Finite Element Simulation of the Fused Deposition Modelling Process

Blake Courter
(Stratasys Corp, Cambridge MA, USA);

Vishal Savane, Jing Bi
(Dassault Systemes Simulia Corp, Johnston RI, USA);

Siddharth Dev, Christopher J. Hansen
(Department of Mechanical Engineering, University of Massachusetts at Lowell, Lowell MA, USA).

Abstract

Fused Deposition Modeling (FDM) enables the development of functional prototypes, tooling, molds and fixtures of complex geometries which are difficult to manufacture using conventional manufacturing technologies such as injection molding. In the FDM process, thermoplastic materials such as ABS and ULTEM are extruded as filaments and deposited layer-by-layer to build a part directly from a CAD model. FDM parts experience a constant time-varying thermal profile during the manufacturing process. This results from the molten filament transitioning from a high temperature as one layer is being deposited, to the rapid cooling it experiences during the deposition onto the supporting layer. As a result, parts manufactured using FDM can often deviate from the as-designed geometry, and their mechanical performance can be affected due to the distortions and residual stresses that accumulate during the printing process. These geometric defects and stress states can significantly impact the product development time as well as the performance and in-service lifespan of FDM printed parts. Finite element simulation can be utilized as a predictive tool to understand the effect of these issues as well as mitigate the negative impact they have on the as-printed FDM part.

In this paper, the commercial finite element software package Abaqus is used to simulate the FDM process. A Mobius arm part is used to illustrate the simulation procedure and a sequentially coupled thermo-mechanical analysis is performed. The heat transfer analysis calculates the temperature history which is mapped onto and used to predict residual stress and potential part distortion in the subsequent structural analysis. The tool path patterns dictate how material is progressively added to the part during the build and directly influence the accumulation of residual stresses within the part. Independent tool path events are characterized using event series data such as time, location and bead cross-sectional area. Abaqus solves for dependent events such as progressive element activation, local material orientation, evolution of cooling surfaces, temperature profile, stress and distortion. The results show that finite

element simulations effectively capture the interaction between tool path and thermomechanical physical processes and their impact on the final state of the printed part.

1. Introduction

Fused Deposition Modeling (FDM) is a popular layer additive manufacturing process used in many industries including aerospace, automotive, industrial, commercial and medical. Prototypes and end-use parts are produced by FDM using production grade thermoplastic materials such as ABS, polycarbonate and ULTEM. In an FDM process, 3D CAD files of part designs are pre-processed by slicing the part into layers generating necessary support structures and extrusion paths. The data is then transferred to a machine that constructs the part on a build platform. Support and part materials in filament form are heated in dual extrusion nozzles. The extrusion nozzles then lay down melted materials layer by layer from the bottom up. On each layer, the extrusion tip follows the precise tool path created by the pre-processing software. Solvable support materials are then dissolved in water based solutions. During the building process, each layer of material cools down by conduction, convection and radiation before the next layer of melted material is deposited on top. This temperature gradient is responsible for accumulated residual stresses that cause warpage, distortion and delamination, which in turn impact the product development cycle as well as the in-service performance and durability of the parts.

Mahesh et al. [1] used a benchmark part to evaluate the performance of many layer additive manufacturing processes. Tolerance values such as flatness and symmetry were investigated. However, physically print and evaluate the tolerance is too expensive to be used effectively for the development of every single part. Alternatively, finite element simulation can be utilized as a predictive tool to understand the effect of these issues as well as mitigate the negative impact they have on the as-printed FDM part, at a much lower cost. Many researches were performed to study the effect of process parameters such as deposition path and speed on the product analytics such as temperature, stress distribution and tolerance [2, 3, and 4]. However, most analyses in literatures are difficult to apply to different process types and not suitable for complex geometries due to simplifying assumptions. It is essential to capture the correct physics such as conduction and convection and the evolving change of heat transfer surfaces as the material is being deposited using high fidelity models while having the freedom to achieve faster results with low fidelity models.

In this paper, a new framework in the commercial software package Abaqus [5] was developed to model different additive manufacturing processes including the FDM process. A Mobius arm example with complex geometry is used to illustrate how the new framework can be used effectively for various levels of fidelities with and without a CAD file. Detailed tool path data are translated by python scripts automatically to the Abaqus input format, which determines the dependent events such as progressive element activation, local material orientation, evolution of heat transfer surfaces, temperature profile, stress and distortion that Abaqus solves for.

2. Methodology

A new framework was developed in Abaqus aiming at providing accurate and scalable predictions for Additive Manufacturing processes at part level. The framework is an open user customizable subroutine interface including a pair of element activation subroutines, a pair of moving flux subroutines and many built-in utility routines. Many existing Additive Manufacturing processes can be modelled using the framework such as FDM, SLM, DLM, Polyjet and etc. New type of process technologies can also be customized with the framework.

The Additive Manufacturing framework in Abaqus contains technologies in the following categories to address many simulation challenges: supporting finite element mesh with varying mesh density as AM parts usually have complex shapes and uniform mesh is not practical; supporting input of process parameters and specifics for different AM processes such as deposition head dimension and moving speed, stack direction and etc; supporting intersection of finite element mesh with the tool path in a geometric sense and progressive element activation based on intersected elements computed; supporting progressive heating computation taking account actual tool path data; supporting progressive cooling computation taking account of the evolving heat transfer surfaces during the printing process.

In this paper, the Additive Manufacturing framework in Abaqus is applied to the FDM process simulation. Detailed methodology for FDM process such as machine tool path generation and conversion, process parameters and process specifics input, material characterization, meshing options will be discussed in the rest of this section.

Detailed tool path data was generated by Stratasys including time, spatial position of the deposition head, deposition bead area, material types and material deposition description such as infill, base, contour and etc. The tool path can be designed and customized in Stratasys software to incorporate different infill densities, different contouring and etc. *Figure 1* shows the tool path generated for an aerospace rocker model including base (bottom green), support (yellow) and the actual part (upper green). The tool path of one middle layer is highlighted with light green color.

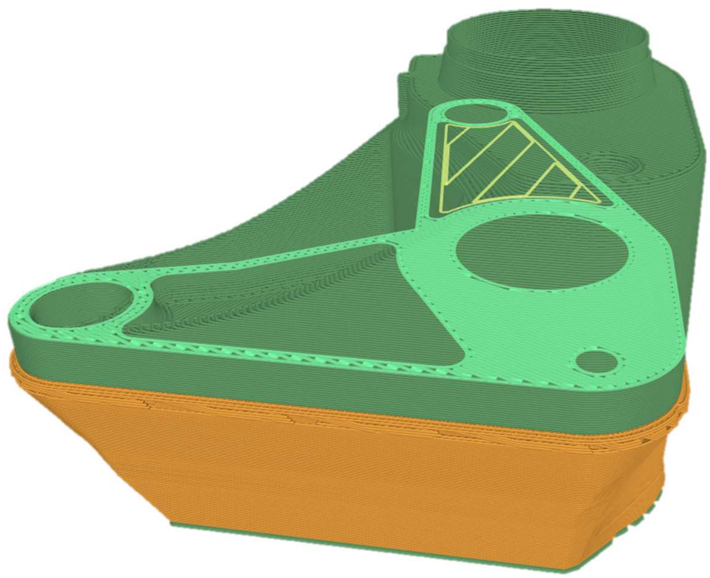


Figure 1: *Tool path generation for an aerospace rocker model with support and base*

The tool path data can be extracted in different formats including readable text file format. A general python script was written to process the tool path data and convert it into the appropriate format as Event Series data for Abaqus analyses. Multiple Event Series data are supported by the Abaqus Additive Manufacturing framework for possible scenarios including multiple print head, multiple laser heat sources, multiple heating lamps or a powder recoating process without any heat source. In an FDM process, polymer materials are deposited at the melting temperature, therefore, there is only one single set of Event Series data file. This file can includes all necessary information from the tool path description including time, spatial location of the print head center and bead flow information such as varying or constant bead cross section area or varying or constant mass rate. Utility routines were developed in Abaqus to efficiently extract corresponding information at appropriate times for relative calculation. Tool path data are consisted of Event Series line segments as shown in *Figure 2*. Deposition bead width and height are used to create moving rectangular boxes with line segments as the center of the boxes. Two levels of filtering are performed to first find adjacent segments out of the whole Event Series data by a search radius defined at the scale of the diagonal length of the bead cross section, and then determine whether the element or material is within the rectangular box. The elements and materials properties are activated at appropriate time according to the Event Series data. *Figure 3* shows a simple square example how materials and elements are activated at molten temperature according to the tool path including contour and infill.

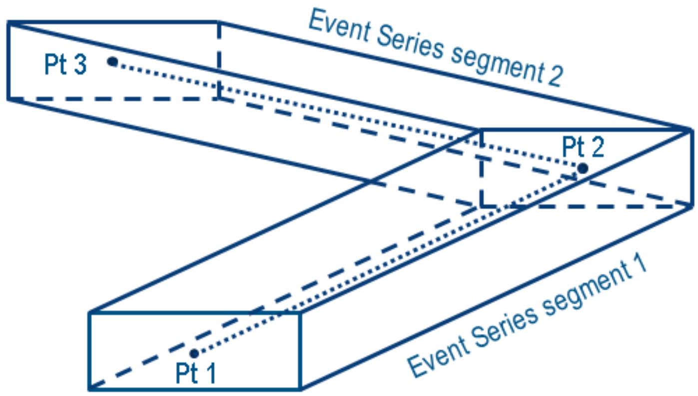


Figure 2: Event Series segments and moving boxes for element/material activation

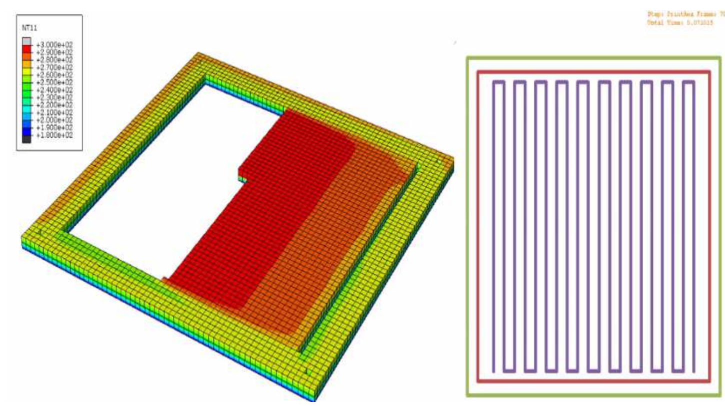


Figure 3: Element activation driven by tool path data

Material characterization is performed experimentally in order to obtain as-built material properties including the manufacturing defects, namely the bead aspect ratio, void ratio, void shape and bonding between beads. Material is treated as orthotropic with the deposition flow direction as the local 1-direction, in-plane orthogonal direction as 2-direction and the out-of-plane direction as 3-direction. The local material orientations are assigned automatically according to the tool path through a user orientation subroutine available in Abaqus - ORIENT. At the beginning of the Abaqus/Standard analysis, relative tool path segments will be filtered out and their orientation is used to determine the local material orientation of each element. This is a one-time calculation in the beginning of the analysis and does not affect the solver performance. *Figure 4* shows a simple square example how materials orientations are assigned according to the tool path including contour and infill.

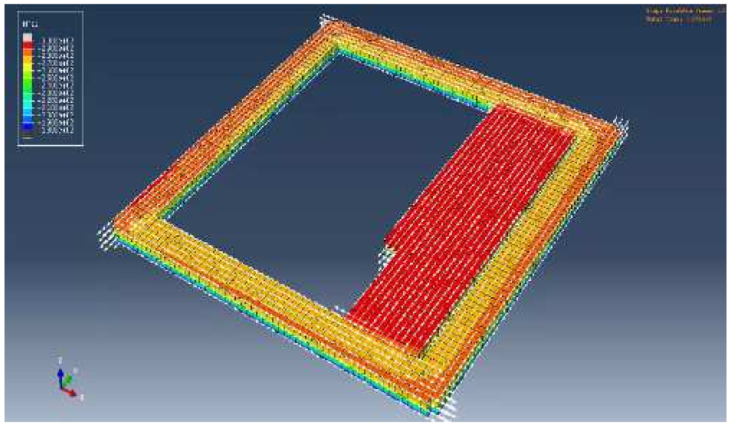


Figure 4: Material orientation assignment according to tool path data

A customizable Abaqus input deck framework called "Table Collection" was developed to support customized user designed data inputs including event series types such as material power roll in event, laser heating event and melting material deposition event, process parameters and specifics such as stack direction, bead width or bead height, element activation algorithms such as two level segment filtering algorithm, tool path description types such as variable bead cross section area or variable mass flow rate.

Due to the difficulty to obtain CAD geometries of the base and support structure, both CAD geometry based meshing and voxel mesh without CAD geometry are supported by the framework. In the next section, we will describe the application of the Additive Manufacturing process simulation framework for a realistic example of a Mobius arm part that has a complex geometry. Details of the simulation model including both mesh options will be discussed.

3. Finite element simulation

The intention is to monitor the thermal mechanical process during printing. Sequentially coupled thermal mechanical analysis is used with a heat transfer analysis first providing temperature history of the process, which is then mapped to the subsequent static analysis and drives the stress and distortion calculations. Experiments were carried out for printed ABSplus P430 material coupons for orthotropic mechanical and thermal properties of the as-built material. Table 1 lists the basic orthotropic mechanical and thermal properties that come out of the experimental testing as well as literature search [4]. More details about the characterization procedures as well as virtual characterization using multiscale RVE plugin can be found in the paper by Dev et al. [6].

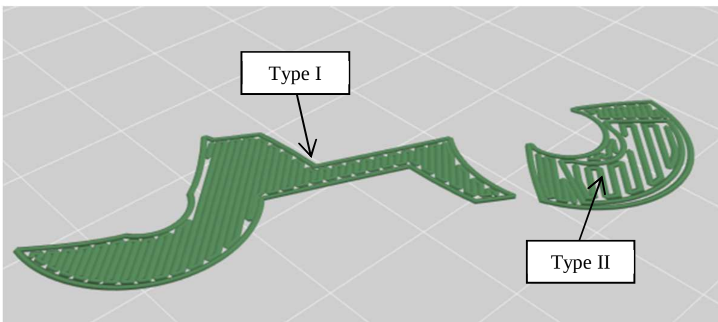
Description	Tensile Moduli (***GPa***)	Shear Moduli (***GPa***)	Poisson's

	E_{11}	E_{22}	E_{33}	G_{12}	G_{13}	G_{23}	ν_{12} (ν_{13})	ν_{21}(ν_{31})	ν_{23}(ν_{32})
Averaged Values	2.45	2.31	2.34	1.00	0.88	1.05	0.34	0.315	0.37

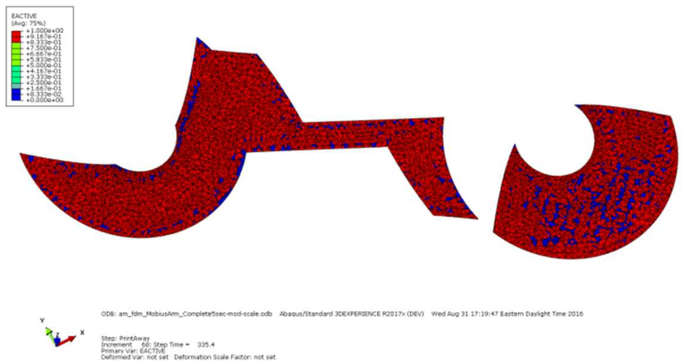
Description	Conductivity (*W/m K*)	Thermal expansion coefficient ($\mu m/mK$)	Specific heat (*kJ/kgK*)
Averaged Values	0.19	80	0˚C 1.62 105˚C 1.62 130˚C 3.00 280˚C 1.68

Table 1: Basic elastic moduli from experimental testing of printed coupons.

Voids commonly exists in an FDM process including the between beads voids mentioned in the previous section. Experiments of printed coupons will include the void effect and material properties obtained represent the slightly degraded properties after printing comparing to raw material due to these voids. Other types of voids also exists in FDM parts such as voids between printing styles, e.g., between the contour and infill (Type I) and as-designed voids for partial infill (Type II). *Figure 5* shows these types of voids and how they are captured by the element activation algorithm with CAD based tetrahedral mesh.



a. Voids between contour and infill (type I); voids for partial infill (type II)



b. Type I and type II voids captured by the element activation algorithm

Figure 5: Treatment of voids in the finite element simulation

4. Result and discussion

We use a Mobius arm part to illustrate the process simulation methodology with a focus on how the new Additive Manufacturing framework can be applied for even geometric complex parts. Simulation prediction such as temperature and distortion is compared to experimental printed samples qualitatively, with a target for quantitative prediction in future work.

First, the CAD model of the Mobius arm (*Figure 6*) was generated and imported into Stratasys printing preparation software. The Mobius arm part connects shafts with axes in different 3D directions and exhibits complex geometry that challenges the printing process. For example, the non-flat bottom surface requires usage of support materials; the rotating axis from bottom to top requires optimization of printing direction and tool path generation. Stratasys software handles the geometry and generates the detailed tool path for the Monbius arm. Support styles, fill styles can be chosen from a library. Different tray settings can also be used such as choice of slice height, model material type, support material type and etc. The tray can be arranged and the part/parts can be placed in desired positions. Multiple parts can be placed in the tray as well and tool path for multiple parts can be generated for one printing process. Geometry tools such as part scaling, unit system selection can also be performed. Stratasys software then slices the part with preview of the tool path (*Figure 7*).

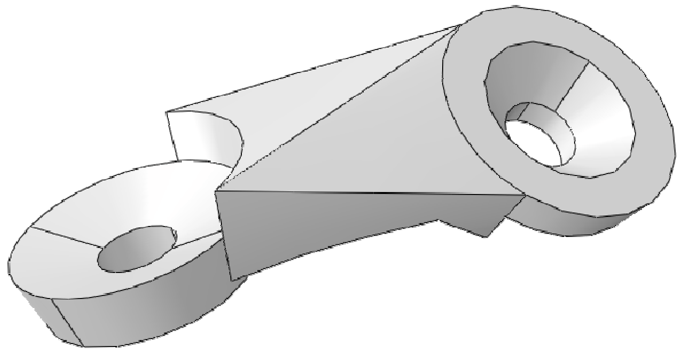


Figure 6: CAD model of a Mobius arm part

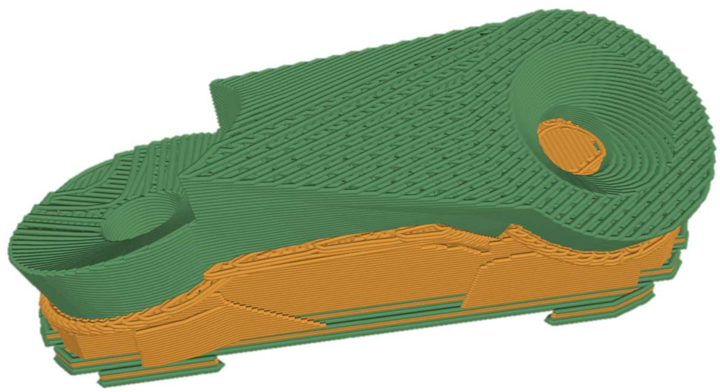


Figure 7: Tool path generation for a Mobius arm part with support and base

The detailed tool path is exported from Stratasys software in a readable text file format before converting to the Abaqus Event Series data and used in thermal mechanical simulations. The general python script written before was applied to perform the conversion. This includes automatic identification of the change in printing styles and printing regions and modification to proper data in Event Series for Abaqus to capture the exact movement and deposition ON/OFF processes.

Sequentially coupled thermal mechanical analysis is performed in Abaqus/Standard. This is done by a heat transfer analysis first. The temperature result and history is used to drive the static stress analysis. *Figure 8* is the nodal temperature result of the heat transfer analysis in the middle of the finite element simulation for the Mobius arm part using voxel mesh. The benefit of using voxel mesh is that since no CAD model is needed, we skip the complexity of generating support/base CAD models in the slicing software. One can also improve the computational performance of voxel mesh by starting with a block of voxel mesh and perform a single increment analysis to obtain IDs of all active elements and then re-run the analysis only using relevant elements. The partial infill patterns' area is captured well by the element activation algorithm. The spring back analysis is also performed by removal of the boundary condition at the bottom of the base material that simulates the build separation from the print bed.

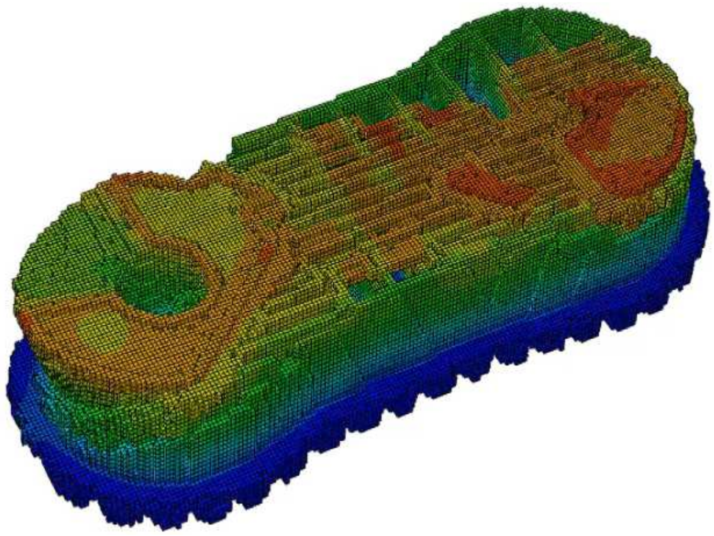
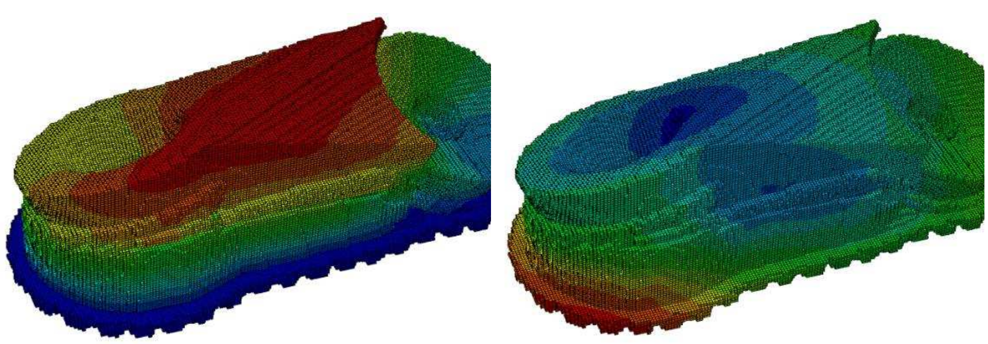


Figure 8: Temperature result in the middle of the finite element simulation for a Mobius arm part with support and base

The nodal temperature result at the end of the printing process and spring back before further cooling down is shown in *Figure 9-a*. The closer to the top of the part the higher the temperature is observed due to less time cooling down. The print tray is not simulated but simplified with fixed temperature boundary conditions at the bottom of the base material. Therefore, there could be a difference in the cooling down rate in the first few layers at the bottom of the base material.



a. Nodal temperature *b. Residual stress*

Figure 9: Finite element simulation results for a Mobius arm part with support and base

Figure 9-b shows the residual stress at the end of the static stress analysis and following spring back analysis. The bottom left and right shows the largest amount of residual stress and distortions. The Mobius arm part is curved up on two sides (left and right end in *Figure 9-b*) from the simulation prediction. This is consistent with the visual observation of the final printed part that bottom left and right curve up.

Detailed single or multiple layer finite element simulations can also be performed for interested layers that may have potential of delamination problems during printing (*Figure 10*). Detailed print path can be analysed and print path pattern can be analysed and optimized.

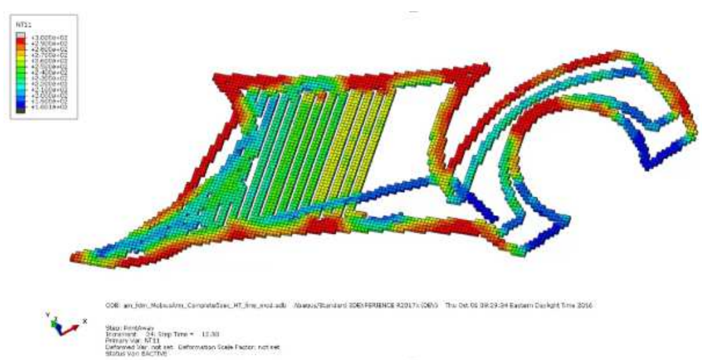


Figure 10: Detailed single layer finite element simulation results for a Mobius arm part with support and base

5. Future work

The new Additive Manufacturing process simulation framework is proved working efficiently and properly for the FDM process of a geometric complex part. The temperature history and distortion predictions are qualitatively correct comparing to the physical printed samples. Further research work are to be carried out to obtain realistic as-built material data for different types of FDM materials such as ULTEM as well as the characterization of support and base materials, including temperature dependent mechanical moduli; temperature dependent orthotropic conductivity; temperature dependent specific heat; temperature dependent orthotropic thermal expansion coefficient. Quantitative verification and validation is to be done starting from simpler geometries with distortion concerns to geometric more complex parts such as the Mobius arm. Temperature history will be compared to thermal scanned images and distortion will be compared to digital scanning of printed parts.

Other follow up research topics may include using multi scale homogenization tools to perform microstructure level analysis by driving repeating microstructure unit cell models by far field stress/strain history from macro scale process simulations and look at the micro structure level deformation as well as in service performance simulation with mapped residual stresses from process simulation results.

6. References

[1] Mahesh, M., Wong, Y. S., Fuh, Y. H., and Loh, H. T. (2004). Benchmarking for comparative evaluation of RP systems and processes. Rapid Prototyping J., 2004, 10(2), 123–135.

[2] Finnie, S., Cheng, W., Finnie, I., Drezet, J. M., and Gremaud, M. The computation and measurement of residual stresses in laser deposited layers. Journal of Engineering Materials and Technology, 2003, 125, 302–308.

[3] Beuth, J. L. and Narayan, S. H. Residual stress-driven delamination in deposited multi-layers. International Journal of Solids and Structures, 1996, 33(1), 65–78.

[4] Zhang, Y. and Chou, Y. K. (2006) Three-dimensional finite element analysis simulations of the fused deposition modelling process. Proceedings of the Institution of Mechanical Engineers, Part B: Journal of Engineering Manufacture. 2006, 220(10), 1663-1671.

[5] Abaqus Analysis User’s Guide, Abaqus 2017, Dassault Systèmes Simulia Corp., 2017.

[6] Dev, A., Courter, B., Bi, J., Savane, V., and Hansen, C. J. (2017). Mechanical Testing of FDM Parts for Process Simulation. Proceedings of the NAFEMS World Congress 2017, Stockholm, Sweden, June 11-14.