for the small-scale hole, the hole printed with 90% of the surface being within 0.07D (0.05 mm)
of the design intent. The deviation contours of Figure 3 indicate that the variation in the surface caused the hole to be slightly oversized; specifically, the A_{min} was 10% larger than the design intent. The inside of the hole has the inherent roughness expected from the L-PBF process. The inherent roughness features come from partially sintered powder or balling. The relative arithmetic mean roughness from a region on the leeward side of the diffuser was calculated to be R_a/D = 0.026 (R_a = 20.1 \mu m).

Both large-scale holes had metering cross-sectional areas that were slightly smaller than their design intent. The large-scale smooth and rough holes had minimum cross-sectional areas 1.5% lower and 3.5% lower than the design intent, respectively. Figure 4 shows the deviation between the L-PBF rough hole and the large-scale rough hole. Similar to the 1X hole, the rough hole printed using SLA had 90% of the surface being within 0.04D (0.22 mm) of the intended design.

The large-scale smooth hole printed using SLA was hydrodynamically smooth. Within the diffuser of the large-scale rough hole shown in Figure 5, two types of roughness features were present. The first roughness feature was intentional and was based on the roughness features of the 1X hole. These intentional scaled roughness features were only slightly smaller than intended as indicated by the blue contours. The second type of roughness features are the result of residual support material and are the regions within the hole that are over-built (pink contours).

The difference in the scaled surface height between the small- and large-scale holes can also be seen in Figure 5. Most of the large-scale surface was within the expected range of the small-scale surface. The deviations that do occur are related to the two roughness features previously discussed. In the large-scale hole, the large roughness features that were scaled from the
In this section we compare the time-averaged flow structure within the cooling holes. Of particular interest are the similarities and differences between the smooth and rough holes at a fixed blowing ratio. Varying the jet velocity is used to assess consistency of the trends across moderate changes to the blowing ratio.

Figure 7 plots contours of normalized velocity magnitude on the $z/D = 0$ centerplane of the hole for each case. In all cases, flow separation at the inlet to the metering hole produces a jet.
along the windward side of the hole. This jet decelerates and spreads within the diffuser section of the hole and then quickly turns to align with the main flow at the hole exit plane. Comparing the smooth and rough cases, it is observed that the region of nearly stalled flow along the diffusing wall is larger for the smooth case than for the rough case. This effect is amplified at the higher blowing ratio.

Cross-sectional planes of the holes show similar trends. Figure 8 displays a series of locally streamwise-normal planes within the holes using a coordinate system aligned with the metering hole axis to provide a more complete picture of the 3D flow structure. The contours are of the normalized through-plane velocity component. At a fixed blowing ratio, the momentum distribution within the metering sections of the smooth and rough

Figure 6: Overview of the flow field for the BR1 case. (left) Cross-sectional and streamwise planes showing normalized velocity magnitude within the main channel and the smooth and rough cooling holes. (right) Boundary layer profile upstream of injection.

Figure 7: Normalized velocity magnitude on the spanwise centerplane (z/D = 0) for the smooth and rough holes at BR1 (left) and BR1.5 (right).
holes is similar. Recall that the rough hole metering section was designed to be nearly smooth. Only slight differences at the inlet plane are observed at the highest blowing ratio. The inlet flows at the lower blowing ratio are equivalent to within experimental uncertainty considering statistical convergence, spatial resolution, and image segmentation. The size of the roughness elements increases beyond \( x'/D = -4 \) (towards the end of the metering section) and within the diffuser section. In this region, the smooth hole displays a larger region of low momentum fluid on each cross-sectional plane. Conversely, while the rough hole also produces a skewed region of high momentum fluid on the positive \( z'/D \) side of the hole, the overall momentum distribution is more uniform, and it increases in uniformity as the flow approaches the hole exit.

The 3D-3C data allow the redistribution of mean flow momentum due to in-hole roughness to be quantified using integral metrics. Specifically, the momentum distortion integral parameter, \( D_M \), quantifies the deviation of the velocity distribution from a uniform plug flow. It has been applied to MRV data in the analysis of strut wakes [29] and curved passages [28] in gas-turbine related application areas. It is defined as

\[
D_M = \frac{1}{U_a^2 A_c} \int_{A_c} (\mathbf{U} \cdot \mathbf{n})^2 dA - 1, \tag{1}
\]

where \( A_c \) is the cross-sectional area of a plane of interest, \( U_a \) is the average velocity perpendicular to the plane, \( \mathbf{U} \) is the velocity vector, and \( \mathbf{n} \) is the unit normal to the plane. Therefore, the parameter measures the momentum flux relative to that of a uniform flow. Representative values of \( D_M \) include zero for a uniform flow, 0.022 for a turbulent pipe flow approximated using a 1/7th power law velocity profile, and 0.33 for fully-developed laminar pipe flow.

The momentum distortion parameter is computed for a wall-parallel plane near the hole outlet as shown in Figure 9. This plane was selected to have a large cross-section area in order to reduce uncertainty in the integral due to voxel resolution and image segmentation. At each blowing ratio, the momentum distortion is largest for the smooth hole and lowest for the rough hole, in agreement with the observations above. Contours of the normalized \( y \)-velocity component on this plane are also shown in Figure 9. All cases have a momentum distribution skewed towards the positive \( z/D \) side of the hole and most of the flow exits from the upstream half of the cross-section due to the separated flow pattern. The former is due to the direction of the plenum feed, which is from negative \( z/D \) to positive \( z/D \). While the momentum deficit on the downstream side of the hole is still large for the rough hole \( (D_M > 0.25) \), an increased percentage of the flow exits from positions \( x/D > 0 \).

The impact of the momentum distribution above and downstream of the hole is highlighted by Figures 10 and 11, which plot

---

**Figure 8:** Cross-sectional planes within each hole in a coordinate system aligned with the metering hole axis. \( U' \) is the normalized through-plane velocity component (i.e. \( x' \)-velocity).
contours of the normalized streamwise velocity on streamwise-normal planes located at $x/D = 0$ and $x/D = 2$ (i.e. the downstream edge of the hole), respectively. The regions of large velocity deficit within the hole are again visible in Figure 10. The BR1.5 case produces a local velocity maxima that perturbs the external boundary layer above the hole (c.f. Figure 7 as well), and is most pronounced for the smooth hole due to the larger area occupied by the slow flow beneath it.

The in-plane velocity vectors in Figure 11 show a mild vertical flow that is significantly smaller than the bulk velocity of the main flow, as expected for the diffuser hole geometry. While the vertical flow velocity increases with increasing blowing ratio, the streamwise momentum deficit decreases because the local hole velocity at the diffuser outlet is more closely matched to that of the external flow. Most significantly, the impact of the rough hole geometry reduces quickly with downstream distance, as evidenced by the similarity between the smooth and rough hole velocity distributions at this plane.

**Discussion**

In general, the MRV measurements show that in-hole roughness results in reduced flow separation (or near separation) and therefore a more uniform momentum distribution within the 777 diffuser hole. We hypothesize that this is due to increased turbulent mixing due to the rough wall condition. This would be consistent with the fact that the effect increases when the blowing ratio is raised from BR = 1 to 1.5. Schroeder and Thole [12] observed higher turbulence levels at the outlet of rough holes and concluded that increasing the blowing ratio raises the roughness-based Reynolds number, which leads to increased velocity fluctuations produced by the roughness elements.

It is interesting to note, however, that Schroeder and Thole observed increased core flow velocities due to thickened in-hole boundary layers when the surface was rough. This is in con-
Figure 11: Contours of normalized streamwise velocity with in-plane velocity vectors on the x/D = 2 plane. The reference vector length corresponds to a normalized velocity magnitude of 0.5.

Contrast to the present study, where roughness promoted a more uniform momentum distribution. There are several possibilities for this difference. First, although the mean roughness heights were similar in both studies, the AM-derived roughness in this work is preferentially distributed on the leeward side of the hole. The windward boundary layer within the AM-derived hole remains thin and we hypothesize that roughness on the diffuser wall promotes mixing of high momentum fluid into the low momentum region. Second, additional details of the roughness height probability distribution function, to include two-point statistics, are expected to differ between the two studies and are known to affect flow behavior in canonical boundary layers [30]. In light of these differences and without the availability of combined in-hole and turbulence measurements from each study, it is difficult to draw further comparisons at present. Future research should investigate turbulence statistics and film cooling performance external to the rough holes studied here in order to more closely benchmark the results against this prior work.

We also emphasize that the impact of roughness on the time-averaged flow field decreases rapidly with increasing downstream distance. Measurements of adiabatic effectiveness, or 3D coolant distribution measurements using MRC [18, 22], would help explain the integrated effect of near-field differences on the far-field cooling performance. This observation also suggests the importance of roughness enhanced turbulence intensity on cooling hole performance.

Perhaps most significantly, this study demonstrates the opportunity for rapid and systematic investigation of the impact of manufacturing defects on the 3D-3C in-hole flow field. Once the CT scan geometry was integrated into the test section design, the data were acquired over the course of two 6-hours scans using clinical grade MRI systems available in most university radiology departments, including set-up, data collection, and clean-up. Paired with MRI-based 3D concentration measurement techniques, and more conventional techniques for external flow field measurements (e.g., PIV, PLIF, infrared thermography), we expect this process to offer substantial insight into as-built geometries. Future work could include investigations of cooling performance for variations in parameterized roughness statistics, manufacturing defects at hole inlets, ceramic matrix composite surface topology, and the relative importance of internal versus external roughness.
CONCLUSIONS

Magnetic resonance velocimetry measurements were performed to understand the impact of in-hole surface roughness on the three-dimensional, time-averaged flow structure within film cooling holes. The 777 geometry of Schroder and Thole [24] was used for the baseline smooth hole, and CT scans of metal additively manufactured 777 holes were used to develop an MRI-compatible rough hole model. Roughness features external to the hole and at the metering hole inlet were removed so that the impact of roughness in the diffuser section could be isolated. Roughness features within the hole were reliably reproduced in the scale models via 3D printing with stereolithography. Measurements were made at fully-turbulent Reynolds numbers, unity density ratio, and blowing ratios of BR = 1 and 1.5 for both the smooth and rough holes.

The data show that in-hole roughness produces a more uniform momentum distribution within the cooling hole. Specifically, the high momentum deficit along the downstream wall of the diffuser is reduced. The effects are more pronounced at the higher blowing ratio case. Despite notable in-hole differences, the impact of roughness reduces rapidly with downstream distance outside of the hole.

This work demonstrates a synergistic, rapid, and systematic approach to studying the impact of realistic geometric defects on the 3D flow within film cooling holes that is not accessible by optical and probe-based measurement techniques. It is of interest in future work to apply the methods to higher blowing ratio conditions, systematic roughness variations, and as-built variability of metering hole inlet geometries.

ACKNOWLEDGMENT

A.J.B., M.J.B., and F.T.D. were supported by the Air Force Research Laboratory. W.Z., A.B., and C.B. were supported by the United States Army under contract number W911NF-22-2-0100. A.B. and C.B. also acknowledge support from the National Science Foundation under award number 2024346. E.M.V. and K.A.T. were supported by the Department of Energy under award number DE-FE0025011.

Special thanks is given to the Zuckerman Institute at Columbia University for use of the MRI facilities and to the William M. Keck Center for 3D Innovation at the University of Texas at El Paso for manufacturing the SLA model. The authors are also grateful to Sara Mueller at the Pennsylvania State University for help with the CT scans.

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions expressed herein are those of the authors and do not purport to state or reflect the position of the United States Government or any agency thereof, including the United States Military Academy, the Department of the Army, or the Department of Defense.

REFERENCES


