FDM for Composite Tooling 2.0

DESIGN GUIDE

stratasys®

DESIGN GUIDE

——

stratasys

FDM FOR COMPOSITE TOOLING 2.0

—

What's New in Design Guide 2.0?

The Stratasys Composite Solutions team is continually evaluating new techniques, characterizing performance and validating best practices to add value and expand the applications for FDM composite tooling. Version 2.0 of the design guide includes additional material data, empirical test results, tips, tricks, procedures, and new customer case studies. Specifically, new empirical data for sparse-style construction, recommendations on scribe lines, localized reinforcement (for such things as inserts) and additional mechanical characterization data has been added to this design guide. A full list of the sections in this guide that are new or include additional updated information are listed below:

FDM FOR COMPOSITE TOOLING 2.0

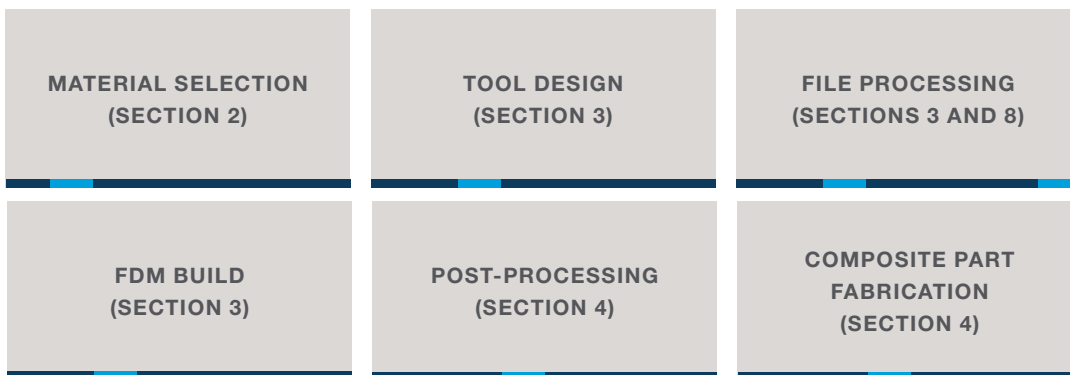## SECTION 1 – INTRODUCTION AND BACKGROUND

Scope

This technical design guide describes the design, processing, manufacture, and post-processing techniques and procedures for additively manufactured composite lay-up tooling using Stratasys FDM® technology. The principles discussed and requirements provided in this guide for part creation and implementation should be followed whenever possible. Due to the various composite industry best practices, deviations to this design guide may be implemented at the discretion of the individual user's expertise.

Application Overview

FDM® (fused deposition modeling) is becoming the technology of choice for cost effective rapid production of low volume tooling and manufacturing aids. FDM technology can also be utilized for the production of high-temperature (>350 °F [177 °C]), low-volume, composite lay-up and repair tools, as well as for high-temperature (<350 °F [177 °C]) production sacrificial tooling.

Relative to traditional tooling materials and methods, FDM offers significant advantages in terms of lead time, tool cost and simplification of tool design, fabrication and use, while enabling increased functionality and geometric complexity. This design guide is focused on tools for hand lay-up, but the vast majority of the principles and guidelines are applicable to other processing methods as well.

An overview of the process for producing FDM lay-up tooling follows. Additional details regarding each step in the process will be provided in the referenced sections.

MATERIAL SELECTION (SECTION 2)	**TOOL DESIGN (SECTION 3)**	**FILE PROCESSING (SECTIONS 3 AND 8)**
FDM BUILD (SECTION 3)	**POST-PROCESSING (SECTION 4)**	**COMPOSITE PART FABRICATION (SECTION 4)**

Background and Purpose

Traditional manufacturing methods for high-performance, fiber-reinforced polymer matrix composite structures require hard tooling for the mold or mandrel to control the surface profile of the final part. These tools are commonly made of metal (aluminum, steel, or Invar alloys), although non-metallic materials like high-temperature tooling board and specialized composite tooling materials are also used. Regardless of material, tool fabrication typically requires significant labor and machining, which leads to high costs, material waste, and long lead times of weeks for relatively simple shapes and many months for more complex tools.

In contrast, FDM technology has demonstrated considerable cost and lead-time reductions for composite tooling while providing numerous other advantages such as design freedom and rapid iteration, regardless of part complexity. It has been successfully used for low-volume composite lay-up and repair tooling applications for years. However, its use was limited by the lack of materials capable of the 350 °F (177 °C) cure temperature required for aerospace and similar high-performance structures, and the absence of design knowledge and guidance.

FDM FOR COMPOSITE TOOLING 2.0

—

Regarding materials limitations, FDM ABS-M30/ASA, polycarbonate (PC), and ULTEM™ 9085 resin are effective up to 180 °F (82 °C), 270 °F (132 °C), and 300 °F (149 °C), respectively. With the introduction of ULTEM 1010 resin, FDM technology has demonstrated numerous advantages for fabrication of composite structures cured at temperatures exceeding 350 °F (177 °C) and pressures of 100 psig (690 kPa).

This design guide provides best practices for the design, fabrication and preparation of 3D printed composite tools, as well as relevant performance characterization data.

NOTE Although applicable to other materials and slice heights, the data and recommendations provided in this guide are based on tooling built with ULTEM 1010 resin using a 0.020 inch slice height, unless otherwise noted.

Benefits of FDM for Composite Tooling

- Reduces lead time from months to days

- Lowers tooling costs by >50%

- Enables cost-effective composite part prototyping

- Simplifies tool design and fabrication with increased functionality

- Withstands high-temperature autoclave and oven cure cycles [>350 °F (177 °C), 100 psig (690 kPa)]

- Provides low-hassle sacrificial and wash-out solutions for complex, trapped-tooling applications

- Permits trouble-free design changes and iteration

- Reduces tool mass by >80% (relative to metallic tooling in particular)

Application Best Fits

This application is most suitable in the following conditions:

- Lay-up and repair tools required in days, not months

- Sacrificial tooling cures at moderate temperatures [<350 °F (177 °C)]

- Part volumes are relatively low (10s – 100s vs. 1000s)

- Tool sizes fit within the build volume of the Fortus 900mc™ 3D Printer, although segmented tools have been successfully demonstrated as well

- Tool geometries that can be adjusted to compensate for thermal expansion or benefit from higher CTE materials (e.g., male mandrels for increased ply consolidation)

Design Guide Objectives

This design guide aims primarily to provide:

- Overview of FDM technology

- Key properties and characteristics for relevant materials

- Advantages and key considerations for composite tooling

- Best practices for design, construction and optimization of lay-up tools

- Best practices for file preparation, processing and fabrication

- Best practices for post-processing lay-up tools (preparation and sealing)

- Use-case examples

- Tool life and characterization data

- Introduction to sacrificial tooling

stratasys

FDM FOR COMPOSITE TOOLING 2.0

Design Guide Approach

This guide is broken into key sections that provide the necessary information to efficiently and successfully produce, prepare and use FDM composite lay-up tooling, referred to herein as "FDM composite tooling." It offers technical information, material properties and test data to demonstrate the performance of FDM composite tooling. Stratasys has worked with industry leaders and tooling experts from aerospace, automotive, sporting goods and academia to characterize and validate performance. Key use cases and examples from these collaborative development efforts are provided, although partner identity is often concealed to protect proprietary information.

Two key partners were Aurora Flight Sciences (AFS) and Abaris Training. AFS is a recognized leader in aviation and aeronautics research that specializes in the design and construction of special-purpose aircraft. In the development and production of multiple manned and unmanned aircraft structures, AFS worked with Stratasys to implement FDM for composite tooling and ancillary tooling (jigs, fixtures, trim tools, etc.), as well as fly-away parts. Stratasys also worked closely with Abaris Training, the world-renowned leader in advanced composites training, for additional technical input, tool evaluation, and the development of FDM composite tooling training curriculum.

Overview of FDM

FDM is a Stratasys-patented additive manufacturing technology that builds parts layer-by-layer by heating and extruding thermoplastic filament. FDM builds using standard, engineering-grade and high-performance thermoplastics.

The FDM process begins by processing the CAD file using Insight™ software, which comes with the 3D printer. This software allows the user to select all of the parameters for the build, from slice height to part orientation, providing capability for complete part customization. FDM machines are capable of dispensing two materials, serially: the primary model material that makes up the final part and a secondary support material used as required to prevent collapse in areas of overhangs. Since Stratasys FDM support material is inherently designed to be sacrificial and easily removable, it is a practical material from which to produce one time use, sacrificial layup tooling. More information on sacrificial lay-up tooling is available in section 7.

FDM filament is wound into canisters that feed material through the system to an extrusion nozzle, or "build tip." The build tip is heated by a liquefier, melting the material while depositing it in both primary horizontal axes (x, y) in a temperature-controlled chamber, following a numerically-controlled toolpath. Upon completion of each layer, the build platen moves vertically (z direction), to make room for the next layer to be deposited above.
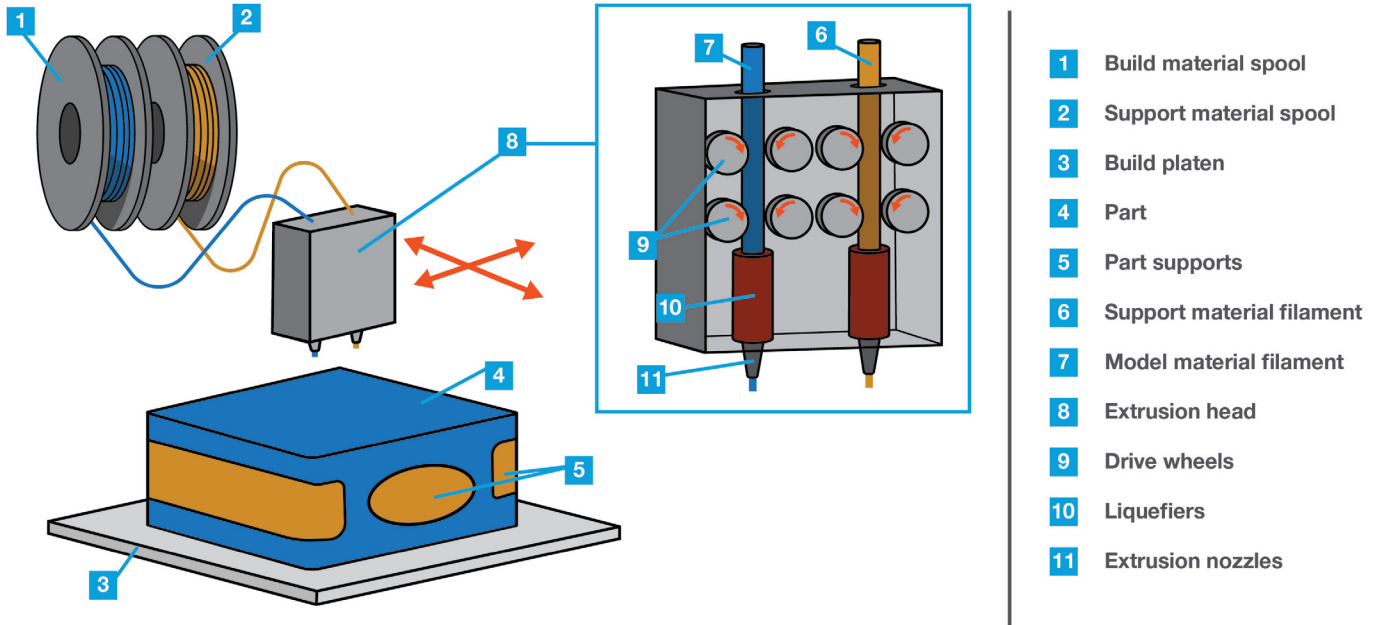
Figure 1-1: The main components of an FDM printer.

1	Build material spool
2	Support material spool
3	Build platen
4	Part
5	Part supports
6	Support material filament
7	Model material filament
8	Extrusion head
9	Drive wheels
10	Liquefiers
11	Extrusion nozzles

Key Design Considerations

Just as design and construction aspects of traditional lay-up tooling varies depending on the material used, effective design and use of FDM composite tooling relies on these considerations:

- Cure temperature
- CTE
- Accuracy and tolerance requirements
- Process parameters (consolidation pressure and vacuum bagging approach)
- Tool preparation (sealing)
- Anticipated use (tool life)

Cure Temperature

The cure temperature of the composite structure is a significant factor in FDM material selection. FDM materials are capable of covering a broad range of cure temperatures, as shown in Figure 1-2 on the next page.

As shown in Figure 1-2, ULTEM 1010 resin has the highest temperature capability of the relevant FDM materials. It also has the lowest CTE, making it the preferred choice for the majority of composite tooling applications. While tools made from PC and ULTEM 9085 resin can withstand the cure cycle for a 250 °F (121 °C) cure material system, ULTEM 1010 resin is still the most appropriate choice to minimize expansion impacts. Additional material properties can be found in Section 2.
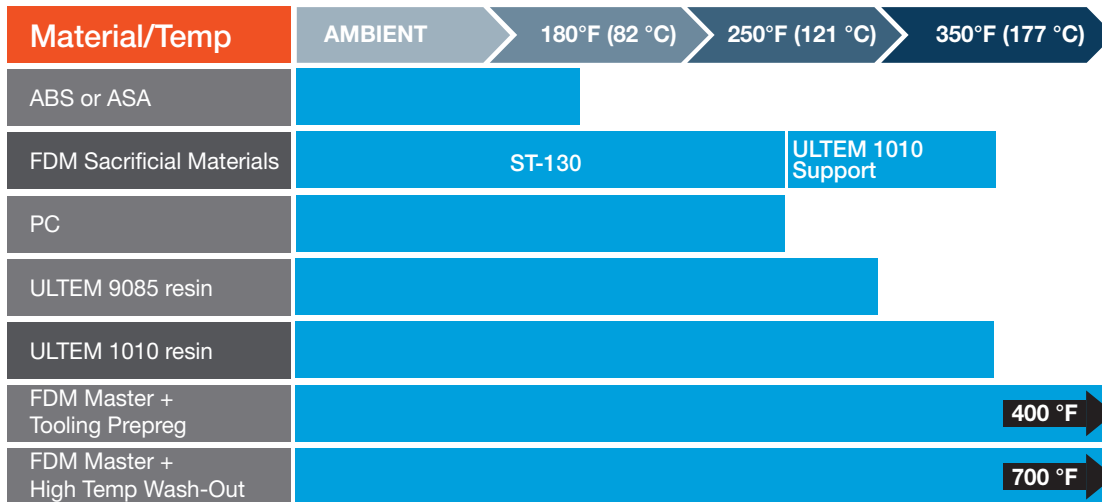
FDM FOR COMPOSITE TOOLING 2.0

Material/Temp	AMBIENT	180°F (82 °C)	250°F (121 °C)	350°F (177 °C)
ABS or ASA				
FDM Sacrificial Materials	ST-130		ULTEM 1010 Support	
PC				
ULTEM 9085 resin				
ULTEM 1010 resin				
FDM Master + Tooling Prepreg				400 °F
FDM Master + High Temp Wash-Out				700 °F

Figure 1-2: Approximate cure temperature capability for FDM tooling materials.

Coefficient of Thermal Expansion

CTE is an important consideration for nearly all composite lay-up tooling since it impacts the final physical shape of the composite structure. Table 1-1 lists the CTE for relevant FDM materials as well as common conventional tooling materials. As a result of the relatively high CTE of FDM materials, it is an important consideration during tool design. Tool designs can and typically should be modified to compensate for the dimensional changes related to thermal expansion at elevated temperatures. Examples of such adjustments are provided in Section 3. In addition to geometric compensation, CTE differences between the tool and part materials are also factors that impact tool type (male versus female tools) and potential complexity. For male tools, simply sizing the tools to compensate for growth is usually adequate. And for some applications, such as mandrels for winding/wrapping, the CTE can be used advantageously to improve ply consolidation and simplify mandrel removal. For female tools, particularly those with steep contours and deep drafts, additional care is required to ensure parts can be safely removed from the tool without inducing damage, as well as to manage residual stresses imparted on the resulting parts. More in-depth examples of successful use of both male and female tools are provided in Section 6.

CTE comparison for FDM and traditional tooling materials

FDM MATERIALS	µIN / (IN•°F)	µM / (M•°C)
ST-130 (soluble/sacrificial)	59	106
ABS/ASA	49	88
PC	38	68
ULTEM 9085 resin	36	65
ULTEM 1010 resin	26	47
ULTEM 1010 Support	33	59
CONVENTIONAL TOOLING MATERIALS	µIN / (IN•°F)	µM / (M•°C)
Tooling Boards	20-40	36-72
Aluminum alloy (AL 6061-T6)	13	23
Tool Steel	6.5	12
Carbon/epoxy	2.0	3.6
Invar	1.0	1.8

Table 1-1

stratasys

FDM FOR COMPOSITE TOOLING 2.0

—

For details on how to modify tool designs to compensate for the effects of CTE, refer to CTE Compensation in Section 3. Details for calculating a scaling factor to modify tool geometries are provided.

Accuracy and Tolerances

FDM is capable of producing tools with accuracies of ±0.0035 inch (0.09mm) or ±0.0015 inch/inch (0.0015 mm/mm), whichever is greater. Note that all accuracies are geometry dependent, primarily due to the thermal nature of the process. Additional information on machine accuracy can be found on www.stratasys.com (including a white paper on the topic). For development of this guide, accuracy data was compiled for various representative geometries, both before and after thermal cycling. Refer to Section 5 for additional data and details.

For composite tooling that requires greater accuracy than can be achieved directly from the FDM 3D printer, production of near-net-shape tools, combined with skim-coat machining is a viable option. Design recommendations and preliminary data for machining recommendations are listed in Section 3, Design Recommendations for Near-Net-Shape Tooling. Additional development work is underway on this topic and will be provided in subsequent design-guide releases.

Process Parameters

Fabrication process and cure cycle parameters, particularly cure pressure and vacuum bagging method, impact the design and style of FDM composite tools. They are generally classified as shell-style or sparse-style tools. (See Figure 1-3 below.) Additional information is provided in Section 3.
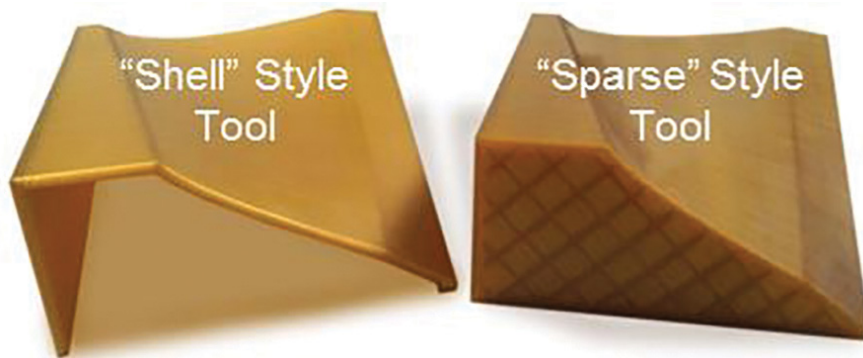

Figure 1-3: UAV Fan-blade tools showing examples of shell and sparse tools.

Shell-style tools are effective for most applications, able to withstand 100+ psig (690+ kPa) autoclave pressure and conducive to both surface and envelope vacuum bagging methods. For many geometries, they are the most cost-efficient design since they minimize material use and build time. Sparse-style tools tend to have greater overall rigidity; some geometries require their use. This will be demonstrated in more detail in Sections 3 and 6. Sparse tools can also be surface or envelope bagged. However, when envelope bagging is used, follow the guidelines in Section 3 regarding construction parameters to avoid damaging the tool.

Tool Preparation

The FDM process inherently produces some level of internal porosity due to physical limitations of the extruded material beads, as depicted in Figure 1-4, which shows the cross-section of toolpaths for an example build layer and the cross-section of extruded bead profiles. The process also produces perceptible build layers, which vary based on the shape of the part and the layer thickness (slice height). As a result, to ensure a high-quality surface finish and vacuum integrity, post-processing of FDM tools is typically required.
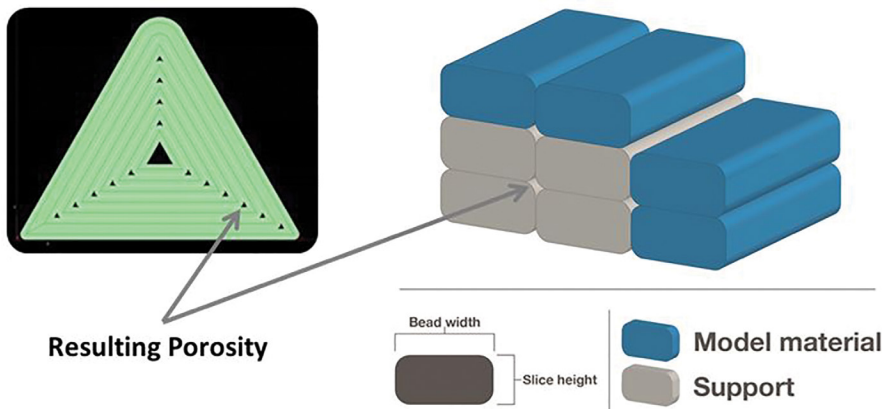
FDM FOR COMPOSITE TOOLING 2.0

Figure 1-4: Top view of an example toolpath (left) and cross-section of bead profiles (right) showing inherent porosity in FDM parts.

Tools are abraded to smooth out perceptible build lines, and sealed. They then undergo a final polish, resulting in surface finishes consistent with typical industry requirements. Although requirements do vary somewhat across industries, a finish of 64 µinches (1.6 µm) Ra is generally considered acceptable. Using the standard procedure provided in Appendix B, a finish smoother than 16 µinches (0.4 µm) Ra can be consistently achieved on FDM composite tools. Sealing can be performed using a range of materials depending on the specific application. The most common materials used to-date have been high-temperature, two-part epoxy adhesives. Epoxy film adhesives, adhesive-backed FEP films and similar products have also been successfully used and have distinct advantages such as ease of application, depending on the requirements of the specific use. Additional information and specific products evaluated to-date are provided in Section 4. Once the part is sealed, common mold-release agents can be applied in preparation for composite part layup. Water-based released agents are recommended.

Anticipated Use and Tool Life

The final consideration for successful design and use of FDM composite tooling is an understanding of the intended application of the tool. The application tends to drive material selection (e.g., cure temperature requirements) and the overall design, and factors into the tool construction and sealing approach (i.e., will the tool be envelope- or surface-bagged and what is the consolidation pressure?). It is also important to evaluate the intended tool life, dictated by the number of cure cycles the tool will experience. Tools intended for a few prototype composite parts can be constructed to minimize cost. Tools intended for an impending, schedule-critical composite repair can be optimized for quick build time. And tools intended for longer-term production use and higher part volumes require greater scrutiny regarding nearly all aspects.

The majority of FDM composite tool use has been for relatively low part volumes (<25 parts). However, in the development of this guide, tool-life characterization testing was initiated and the resulting data indicates that FDM composite tooling is capable of much longer tool life — hundreds of cycles. See Section 5 for more information. Additional tool-life evaluation is ongoing and results will be provided in subsequent design-guide releases.

SECTION 2 – FDM MATERIALS

FDM technology produces tools in a wide range of high-performance thermoplastic materials. Each material has advantages and limitations that must be considered for effective use in composite part fabrication. Application requirements will guide material selection. As general guidelines, ULTEM 1010 resin is the recommended material for nearly all layup tooling (molds and mandrels) and either ABS or ASA is highly effective for ancillary tools (trim tools, holding fixtures, drill guides, etc.), as well as low-temperature masters.

Properties of FDM parts will be anisotropic, primarily due to the nature of the build process. The anisotropy tends to impact mechanical properties; thermal properties, such as CTE, are also impacted, albeit to a lesser extent. For example, the difference in

FDM FOR COMPOSITE TOOLING 2.0

—

CTE between the "flow" (parallel to the extruded bead) and "cross-flow" (perpendicular to the extruded bead) for ULTEM 1010 resin is less than 4%.

The primary role of this guide with regard to FDM materials is to aid in selection and give a sense for capabilities. Additional information and data for all Stratasys materials can be found on www.stratasys.com. Table 2-1 below provides guidance on the FDM materials most relevant for composite tooling applications.

FDM material guidance and general information

| MATERIAL | TYPICAL USE | ADVANTAGES | LIMITATIONS | TG | | CTE | | | |
| | | | | | | µIN/IN-°F | | µM/M-°C | |
				°F	°C	Flow	X-Flow	Flow	X-Flow
ULTEM 1010 resin	All lay-up tools	• Highest temp capability • Lowest CTE	Higher CTE relative to traditional mold materials	419	215	26	25	47	41
ULTEM 9085 resin	Lay-up tools for cure temps <300 °F (149 °C)	Moderate temp capability	• Higher CTE than ULTEM 1010 resin • Not suitable for 350 °F (177 °C) cure temps	367	186	37	--*	65	--*
PC	Lay-up tools for cure temps <270 °F (132 °C)	Lower cost option for low volume, low-temp tools	• High CTE • Not suitable for high cure temps	322	161	38	--*	68	--*
ABS-M30	• Low temp <180 °F (82 °C) masters and patterns • Trim/drill fixtures • Other jigs and fixtures	• Lowest cost • Multiple color options	• High CTE • Not suitable for even moderate cure temps	226	108	49	47	88	84
ASA	• Low temp <180 °F (82 °C) masters and patterns • Trim/drill fixtures • Other jigs and fixtures	• Lowest cost • UV Stable • Multiple color options	• High CTE • Not suitable for even moderate cure temps	226	108	49	46	88	82
ST-130	Sacrificial, wash-out tooling (<250 °F, 121°C)	Soluble material for trapped-tool applications	Not suitable for high cure temps	269	132	59	--*	106	--*
ULTEM 1010 Support	High temp (<400 °F, 204°C) sacrificial tooling	• Higher temp capability vs. ST-130 and ULTEM S1 • Can be embrittled with acetone	Not soluble; must be manually removed	437	225	33	--*	59	--*

Table 2-1   * - Denotes unavailable data.

stratasys

FDM FOR COMPOSITE TOOLING 2.0

SECTION 3 – TOOL DESIGN AND CONSTRUCTION

Design Considerations and Impact

The advantage of an FDM composite tool versus a traditionally manufactured tool is that an FDM tool can have a complex, highly functional design without sacrificing cost or lead time. The design process for an FDM tool is primarily driven by the process parameters for the final composite parts (cure cycle, pressure, bagging approach, etc.). In general, within this guide, tool designs are classified into two main styles: shell and sparse. A shell tool is a relatively simple approach that provides the layup surface of the tool, extended beyond the edge of part (EOP), built at a thickness to provide stability with minimal extraneous material use. Similarly, a sparse tool uses the basis of the shell tool, but reinforces it with a sparse double dense (or similar fill) raster pattern. FDM composite tooling is not limited to these two styles — designs can be as complex, as simple, or as functionally oriented as the application requires. The designs presented in the following sub-sections are intended to demonstrate two basic styles of FDM composite tool design and production.

NOTE Sparse construction in this design guide refers to sparse double dense and hexagram fill patterns in Insight software. The available pattern in Insight known simply as "sparse" (refer to the definition in Appendix C), which provides raster fill in only a single direction, is not recommended for composite tooling applications.

Figure 3-1: Sparse- and shell-style tools for a UAV fan blade.

Shell-Style Tool

The shell-style tool demonstrates the advantages of FDM by using the least amount of material without sacrificing tool performance or resulting part quality. In many cases, only the actual lay-up surface (extended beyond the EOP to provide room for excess material) needs to be 3D printed – no elaborate support structure or backing is required. The thickness of such tools can vary, but empirical data has shown that 0.3 inch (7.6 mm) provides a balance between tool rigidity and material consumption. This style of tool can be envelope or surface bagged, but envelope bagging is recommended when feasible for simplicity and reduced potential for vacuum leaks. Tools of this type can withstand autoclave pressures in excess of 100 psig (689 kPa).

Sparse-Style Tool

Although shell tends to reduce cost and build time, certain designs may require additional rigidity, particularly for large and/or multi-segment tools. Sparse tools incorporate a raster fill pattern for additional strength and rigidity. This design can be envelope bagged or surface bagged, but surface bagging is most common to eliminate any susceptibility to crushing if the raster spacing isn't dense enough.

FDM FOR COMPOSITE TOOLING 2.0

Table 3-1 provides general design guidance for the raster spacing or density of sparse tools relative to consolidation pressure **when using envelope bagging methods**. It should be noted the guidelines in the table are for the default sparse double dense and hexagram fill patterns in combination with ULTEM 1010 resin with a slice height of 0.020 inch (0.508 mm) only. Other patterns (e.g., hexagonal and custom fills) and materials will follow similar guidelines, but performance has yet to be verified. Results may also vary based on the specific tool geometry.

It was determined experimentally that wall thickness (i.e., the number of <u>contours</u>) had minimal impact on results – meaning, increasing wall thickness did not significantly improve performance. The results supporting the guidance in Table 3-1 are for a wall thickness of 0.1 inch (2.5 mm). The geometry of the test coupons is shown in Figure 3-2.

For testing in support of the guidelines in Table 3-1, the test coupon shown in Figure 3-2 was built varying the wall thickness from 0.1 to 0.3 inch (2.5 to 7.6 mm) and the sparse raster spacing from 1.0 inch (25.4 mm) down to less than 0.08 inch (2.03 mm). Results for sparse double dense coupons with 1.0 inch (25.4 mm) spacing are not shown as they had damage/crushing at even the lowest consolidation pressures. All coupons were tested at 350 °F (177 °C).

Table 3-1 also provides information regarding the relative material use for each fill pattern configuration to shed light on relative tool cost. For instance, the cost of a tool with 0.25 inch (6.35 mm) sparse double dense raster spacing would be approximately 30% more than the same tool with 0.5 inch (12.7 mm) raster spacing and 40% more than 1.0 inch (25.4mm) hexagram spacing. Although build time is not included, it typically scales at slightly less than the same rate, so a conservative relative cost calculation remains valid for rough order magnitude comparisons.

Fill pattern raster spacing for sparse tools when envelope bagging**

CONSOLIDATION PRESSURE	WALL THICKNESS	MAXIMUM SPARSE DOUBLE DENSE SPACING	RELATIVE MATERIAL USE*	MAXIMUM HEXAGRAM SPACING	RELATIVE MATERIAL USE*
<40 psig (276 kPa)		0.50 inch (12.70 mm)	1.0	1.00 inch (25.40 mm)	0.9
<60 psig (414 kPa)	0.10 inch (2.54 mm)	0.25 inch (6.35 mm)	1.3	1.00 inch (25.40 mm)	0.9
<80 psig (552 kPa)		0.10 inch (2.54 mm)	2.2	0.75 inch (19.05 mm)	1.0
>80 psig (552 kPa)		<0.08 inch (2.03 mm)	2.5	< 0.75 inch (25.40 mm)	>1.0

Table 3-1   *Relative material use is the material use compared to the equivalent geometry with 0.5 inch (12.7 mm) Sparse Double Dense spacing. **Bag was vented to atmosphere during cure cycle.
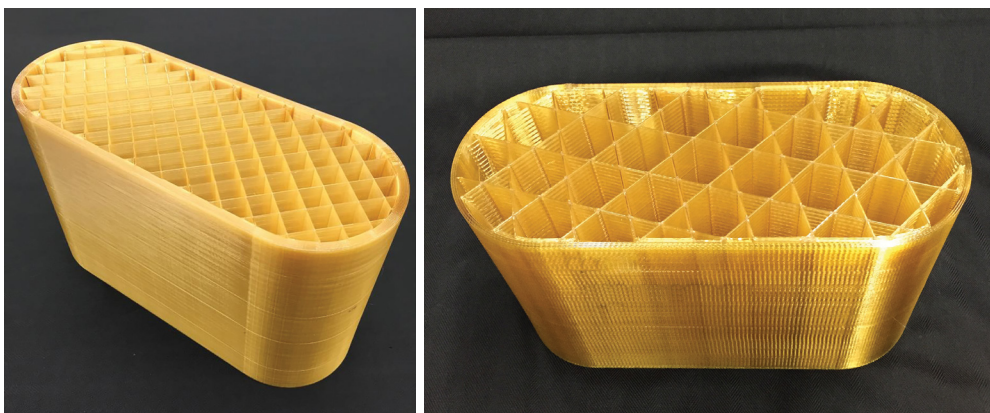

Figure 3-2: Test coupons for structural integrity testing of envelope-bagged, sparse style tools, ULTEM 1010 resin. Left image shows sparse double dense raster spacing of 0.5 inch (12.7mm). Right image shows Hexagram raster spacing of 1.0 inch (25.4 mm).

FDM FOR COMPOSITE TOOLING 2.0

Tool Build Orientation

The orientation of a tool in the FDM machine is an important consideration because it will affect the build time, the amount of support material required, and the resulting surface quality (stair-stepping), as well as overall performance (due to anisotropy of properties). And it is valuable to consider the build orientation during, rather than after, the initial design of the tool since choices made at this stage play a role in the build orientation and thus impact the final performance and cost of the tool.

In general, it is recommended to orient the layup surface of the tool such that it is printed in a "vertical" orientation (refer to Figures 3-3 through 3-5), which is most effective at minimizing stair-stepping, support material use, and build time, all of which directly factor into the cost of the tool. If the tool has multiple, highly contoured surfaces, efforts should be made to orient the tool so that the majority of the surface will exhibit the least amount of stair stepping. The following examples show the primary orientations for producing FDM tools. The described orientations are not the only orientations that can be used, but are intended to provide a reference to show the advantages and disadvantages of each.

Flat Build Orientation: A "flat" orientation, as shown below, tends to be the least preferred as it typically has the most stair-stepping and requires the most support material. Therefore, a tool built in this orientation will take longer to post-process and have a higher cost due to the support material required.
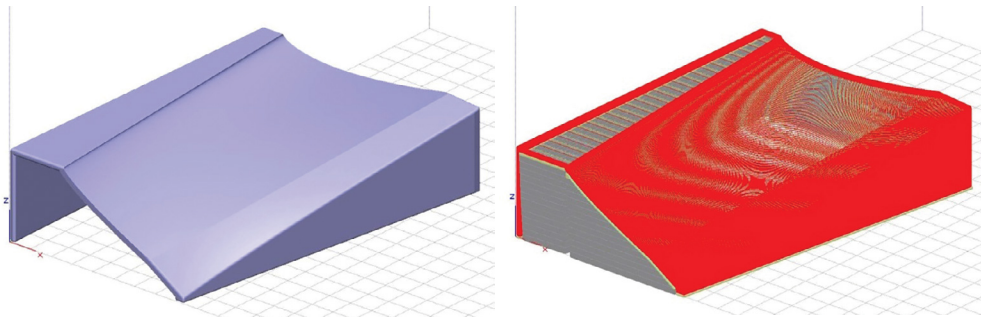
Figure 3-3: "Flat" build orientation.

Sub-Optimal Vertical Build Orientation: Although the tool in Figure 3-4 is in a "vertical" build orientation, it is not the ideal orientation. This orientation will effectively minimize stair-stepping; however, a large amount of support material is required to support the "legs" of the tool, which results in increased build time and cost.
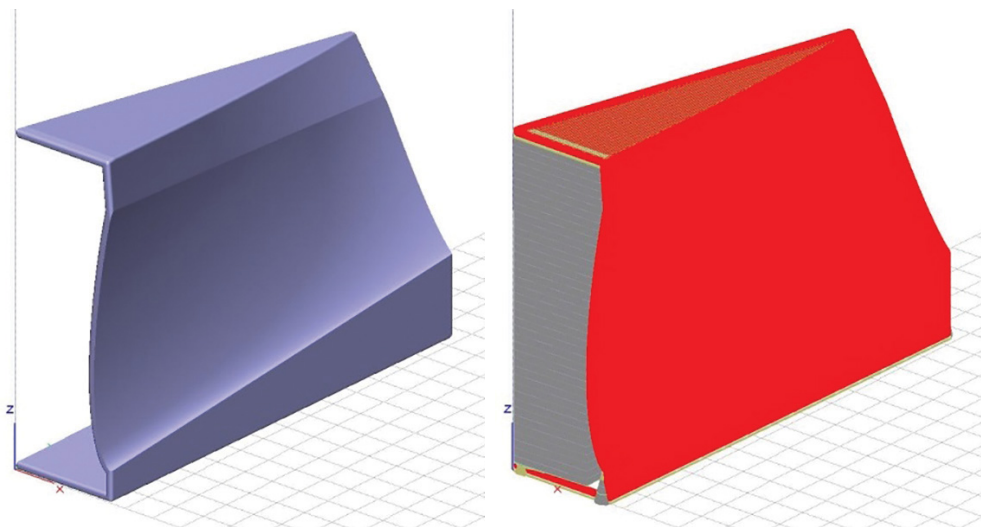

Figure 3-4: Sub-optimal "Vertical-A" build orientation.

stratasys

Optimal Vertical Build Orientation: The vertical build orientation shown in Figure 3-5 (Vertical-B) is preferred as it will help minimize stair-stepping while also conserving support material. As a result, tools built in this orientation will require less build time and post-processing time, as well as material cost.
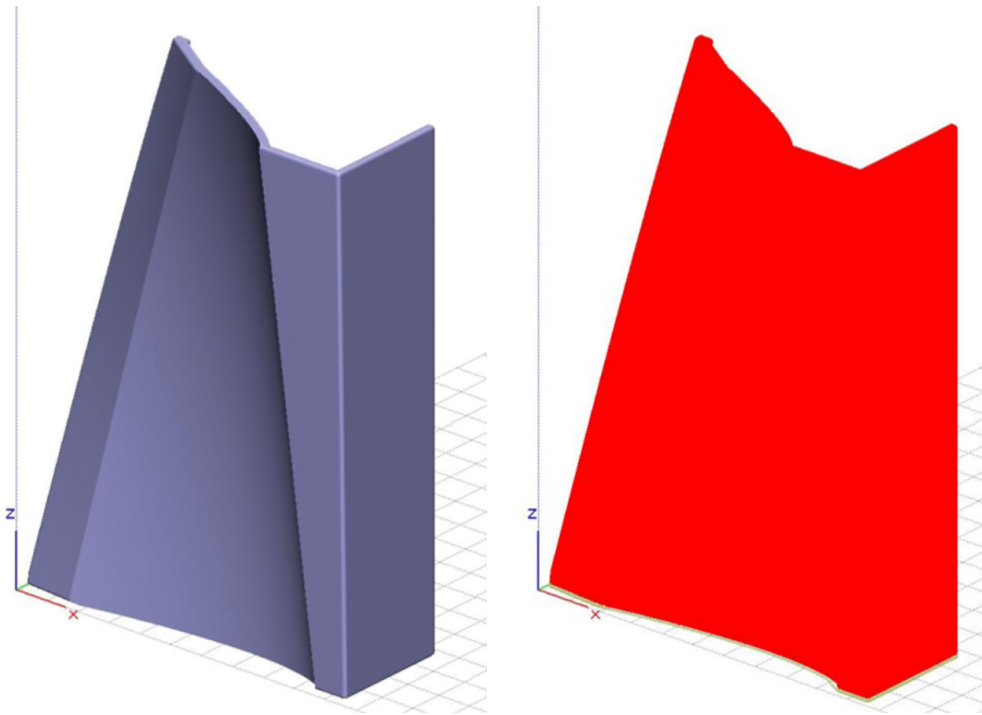

Figure 3-5: Optimal "Vertical-B" build orientation.

Table 3-2 illustrates the impacts of build orientation on build time, material use and surface quality for a given tool. As expected, the amount of model material required to build the tool is essentially the same, regardless of orientation. The impact on build time is quite significant because it can take twice as long to build tools that are not oriented optimally. And finally, the difference in support material consumption is dramatic for this particular example since the optimal orientation uses a near-negligible amount (<1 cubic inch).

Relative comparison of build orientations

ORIENTATION	RELATIVE BUILD TIME	RELATIVE MODEL MATERIAL USE	RELATIVE SUPPORT MATERIAL USE	RELATIVE STAIR-STEPPING
Flat	2.0	1.0	550	High
Vertical A	2.0	1.04	420	Low
Vertical B	1.0	1.0	1.0	Low

Table 3-2

stratasys

FDM FOR COMPOSITE TOOLING 2.0

CTE Compensation

As previously stated, it is important to consider the impacts of CTE from the onset of tool design. The dimensional change of a tool can be calculated using the equations below, as well as a scaling factor that can be used to modify the geometry of a tool to compensate for expansion at elevated temperatures. The scaling factor is used to adjust for tool expansion at the maximum cure temperature, T_{cure}. The initial or starting temperature, $T_{initial}$, is typically room temperature. The variables $\propto_{tool}$ and $\propto_{part}$ represent the CTE of tool and part, respectively.

$$Expansion\ factor = (T_{cure} - T_{initial}) \times (\propto_{tool} - \propto_{part})$$

Dividing one by one plus the expansion factor will provide the scaling factor, by which the tool will need to be adjusted to produce composite parts with the proper final size, shape and dimensions.

$$Tool\ scaling\ factor = \frac{1}{1 + Expansion\ factor}$$

To demonstrate this, assume an ULTEM 1010 resin tool will be subjected to a 350 °F cure cycle. The scaling factor is calculated as shown below.

$$T_{cure} = 350\ °F$$
$$T_{initial} = 70\ °F$$
$$CTE_{ULTEM\ 1010} = 26 \times 10^{-6}\ (in./in./*°F)$$
$$CTE_{part} = 1.5 \times 10^{-6}\ (in./in./*°F)$$

$$Expansion\ factor = (350\ °F - 70\ °F) \times (0.000026\ \frac{in.}{in.* °F} - 0.0000015\ \frac{in.}{in.* °F}) = 0.00686$$

$$Tool\ scaling\ factor = \frac{1}{1 + 0.00686} = 0.993149$$

This implies the tool will expand by 0.00686 inch per inch (0.00686 mm/mm) of tool length (at 350 °F, 177 °C) and the tool will need to be scaled by a factor of 0.993149 to compensate for that dimensional change and produce a composite part with the proper final geometry.

stratasys

FDM FOR COMPOSITE TOOLING 2.0

—

The exact method or steps for scaling the tool will vary based on the CAD software being used, but for SOLIDWORKS select:

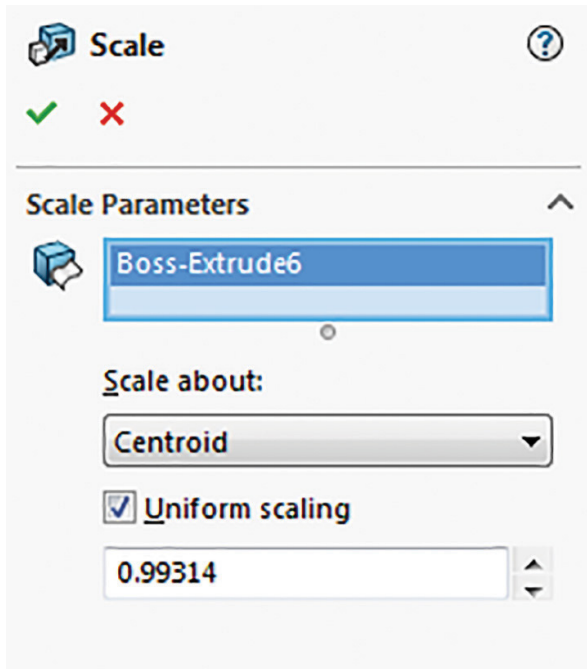**Insert → Features → Scale → Enter the appropriate scaling factor**

Figure 3-6: Model scaling menu in SOLIDWORKS.

Tips for designing FDM composite tools

The following are general tips for cost-effective design of FDM tools.

1. To minimize costs, it is desirable to print the least amount of material possible. In many cases, this means using a shell-style tool capturing primarily just the layup surface of the part without any elaborate support structure.

2. Use self-supporting angles (refer to Appendix C) to minimize the amount of support material required. Overhanging features require support material, which increases the amount of material required and build time.

3. Orient the tool such that the layup surface is printed in a vertical orientation. This orientation typically produces the best surface finish by reducing stair-stepping. See Tool Build Orientation (Section 3) for reference.

Design and Modification of FDM Composite Tools

The following sub-sections contain information on how to design, modify and optimize male and female shell- and sparse-style tools. It is also recommended to adjust final design geometries to compensate for thermal expansion as previously detailed (refer to procedures for determining the tool scaling factor in the previous CTE Compensation sub-section).

DESIGN GUIDE

—

Design and Modification of FDM Composite Tools

The following sub-sections contain information on how to design, modify and optimize male and female shell- and sparse-style tools. It is also recommended to adjust final design geometries to compensate for thermal expansion as previously detailed (refer to procedures for determining the tool scaling factor in the previous CTE Compensation sub-section).

Figure 3-7: Desired shape of composite part.
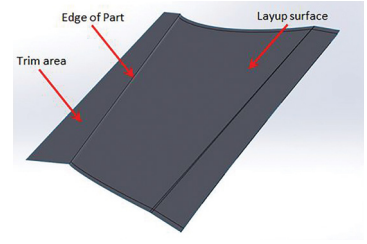
Figure 3-8: Adding the EOP and trim area.

Male Shell Tool Design

1. Begin with the shape (model) of the desired composite part as this will establish the layup surface, EOP, and the trim area of the part (Figure 3-7).

2. Create the trim area of the part by extending surfaces outside the EOP (Figure 3-8).

3. Thicken the tool surface to 0.3 inches (7.6 mm) (recommended for most tools). The thickness may be adjusted as needed based on the specific tool configuration and application requirements (Figure 3-9).

Figure 3-9: Thickening the layup surface.
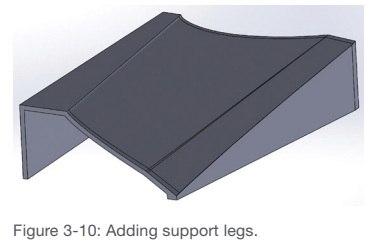
Figure 3-10: Adding support legs.

4. Add stabilizing features such as legs, permitting the tool to sit flat on a table during layup. Alternatively, a low temperature cradle could be printed instead of legs to support the shell tool during lay-up. Stiffening features can also be added, if required. Identification features such as tool numbers can also be incorporated (Figure 3-10).

5. Add fillets at corners to reduce stress concentrations and improve robustness (Figure 3-11).

6. Round sharp corners and edges to prevent piercing of vacuum bagging materials, particularly when envelope bagging. A fillet radius of 0.1-0.5 inch (2.5-12.7 mm) is typically adequate (Figure 3-12).

Figure 3-11: Addition of large-radii fillets to corners.

Figure 3-12: Rounding sharp edges and corners.

The toolpath can now be prepared using Insight. As previously discussed, it is recommended to build the tool in the vertical orientation shown in Figure 3-13 for the best surface finish and least support-material consumption.

Shell tools will have a solid fill raster pattern with a minimum of three contours, as shown in Figure 3-14.
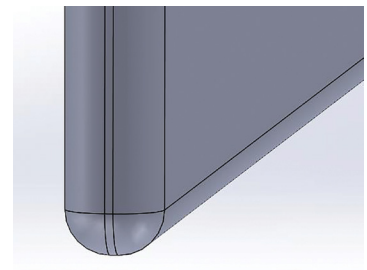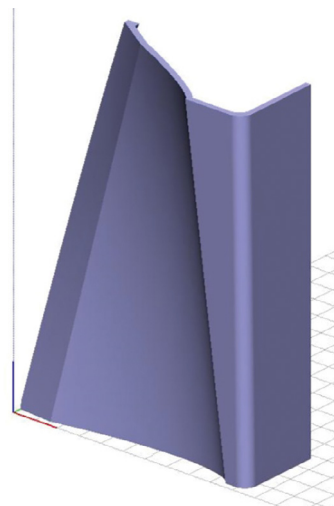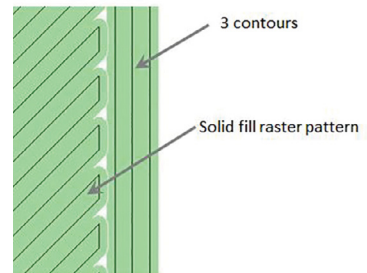

Figure 3-13: Vertical build orientation for the tool.

Figure 3-14: Cross-section of a solid fill toolpath.

Male Sparse-Style Tool Design

The general process for designing a sparse tool is similar to that of a shell tool, with the primary exception being that the tool cavity (indicated by the arrow in Figure 3-15) will be filled with a sparse build construction (the fill density or raster spacing and fill style can vary based on the consolidation pressure requirements of the application; see table 3-1 for general guidelines). Rounding of sharp corners and edges is still recommended, particularly if the tool will be envelope bagged. It is also recommended that sparse style tools be designed with a ventilation path for air within the tool to escape as it is heated and expands during elevated-temperature curing. This can be accomplished in a variety of ways, from leaving an end open to designing small vent holes.
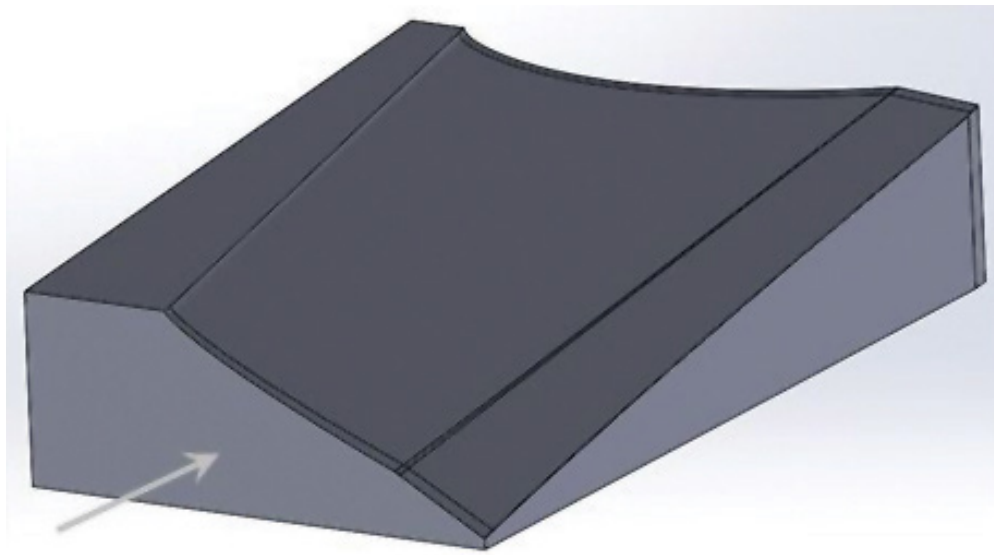
Figure 3-15: Sparse style tool (cavity indicated by the arrow).

It is still recommended to build the tool in this example in the vertical orientation shown in Figure 3-16 to produce the best surface finish (minimize stair-stepping). The design should also use a minimum of three contours (as shown in Figure 3-17). Note that structural integrity testing is based on a wall thickness of 0.1 inch. The interior of the tool should use a hexagram or sparse double dense fill pattern. These fill patterns provide a balance among performance, build time and material use. The cell size can vary, based on the application and amount of consolidation force to which the tool will be subjected, as previously detailed in Table 3-1.
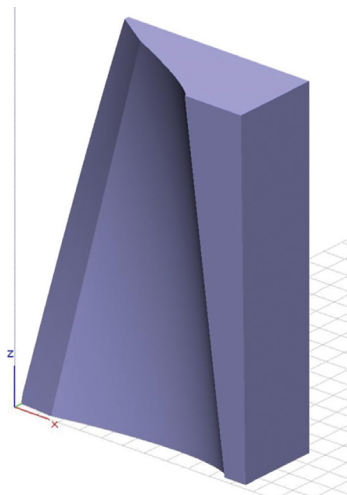

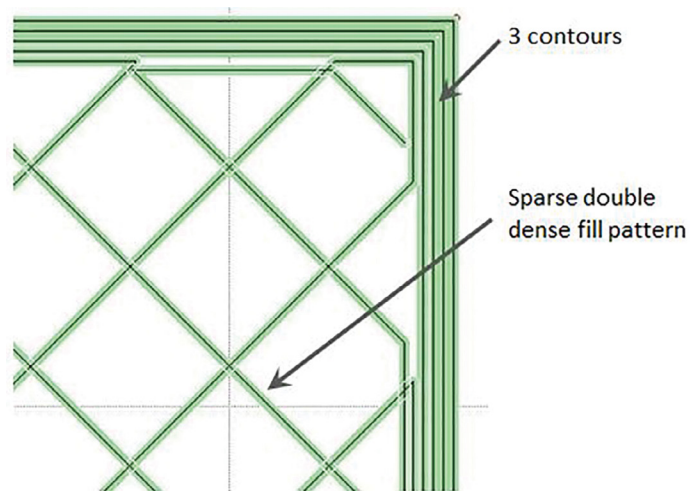Figure 3-16: Recommended vertical build orientation for the tool.

Figure 3-17: Top view of sparse double-dense fill toolpath.

FDM FOR COMPOSITE TOOLING 2.0

Deep-Draft Female Shell-Style Tool Design

Although the relatively high CTE of FDM materials is an even more significant consideration for female tools (due to the possibility of locking the part in the tool after cure), particularly those with deep walls and minimal drafts, it remains feasible to effectively use 3D printed female tools.

The process for designing a female shell tool is similar to a male tool and begins with the desired part geometry and EOP definition.
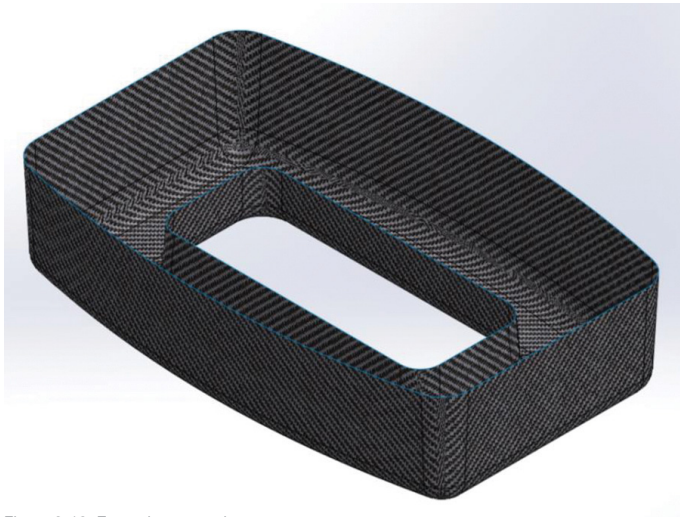
Figure 3-18: Example composite part.

The use of draft angles and flanges will aid part extraction and should be incorporated in the design whenever possible.

1. Thicken the tool surface. A thickness of 0.3 inches (7.6 mm) is recommended. Also, extend the tool lay-up surfaces beyond the EOP to provide space for excess material and bagging.
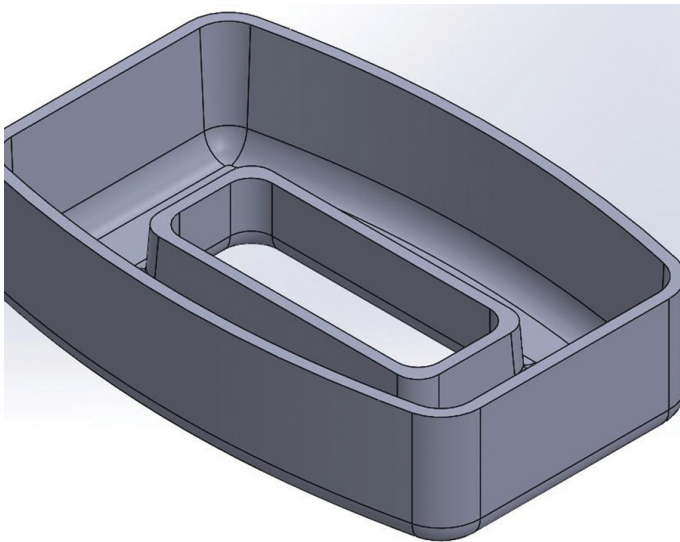
Figure 3-19: Thickening the tool surface.

2. If necessary, incorporate flanges to provide assistance in part de-molding (i.e., flanges will provide an area outside the EOP to grip or apply leverage in removing the part from the tool).
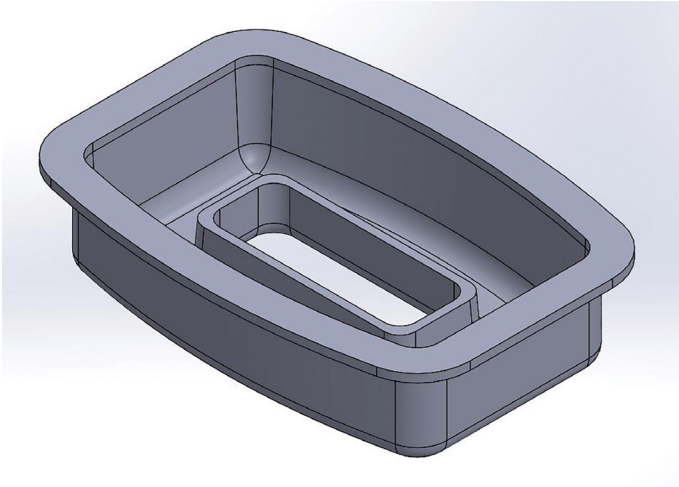

Figure 3-20: Addition of flanges to the top tool surfaces.

3. Remove sharp edges and corners by adding 0.1-0.5 inch (2.5-12.7 mm) fillets or chamfers around the tool.
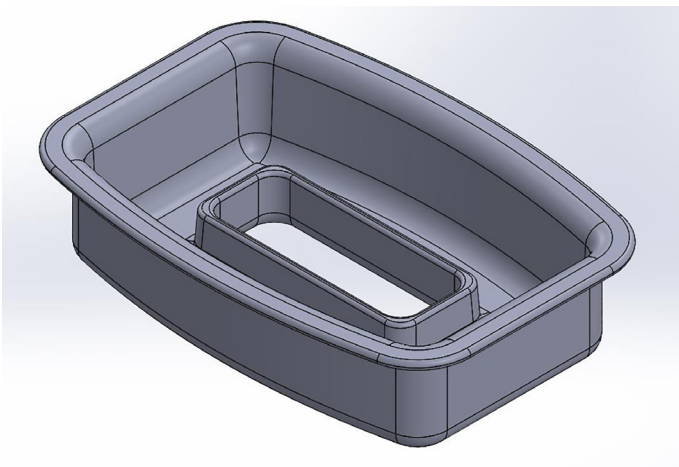

Figure 3-21: Rounding of tool corners and edges.

FDM FOR COMPOSITE TOOLING 2.0

Similar to a male shell tool, the female tool also has a solid fill raster pattern with a minimum of three contours. This particular example tool design is recommended to be printed in a flat orientation, even though there will be some degree of stair stepping along the radii. This orientation is used because it minimizes support material use and build time, providing the best compromise of build considerations (i.e., surface finish, material consumption, and build time).
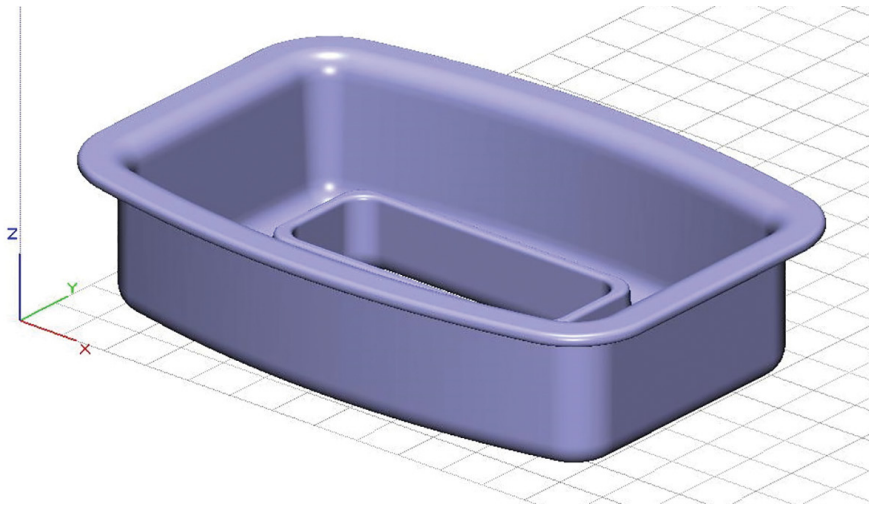
Figure 3-22: Recommended build orientation for the tool.

Although there is an appreciable amount of support material required for the flange overhang in this orientation, it is still less than the support required in other potential orientations. The image below shows a cross-section of the tool with red arrows indicating areas that will have the most pronounced stair-stepping and blue arrows indicating areas that require support material. Alternatively, 45-degree chamfers could be added to the exterior of the flanges to eliminate the need for support material, Figure 3-24.
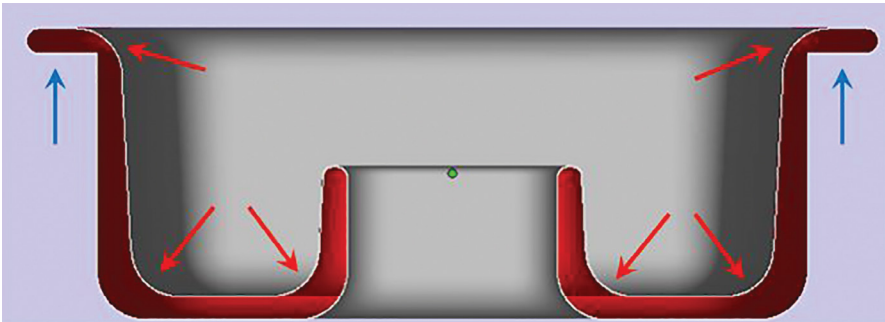
Figure 3-23: Cross-sectional view of tool design with red arrows indicating areas of anticipated stair-stepping and blue arrows indicating areas with required support material.
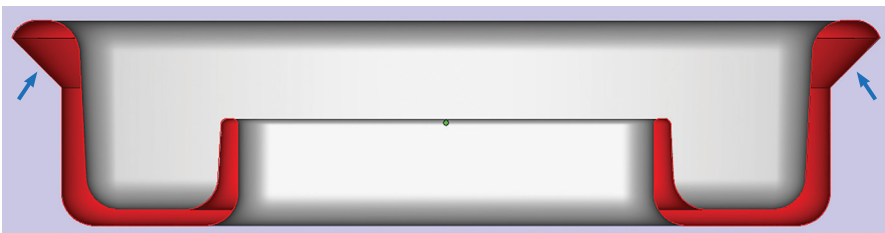
Figure 3-24: Cross-sectional view of the tool design incorporating self-supporting angles to eliminate the need for support material. Blue arrows indicate self-supporting angle.

FDM FOR COMPOSITE TOOLING 2.0

Female Sparse-Style Tool Design

A deep draft female tool with a sparse design most closely resembles the design for a conventional metallic tool. This type of design is generally not preferred as it requires more material than a shell female tool and offers no significant advantages. Such a design will be very rigid, but the shell style equivalent also provides plenty of inherent rigidity. However, for a simpler part cross-section, such as the U-shape of a wing leading edge, using a sparse style (or other stiffening features) may be required to improve tool rigidity and the approach is described below.

Figure 3-25: Sparse style female tool.

1. Begin by extruding the layup surface in the direction away from the surface that defines the composite part. Remove sharp edges and corners by adding 0.1-0.5 inch (2.5-12.7 mm) fillets or chamfers around the tool (Figure 3-25).

2. Process the file using Insight and use a sparse double dense or hexagram fill pattern with at least 3 contours. The fill density (raster spacing) should follow the guidelines provided in Table 3-1 (when envelope bagging; not required when surface bagging) (Figure 3-26).

Modification of Existing Tool Designs for FDM

In many cases, an existing tool design (intended for machining) will be available and will be considered for 3D printing. Although this typically does not provide a design that is optimal for the FDM process, such designs can be modified with relative ease to better suit FDM. For such an approach, trim/scribe lines (and similar fine features) should be removed as they do not typically print well and are likely to be removed during sanding and sealing. See the section on "Scribe Lines" later in this chapter for further detail. Alternatively, a separate trim tool (or similar processing aids) can be printed to accurately trim laminates to the proper EOP definition. Additionally, holes, pins and thermocouple ports, as shown in Figure 3-27, should be removed or redesigned (as described herein) due to the additional support material required to produce them, as well as the fact that such FDM features typically have significantly different designs.
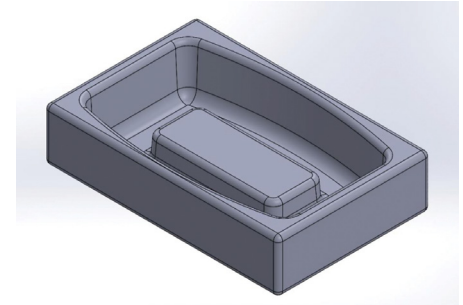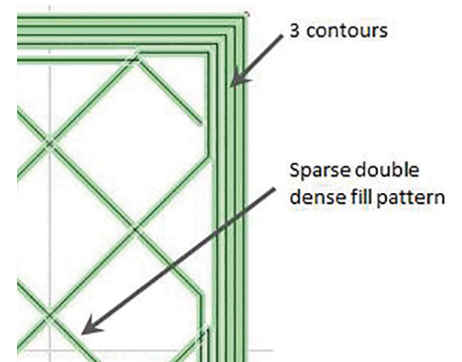

Figure 3-26: Top view of sparse double-dense fill toolpath.
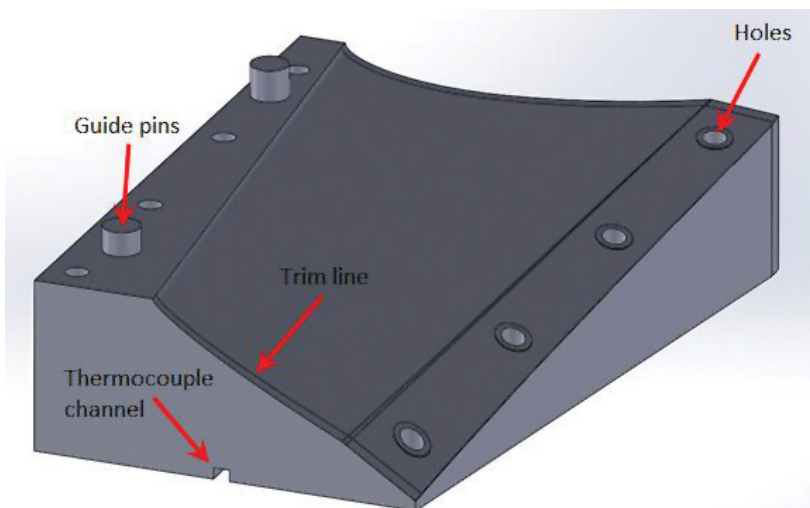

Figure 3-27: Tool features that typically must be removed or redesigned for FDM.

FDM FOR COMPOSITE TOOLING 2.0

—

In many cases, existing tool designs contain excess material that can be removed, which will reduce material consumption, build time and cost. The arrows in Figure 3-28 below show areas where excess material can be removed from the tool without harming performance.
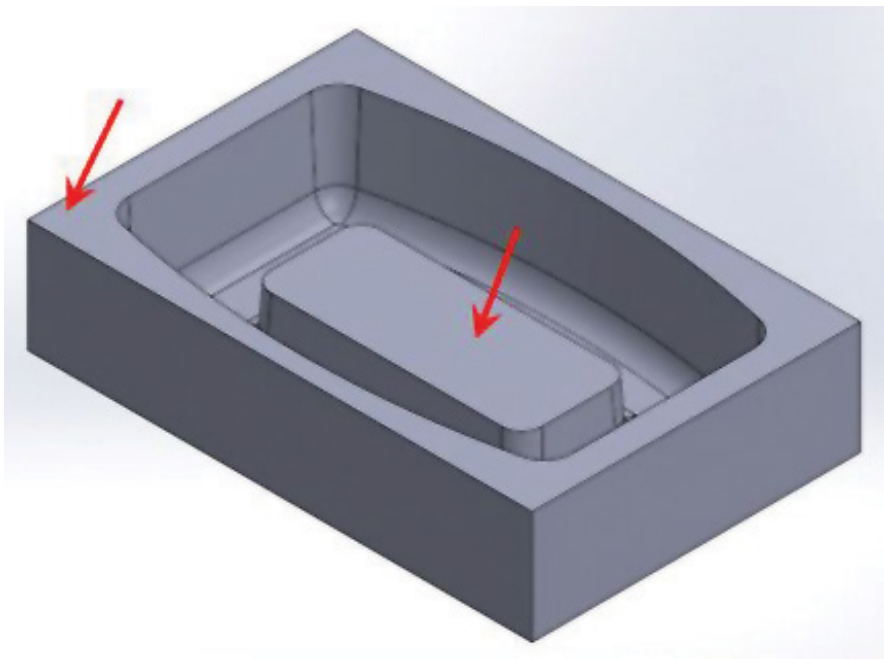

Figure 3-28: Excess material from a conventional tool design that can be removed.

Scribe Lines

Nearly all composite parts are produced oversized and trimmed to their proper dimension with a secondary process. To aid in this process, fine grooves or "scribe lines" are added into traditional composite tools to mark off on the composite part and provide a feature to follow or pick up in secondary operations. Scribe lines of typical size (0.005 - 0.010 inch, 0.127 - 0.254 mm) do not reliably show up in FDM tooling regardless of current slice height offerings. **Therefore, they are not recommended**. In lieu of printed scribes, one may consider using a printed trim fixture (examples and recommendations are shown in the section below) or print a guide piece and manually scribe the tool following sealing operations.

A simple study of scribe line geometries was performed at various slice heights and print orientations to determine general guidelines for users. A geometry was created with rectangular, V, and semi-circular scribe lines at widths of 0.020 inch (0.508 mm), 0.030 inch (0.762 mm), 0.040 inch (1.016 mm), 0.060 inch (1.524 mm), and 0.080 inch (2.032 mm), Figure 3-29. Figures 3-29 and 3-30 show various print orientations for a slice height of 0.020 inch (0.508 mm) (T40 tip) and a slice height of 0.007 inch (0.178 mm) (T14 tip). The images show that scribe lines are highly orientation- and slice height-dependent. Finer slice heights improve scribe line resolution. However, even at the 0.007 inch (0.178 mm) slice, the scribe is orientation-dependent and must be much larger (0.030 - 0.040 inch, 0.762 – 1.016 mm) than traditional scribe geometries.
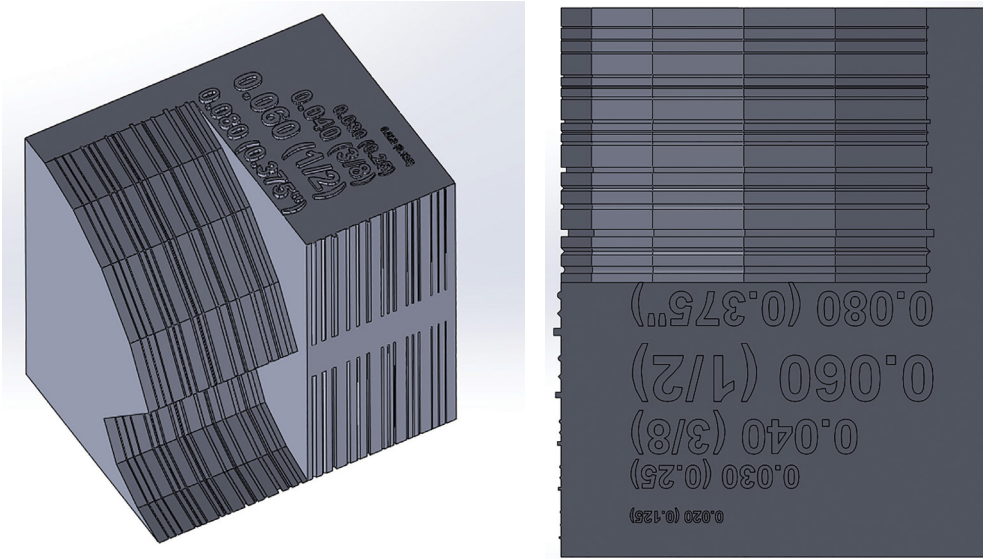
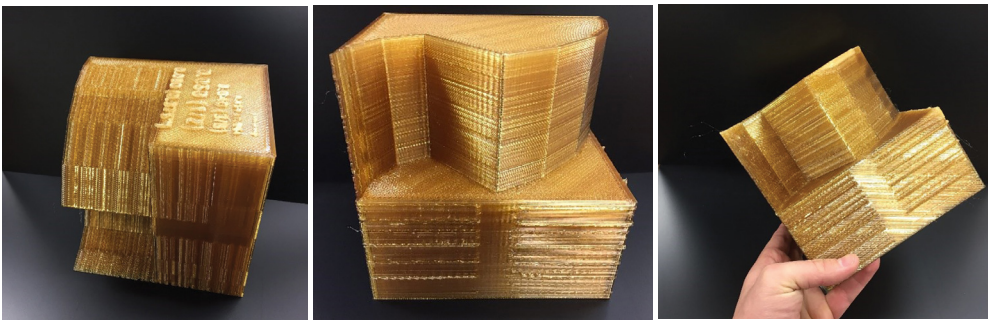Figure 3-29: Views of the scribe line test piece.

Figure 3-30: Views of the test piece printed with a 0.020 inch (0.508 mm) slice height in various orientations (vertical, horizontal, 45°).
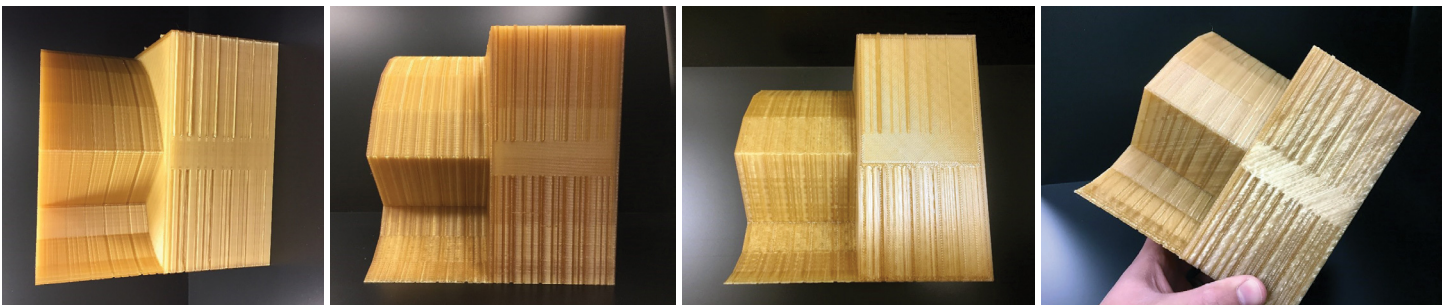

Figure 3-31: Views of the test piece printed with a 0.007 inch (0.178mm) slice height in various orientations (vertical, horizontal, 45°).

FDM FOR COMPOSITE TOOLING 2.0

Design of Trim Tools, Drill Guides and Similar Ancillary Tooling

Up to this point, the focus of this design guide has been has been composite layup tooling, but FDM technology has also proven robust for the creation of ancillary tooling. Tooling such as trim and drill fixtures, assembly aids, inspection fixtures, and other similar tooling can generally be designed to be printed quickly and inexpensively, and unlike layup tooling, can usually be printed with any material and do not require sanding or sealing. It is also possible to incorporate hardware like bushings and inserts when needed. Examples are provided in the following procedures.

1. Begin by offsetting the tool surface by 0.005 inch (0.127 mm) to provide adequate clearance and improve usability.
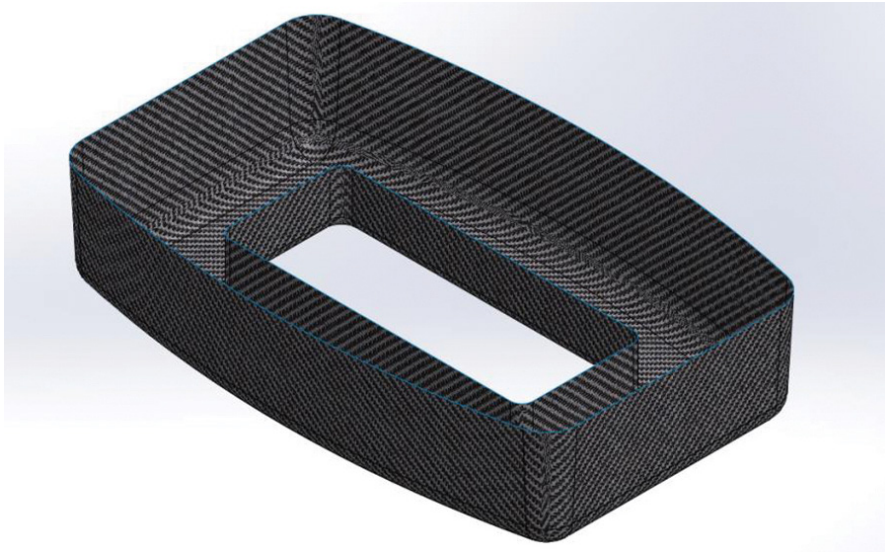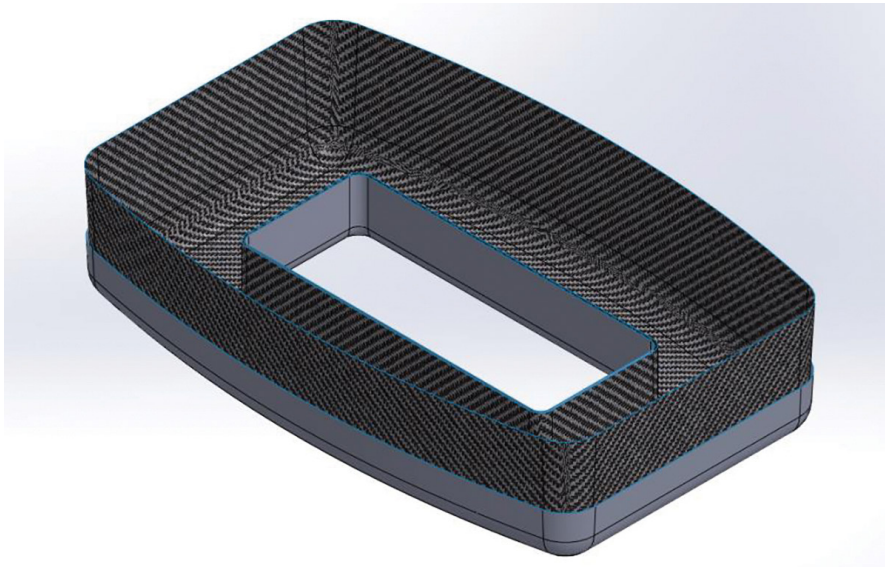

Figure 3-32: Example composite part.

Figure 3-33: Offsetting the trim tool surface from the composite part.

—

2. Thicken the resulting surface by 0.2 inch (5.1 mm) (recommended) and enclose to form a solid.
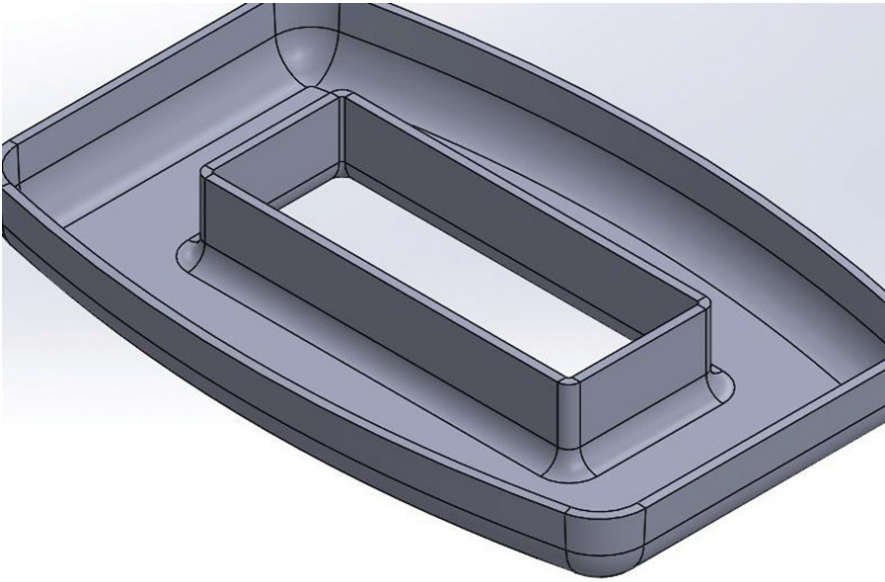
Figure 3-34: Trim-tool surface thickened.

3. Drill guides can be created as a separate tool or, in many cases, incorporated into the trim tool as shown in Figure 3-35. The tools can be printed with a single contour and solid raster fill pattern.
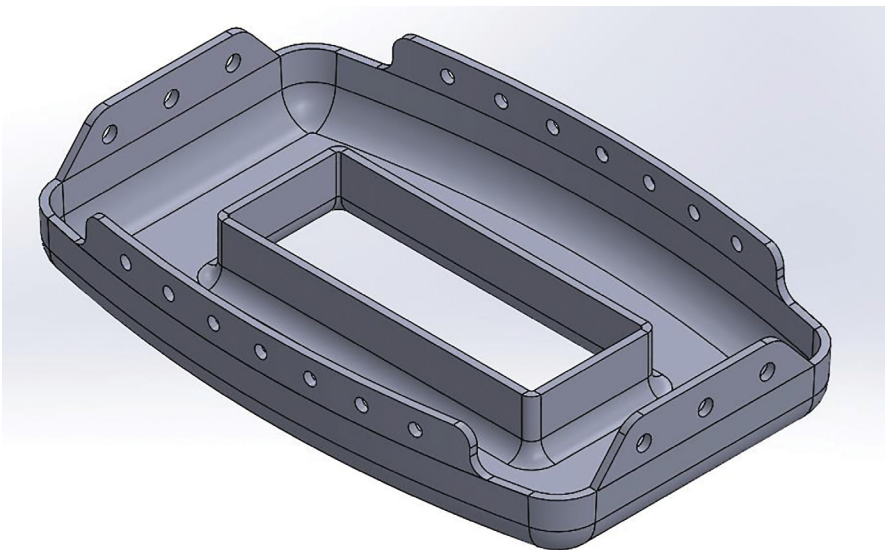
Figure 3-35: Trim profile with drill-guide features added.

Internal Feature Design and Modification

Internal features, such as holes or thermocouple ports within a tool, should incorporate self-supporting angles (>45° from the build platen) into the tool design whenever possible to eliminate the need for additional support material. The examples in Figure 3-36 and 3-37 show how circular and rectangular holes and cavities require support material, whereas shapes that use self-supporting angles do not.
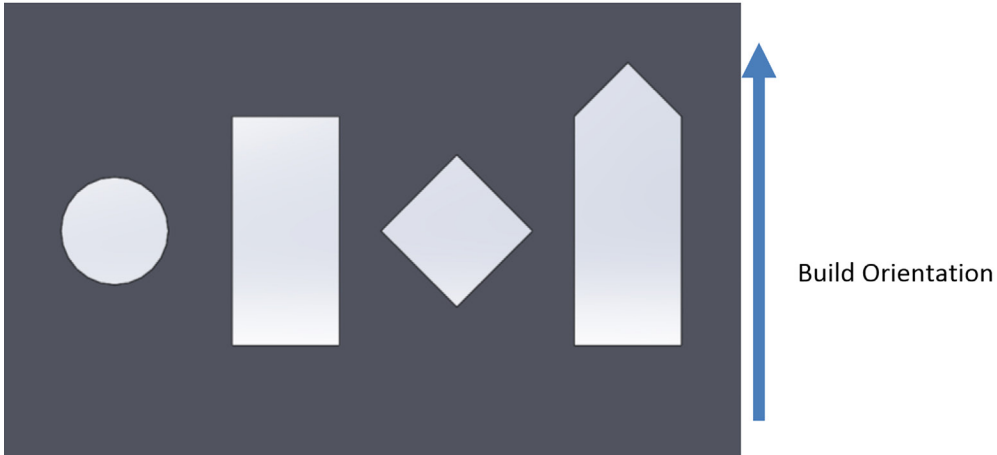

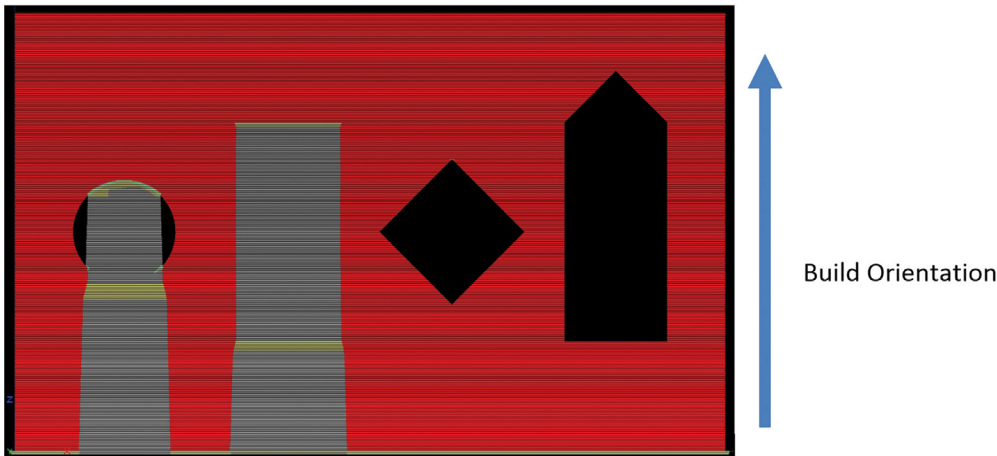Figure 3-36: Example internal features for an FDM part.

Figure 3-37: Example internal features with support material shown where required.

FDM FOR COMPOSITE TOOLING 2.0

In addition to requiring support material, horizontal holes (relative to the X-Y build plane) will have poor resolution due to stair-stepping. The best method to address this is to design and print an undersized diamond-shaped pilot hole that is drilled or reamed, in a secondary operation, to the final dimensions.

External Feature Design and Modification

External features such as guide pins require the same incorporation of self-supporting build angles as described for internal features, but for both horizontal axes (refer to Figure 3-39). The example in Figures 3-40 and 3-41 shows various shapes and their corresponding amounts of required support material.

By adding a self-supporting angle to both the X and Y axes as shown in Figure 3-41, a pin can be produced with no support material.
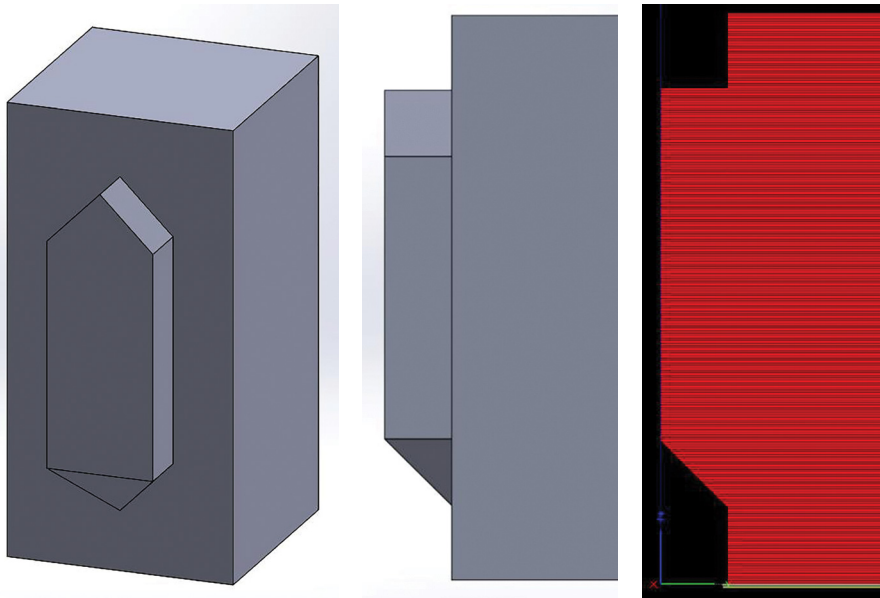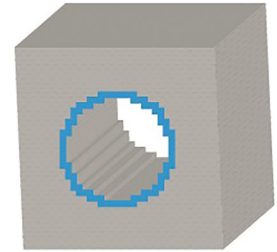

Figure 3-38: Resolution of a horizontally-oriented hole after printing (reaming recommended).
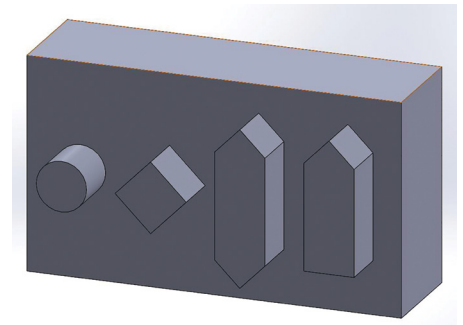
Figure 3-39: Example external features (typically used for guides or pins).
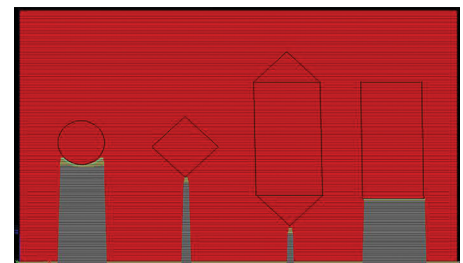
Figure 3-40: Example external features with support material added.

Figure 3-41: Guide feature designed with self-supporting angles in both horizontal axes, resulting in no required support material.

Localized Reinforcement or Variable Density (Inserts, Machining, Bushings)

In many tool designs there will be a desire to incorporate localized reinforcement or regions of variable density. The two most common needs for localized reinforcement are added material around bushings/inserts and skim-coat machining for near-net-shape tooling. The following sections will discuss methods to incorporate localized reinforcement for machining and incorporating inserts.

Design Recommendations for Near-Net-Shape Tooling

If the tolerance of the FDM printer exceeds requirements for the desired composite tool, users can print the layup tool slightly oversized (with additional material on

the layup surface) and skim-coat machine the tool to the final dimension. An added thickness of 0.04-0.12 inch (1.02-3.05 mm) is recommended for the machining face. Please note that thicker machining faces use more material which will increase print time, cost, and waste.

Preliminary investigations have shown that ULTEM 1010 resin can be machined through a broad range of cutting rates. Therefore, it is highly recommended to print test coupons to dial-in feed and speed rates for your specific needs and mill setup. Feed and speed rates listed by common industry references as well as two customer examples are listed below:

Common Industry Reference: ¾ inch high-speed steel end mill with a cutting depth of 0.25 inch and speed of 3,240-5,400 in/min.

Customer 1: 2 inch facing mill with a spindle speed of 900-950 rpm and a feed rate of roughly 100 in/min for simple facing operations.

Customer 2: 3/4 inch end mill with a spindle speed of 12,500 rpm and a feed rate of 433 in/min for their initial coarse passes and a 1/2 inch ballnose mill with a spindle speed of 12,500 rpm and a feed rate of 244 in/min for finishing passes.

There are two primary ways to incorporate additional material for machining or localized reinforcement:

Single Body - The tool can be designed as a single solid body with the added machining thickness incorporated, Figure 3-42. Although this method is quicker and easier from a design and file preparation standpoint, it can result in significantly more material usage. This is because the amount of solid material needed at the machining face (an additional 2-3 contours for a slice height of 0.020 inch, 0.508 mm) will be added around the entire outer surface of the tool.
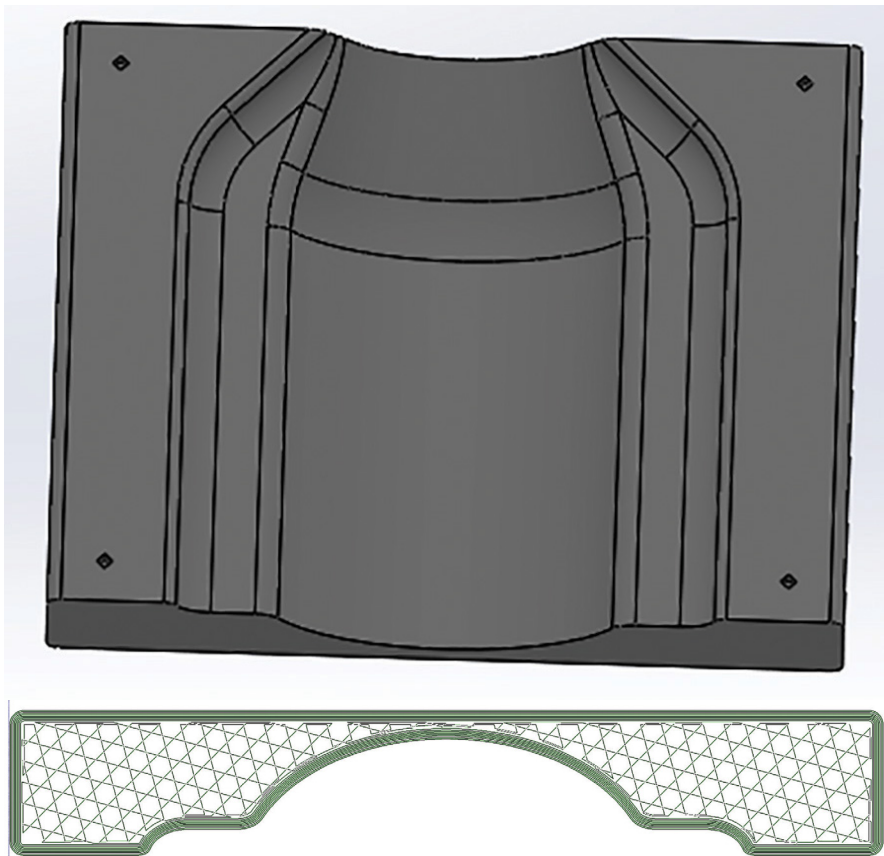

Figure 3-42: Rendering of tooling design (top) with a view of the insight tool paths showing the added thickness for the machining surface (bottom).

stratasys

—

Multi-Body (Recommended Method) – Alternatively, the tool can be designed to the final specification with the added machining face as a separate body, Figure 3-43. Although it is more complex from a workflow perspective (design and file preparation), the multi-body method will typically save significant amounts of material. This is because the added material thickness is only applied where it is needed rather than the entire outer surface of the tool.
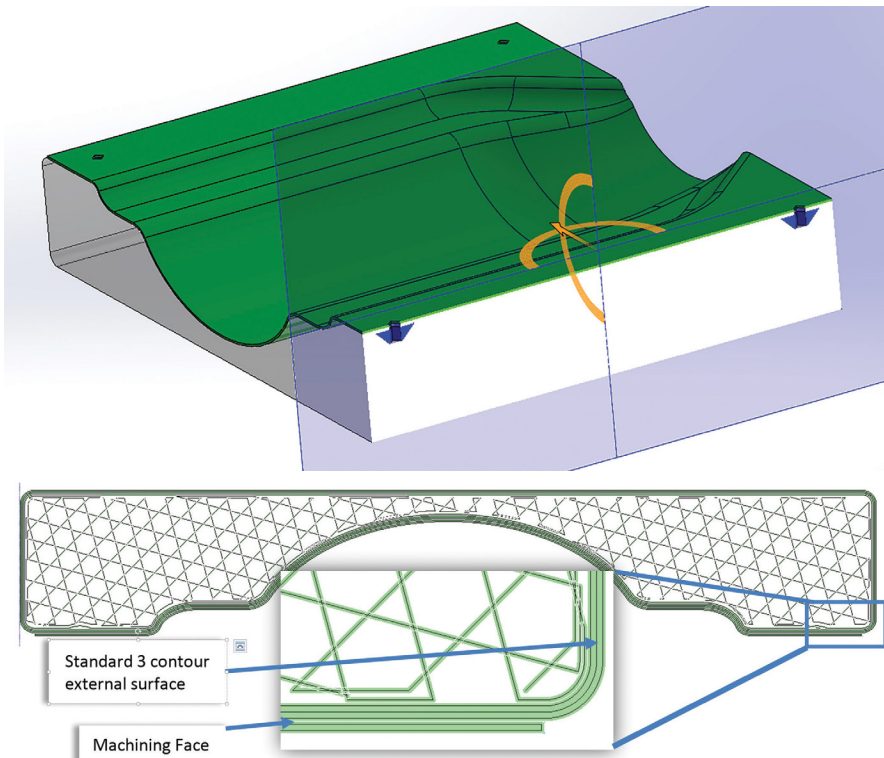

Figure 3-43: Rendering of tooling design showing multiple bodies for the tool body, machining face, and bushing plugs (top) with a view of the insight tool paths showing the added thickness at the machining surface only (bottom).

Multi-body Design and File Preparation Example: Generic Fairing

This generic fairing example will show the basic steps for designing and preparing multi-body composite tooling files for printing. The example begins with the basic fairing tool that has been optimized for additive manufacturing, utilizing the principles and techniques discussed in the previous sections, Figure 3-44. The tool will be designed for skim-coat machining of the tool surface and also incorporate plugs for tooling ball bushings.
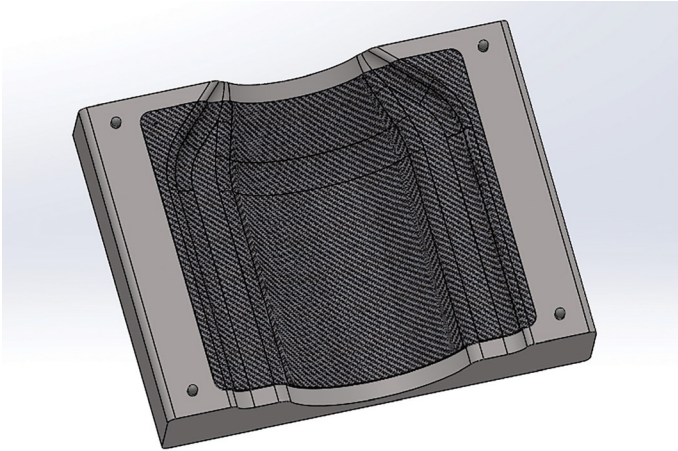
FDM FOR COMPOSITE TOOLING 2.0

Figure 3-44: CAD rendering of a generic fairing tool design.

The first step is to create individual solid bodies for the machining face, each bushing and/or any other localized reinforcement or variable density regions that are desired within the tool.

Machining Face Design: Depending on the CAD system being used, the process may vary, but the basic workflow using SolidWorks is as follows.

1. Offset the critical surface (the area to be machined) the desired machining thickness.

    a. Note: Initial testing has revealed that a machining face thicknesses between 0.04 inch (1.02 mm) and 0.12 inch (3.05 mm) is typically sufficient.

2. Thicken the offset surface back toward the tool the same distance, making sure not to merge entities.

3. Create a reference geometry for proper alignment of all tool sections in the Insight slicing software.

    a. Note: Make the reference geometry slightly taller than the tool in the print orientation. Figure 3-45 shows a simple extruded rectangle as a reference geometry.
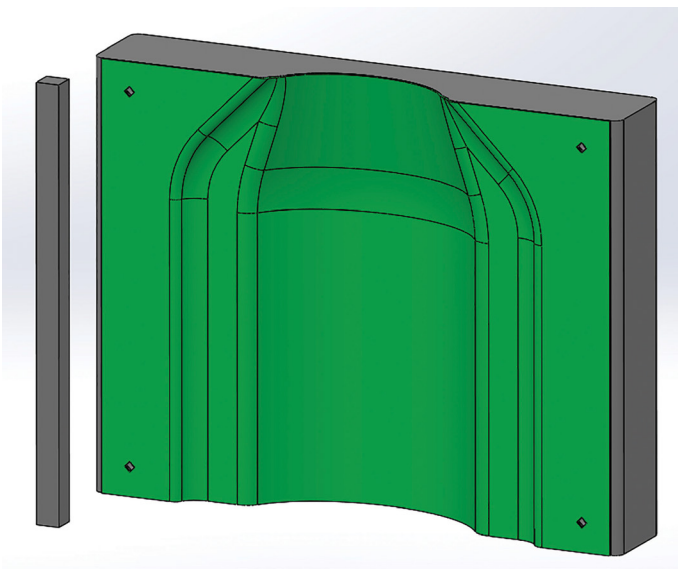

Figure 3-45: View of the tool body with a separate machining face and reference body.

FDM FOR COMPOSITE TOOLING 2.0

—

Localize Bushing Reinforcement Design (Variable Density): CAD system and design methods/preferences will dictate the design workflow, but the end result should be a cavity in the bulk tool with a corresponding plug, as depicted in Figure 3-46. Although not required, recessing the plug and cavity the thickness of the number of contours used for the tool surface (0.12 inch (3.05 mm) which is three contours for a slice height of 0.020 inch (0.508 mm) is recommended. This will hide the seam created at the junction of the two STL files and result in better surface finish. Additionally, chamfering the internal-most face of the cavity and plug is recommended to maximize printability, Figure 3-46.
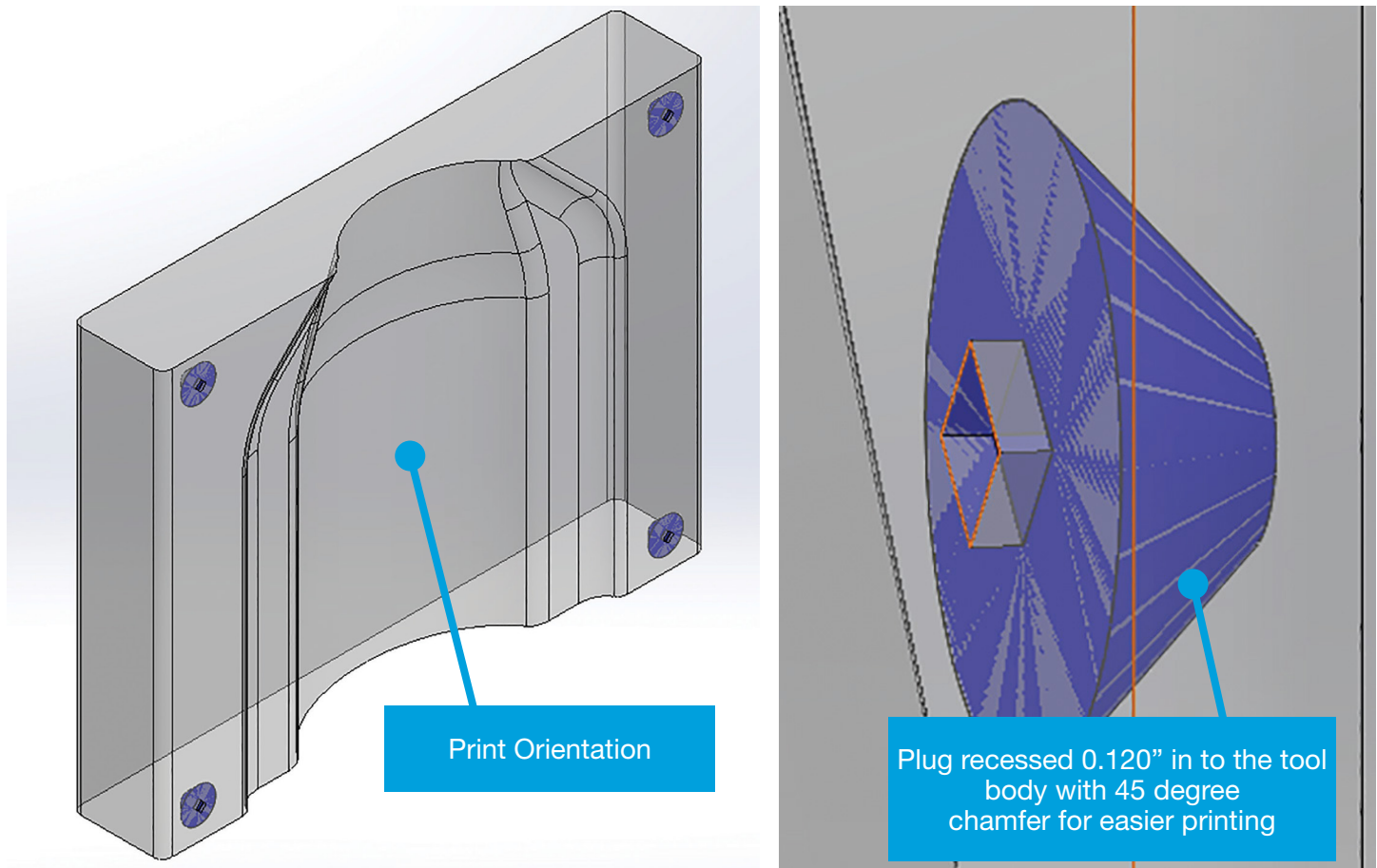

Print Orientation

Plug recessed 0.120" in to the tool body with 45 degree chamfer for easier printing

Figure 3-46 – Showing plug design recessed below the tool surface with 45 degree chamfers for easy of printing.

—

Once the separate solid bodies have been created, export each solid body that will have different build parameters (infill pattern, density, etc.) along with the reference geometry as a separate STL file. In this case, three separate STL files will be generated: the bulk tool and reference geometry; the machining face and reference geometry; and the bushing plugs and reference geometry.

Once the STL files are created, prepare and slice each STL separately in Insight. Each STL should be imported and oriented with the reference body placed in the same location, Figure 3-47. Slice the STLs and group layers according to the needs of that tool section and save it as a job, Figure 3-48. The list below has basic build parameter recommendations for different regions of the tool.

- Bushing and insert plugs should be built solid

- Machining faces should be built solid, preferably with contours and no raster's

- The tool body should have the desired fill pattern and a minimum of 3 contours

With all the files prepared, open up the main tool body file in Insight and add the machining face slice to the tool by going to the slice tab and selecting **Combine slice curve files**, Figure 3-49.

In the **Combine slice curves** pane (right side of the Insight program), browse to the desired slice file and be sure to align the new file to the origin (0,0,0), Figure 3-50. Select ok and accept any warnings about overlapping curves. Repeat this process for the bushing plug slice files and any other slice files that need to be added to the tool assembly.

With all the slice curves combined into a single slice file, delete the curves that define the reference geometry. Examine the slice layers to ensure everything is aligned and grouped properly. Add anchor pins if necessary.

NOTE It is highly recommended to use anchor pins with all ULTEM 1010 resin tooling builds.

Generate support and tool paths. Examine the generated tool paths for any anomalies, misalignments, etc. Figure 3-51 shows how the face sheet, bulk tool and bushing plug tool paths look when properly sliced. There should be no gap between the bodies as this will result in a reduction in properties. If everything is satisfactory, save the file as a job under a new name.
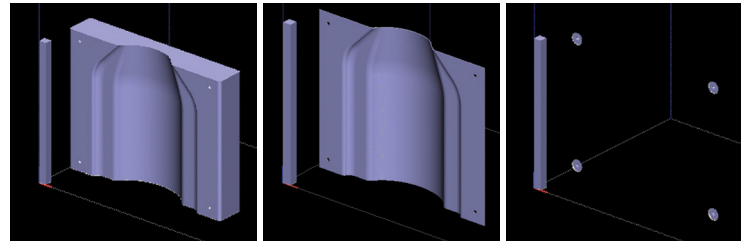

Figure 3-47: Views of the tool body, machining face, and bushing STL files oriented with the reference body at the origin in Insight.
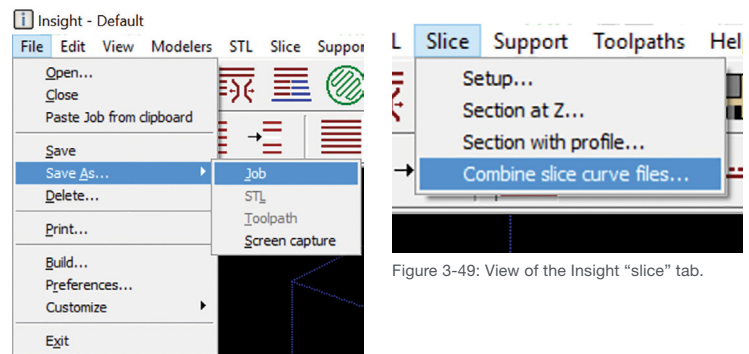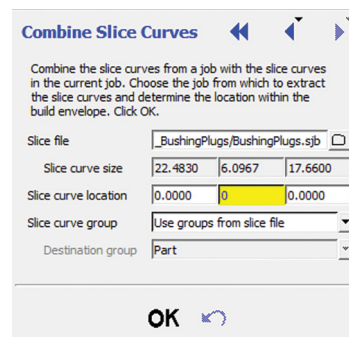

Figure 3-48: Saving files as a job.

Figure 3-49: View of the Insight "slice" tab.

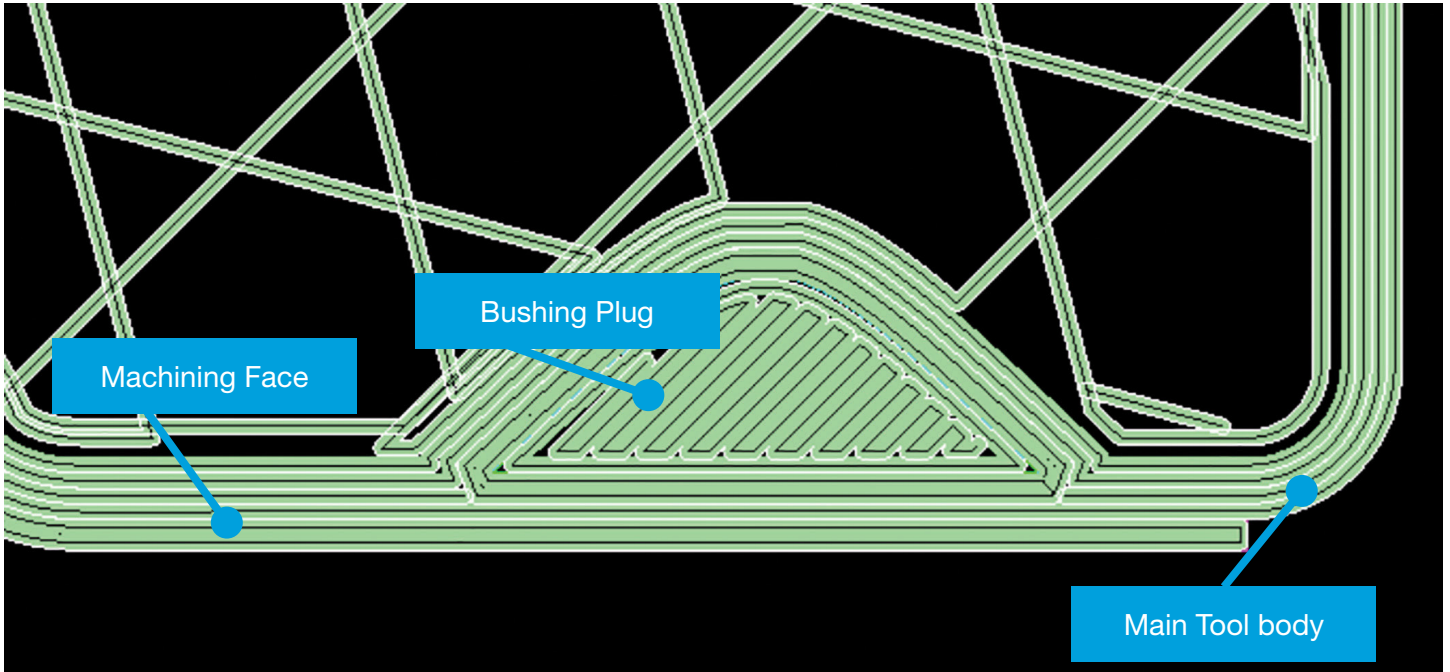Figures 3-50: Aligning the slice curves to the origin.

Figure 3-51: View of the Insight file showing the bushing plug, tool body, and machining face all perfectly aligned with no gaps.

Bushing/Insert Accuracy and Best Practice

Composite tooling often incorporates a variety of inserts for tooling balls, index tabs/pins, clamps, etc. Metal inserts and hardware can be inserted into FDM parts and tooling with a variety of methods, such as heat staking, press fitting, and potting. For best results with inserting hardware, utilize the Stratasys Inserting Hardware Post-Build best practice guide.

A study was performed in partnership with a leading business jet OEM to determine cycle-to-cycle accuracy of metal bushings. A simple rectangular geometry was created with varying thicknesses. Inserts were heat staked into the geometry and a FARO arm scanner was used to measure the location of the bushings before and after eight thermal cycles at 250 °F (+/- 10) (121 °C +/- 5.5). Figure 3-52 shows an image of the geometry and Table 3-3 shows the measurement data. This simple study shows bushing locations remained within 0.007 inch (0.20 mm) of their initial location after eight cycles. The direction that showed the largest dimensional change (Y-direction) was the direction of the longest tool path (the direction of printing within a single layer). The X direction in Figure 3-52 is the direction of layer deposition, also known as the build orientation or the Z-axis in the printer.
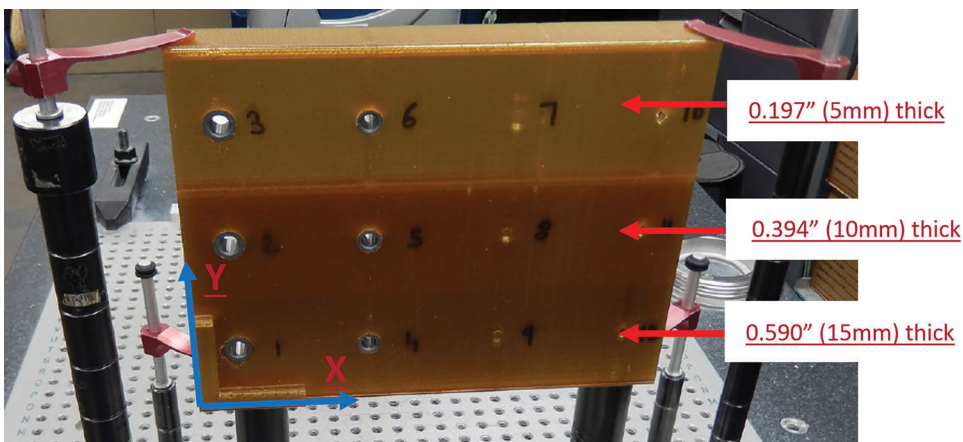

Figure 3-52: Image of the geometry before thermal cycling. The bottom left corner is measurement origin.

FDM FOR COMPOSITE TOOLING 2.0

Standard accuracy measurements for the bushing locations before and after cycling

BUSHING	SAMPLE THICKNESS	NO CURING		AFTER 08 CYCLES (250 °F +/-10)		DIFFERENCES	
	Inches	X Dev	Y Dev	X Dev	Y Dev	X Dev	Y Dev
Ø 8mm Bushing #1	0.197	-0.0174	0.0210	-0.0175	0.0144	0.0001	0.0066
Ø 8mm Bushing #2	0.394	-0.0137	0.0006	-0.0136	-0.0062	-0.0001	0.0067
Ø 8mm Bushing #3	0.591	0.0059	-0.0237	0.0066	-0.0307	-0.0007	0.0070
Ø 8mm Bushing #4	0.197	-0.0110	-0.0057	-0.0118	-0.0126	0.0008	0.0069
Ø 8mm Bushing #5	0.394	-0.0014	-0.0097	-0.0026	-0.0163	0.0011	0.0066
Ø 8mm Bushing #6	0.591	0.0048	0.0162	0.0046	0.0094	0.0002	0.0068

Table 3-3A

Metric accuracy measurements for the bushing locations before and after cycling

BUSHING	SAMPLE THICKNESS	NO CURING		AFTER 08 CYCLES (121 °C +/-5.̄5)		DIFFERENCES	
	Millimeters	X Dev	Y Dev	X Dev	Y Dev	X Dev	Y Dev
Ø 8mm Bushing #1	5.0	-0.4430	0.5340	-0.4450	0.3670	0.0020	0.1670
Ø 8mm Bushing #2	10.0	-0.3490	0.0140	-0.3460	-0.1570	-0.0030	0.1710
Ø 8mm Bushing #3	15.0	0.1500	-0.6020	0.1670	-0.7800	-0.0170	0.1780
Ø 8mm Bushing #4	5.0	-0.2790	-0.1440	-0.2990	-0.3190	0.0200	0.1750
Ø 8mm Bushing #5	10.0	-0.0360	-0.2470	-0.0650	-0.4150	0.0290	0.1680
Ø 8mm Bushing #6	15.0	0.1220	0.4120	0.1170	0.2400	0.0050	0.1720

Table 3-3B

Tool Segmentation and Joining

For tools larger than the build chamber of the FDM machine (e.g., the Fortus 900mc build chamber is 36 inches wide x 24 inches deep x 36 inches high [914 x 610 x 914 mm]), tool sectioning and segmentation is a viable approach. Tools can be built in sections sized to fit the build chamber and joined with secondary operations, such as thermal welding or structural bonding. To assist in assembly, joint features can be easily incorporated into the tool design to ensure proper fit and alignment. Large mechanical joints that align and geometrically lock pieces together are recommended to add structural integrity to the joint and ease the assembly process. Common assembly joints that have been successful in large tool applications include tongue and groove, dovetails, and saw-tooth patterns. Incorporating mechanically locking pieces into joint designs can help aid assembly and add mechanical strength. Butt joints are NOT recommended due to their poor mechanical strength and the lack of self-alignment. Examples of tools requiring segmentation are shown in Figures 3-53 and 3-54; additional information for these specific cases is provided in Section 6.

For joining tool segments, bonding can be performed with compatible structural adhesives with the appropriate capability to withstand cure temperatures. Two part epoxy paste adhesive Hysol EA 9394 has been successfully demonstrated although many other materials may also work effectively.

Alternatively, thermal welding methods can be used. Similar to metal welding, thermal welding involves fusing components using melted thermoplastic material. Although this process tends to require experienced operators to properly execute, the advantage is that the resulting joint will perform in a nearly identical manner (mechanically and thermally) as the surrounding structure, given that the process allows the same thermoplastic material to be used as that of the FDM parts themselves.

There are multiple types of thermal welders available. An extrusion welder will self-feed and apply a bead of material at the joint interface, whereas a hot-air welder requires manual feeding of the material. Either can be used to simultaneously melt the feed material and the joint surface to create a strong bond. An example application of a hot air welder is provided in Appendix A.
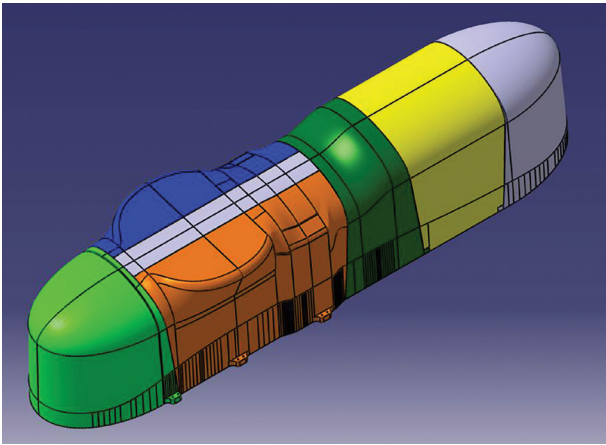

Figure 3-53: Aurora Flight Sciences fairing tool model, sectioned into seven segments (noted by the different colors).
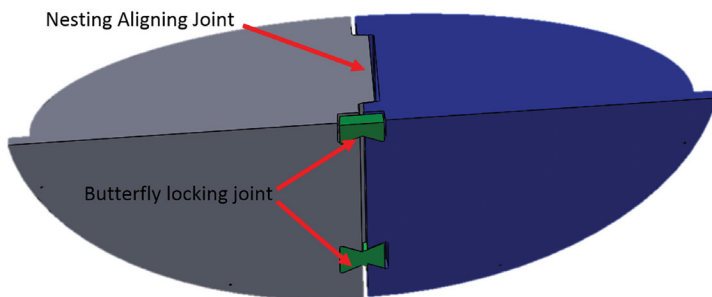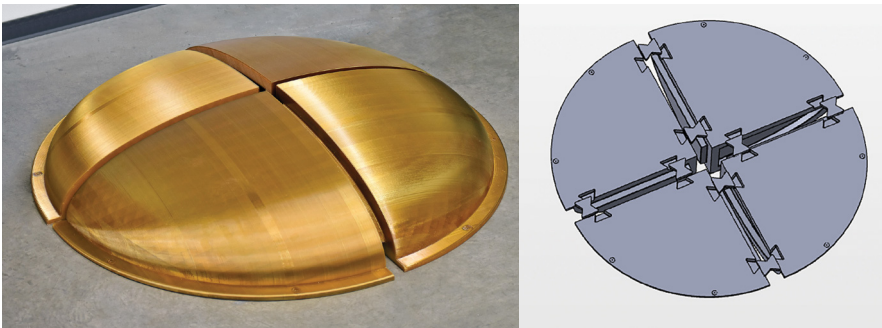

Figure 3-54: Six-foot diameter radome mold utilizing large nesting and aligning features with separate butterfly locking joints to hold sections together during bonding and assembly.

FDM FOR COMPOSITE TOOLING 2.0

—

SECTION 4 – POST-PROCESSING AND PART FABRICATION

The resulting surface roughness of an FDM tool is driven by the geometry, layer thickness and build orientation. As previously stated, as-built FDM composite tools have inherent porosity and a surface finish that is unlikely to be acceptable for producing composite parts for most applications. Post-processing the tool achieves the desired surface finish and provides vacuum integrity.

A variety of methods can be used to improve the surface roughness of the tool including manual abrasion, media blasting and tumbling, all of which have advantages and drawbacks. The current best practice to meet surface-finish requirements and provide vacuum integrity is manual abrasion followed by application of an epoxy sealer. Tools are sanded by hand using a dual-action orbital sander with progressively finer abrasive sandpaper, ranging from 120 to 800 grit. Tool sealing is accomplished with a two-part epoxy or an epoxy film adhesive, although depending on the application, other materials are also used (e.g., adhesive-backed FEP films and similar).

The most appropriate material and method for preparing and sealing FDM composite tools will be determined by the application. Additional information for the most common sealing materials used to date is provided in the following sub-sections.

Epoxy Sealers

The most common approach for sealing FDM tools is the use of epoxy sealers. These materials accommodate nearly all tool shapes and provide the required vacuum integrity for surface bagging. There are numerous epoxy materials that will work effectively. Materials should be chosen to withstand the required cure temperatures, as well as the anticipated life of the tool. They should also be evaluated for compatibility (e.g., adhesion) with the selected FDM material.

Stratasys has primarily used TC-1614 two-part epoxy from BJB Enterprises. It has a desirably low viscosity at room temperature that spreads evenly on tool surfaces while also penetrating into tool material layers. However, thermal cycling for tool life evaluations revealed that TC-1614 and other common epoxies typically oxidize and break down after roughly 30 cure cycles at 350 °F (177 °C) in an oxygenated environment. Envelope bagging and/or curing in an inert environment should significantly extend the life of tools sealed with high temperature epoxies. Evaluation of alternative resin systems and sealers better-suited to continuous exposure at 350 °F (177 °C) cure temperatures is in progress; results will be provided in subsequent design-guide releases.

The procedure for sealing tools using two-part epoxies such as TC-1614 can be found in Appendix B.

Tool sealing can also be effective using epoxy film adhesives (unsupported films are used to avoid exposing a carrier material during abrading/polishing). Again, numerous material options will work, including AF-163 and AF-555 from 3M. Alternative materials can also be considered and should be evaluated based on ability to withstand continuous exposure to the required cure temperature as well as compatibility with the selected FDM material. The primary advantages of film adhesives compared with epoxy pastes are ease of application and assurance of even coverage.

Adhesive-Backed Films

Adhesive-backed FEP (and similar) films offer an alternative to epoxy sealing materials. Aerospace OEMs have used films such as Tooltec CS5 and Toolwright 5 from Airtech for years to provide an effective lay-up and release surface to tools made from traditional materials and processes. Such films are best-suited for relatively simple and flat tool shapes with minimal and gradual complex contours since they exhibit minimal elongation. FEP films are also limited to relatively low volumes of parts before they lose effectiveness due to nicks, tears and adhesion to the tool. One application for which they are ideal is repair tooling, since they can be used without any tool sanding, and part volumes for repair tools tend to be in the single digits. Note that tools sealed with adhesive-backed films must be envelope bagged since surface bagging to such films is not effective, unless they are combined with another sealing method.

FDM FOR COMPOSITE TOOLING 2.0

Surface Finish Results

Figure 4-1 below shows the resulting surface finish for as-built FDM surfaces, tools sealed with an adhesive-backed film (Tooltec CS5) with no sanding, tools sealed with adhesive-backed film after manual sanding, and manually abraded tools sealed with an epoxy sealer. The most common surface finish requirement for composite tools is also shown for comparison (64 µin (1.6 µm), Ra). As can be seen, the common methods used to seal FDM composite tooling produces surface finishes considerably smoother than typical requirements.

SURFACE ROUGHNESS (0.01" LAYER HEIGHT)
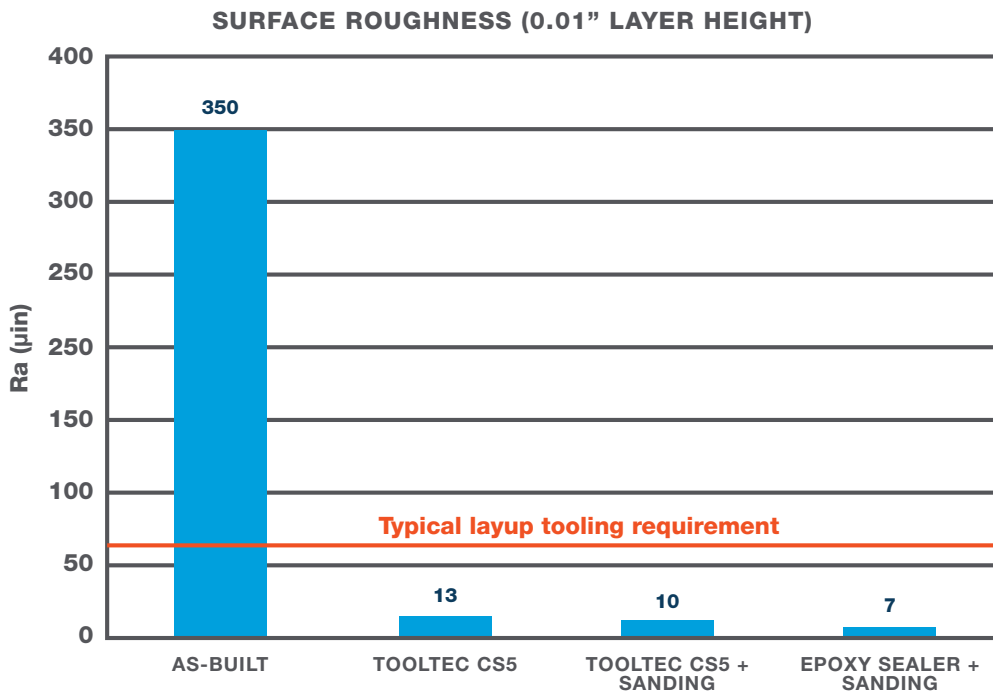
Figure 4-1: Resulting surface roughness for common FDM post-processing techniques.

SECTION 5 – TOOL LIFE AND CHARACTERIZATION DATA

A broad range of testing and characterization was performed during design guide development to validate the performance of FDM composite tooling. Testing included evaluations of solvent exposure, outgassing (to verify a lack of potential contaminants), moisture exposure, accuracy and thermal stability, and initial assessments of tool life. Summaries of the evaluations and key results are provided in the following sub-sections. All testing was performed on tools or test coupons produced in ULTEM 1010 resin.

Accuracy and Thermal Stability

To assess accuracy and stability, multiple tools were evaluated before and after thermal cycling. Three tool designs were produced and build style (shell vs. sparse) and sizes were varied for a total of five variants (refer to Figure 5-1). The tools were sent to an external inspection facility for precision 3D scanning. A Platinum FaroArm (from FARO Technologies) with an SLP 300 laser head (from Laser Design) was used. The scan data was compared with the CAD model for each tool variant using PolyWorks View 3D metrology software (from Innovmetric).

The composite tools used for this evaluation were post-processed (abraded) prior to the initial 3D imagery. This configuration was selected since nearly all FDM composite tooling will undergo such preparation, making the accuracy of a post-processed tool the most relevant data. Although there is likely some variability in post-processing between operators, the overall amount of material

removed during abrasion was found to be quite small (using standard best practices) and does not represent a significant portion of overall dimensional variation.

As stated, tools were scanned before exposure to elevated temperatures and then sent for thermal cycling. For cycling, the tools were vacuum bagged (envelope bagging scheme), heated to 350 °F (177 °C), held at temperature for two hours (minimum) under full vacuum, and then ramped back down to below 150 °F (66 °C) between cycles for a total of 10 consecutive oven cycles.

The accuracy of the 3D scanner is ± 0.0015 inch (0.0381 mm) and the accuracy of the FaroArm is also ± 0.0015 inch (0.0381 mm) for a total accuracy of ±0.003 inch (0.076 mm). This limit applies to tools that are scanned and compared to the original CAD data. For comparing tool geometry after cycling to the scan data from before cycling, the accuracies must be taken in aggregate or "stacked," resulting in accuracy limits of ± 0.006 inch (0.152 mm).
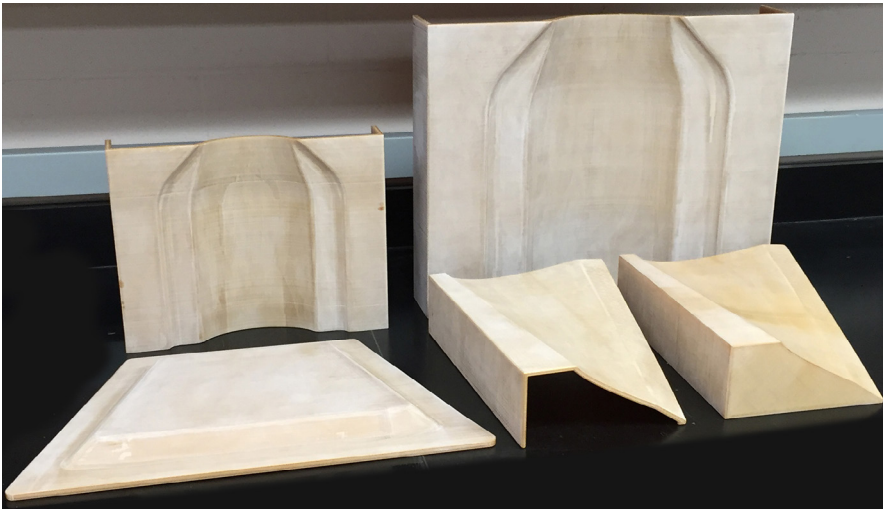
Figure 5-1: Tool designs used for thermal stability testing (shown sanded, but unsealed).

Accuracy and Thermal Stability – Results

Representative data sets from the evaluation can be seen in Figures 5-2 through 5-5 for the resulting comparison between the printed example tool and the original CAD model data (no thermal cycling). As shown in Figure 5-2, the scan data shows the shell-style tool has areas that vary from the model by as much as approximately 0.019 inch (0.482 mm) and over 92% of the tool is within ± 0.015 inch (0.381 mm). And for this particular example, the majority of the area that exceeded that value was outside the EOP. For reference, the subject tool is approximately 14.5 x 10.5 x 4 inches (368.3 x 266.7 x 101.6 mm) in size.

For the same example tool geometry as described above, but designed and printed in a sparse build style, the data shows the tool has areas that vary from the model by as much as approximately 0.018 inch (0.457 mm), as shown in Figure 5-3. For this tool, over 90% of the tool is within ± 0.015 inch (0.381 mm) and again, as can be seen, the majority of areas with greater variation is outside the EOP and concentrated on the vertical faces of the tool.
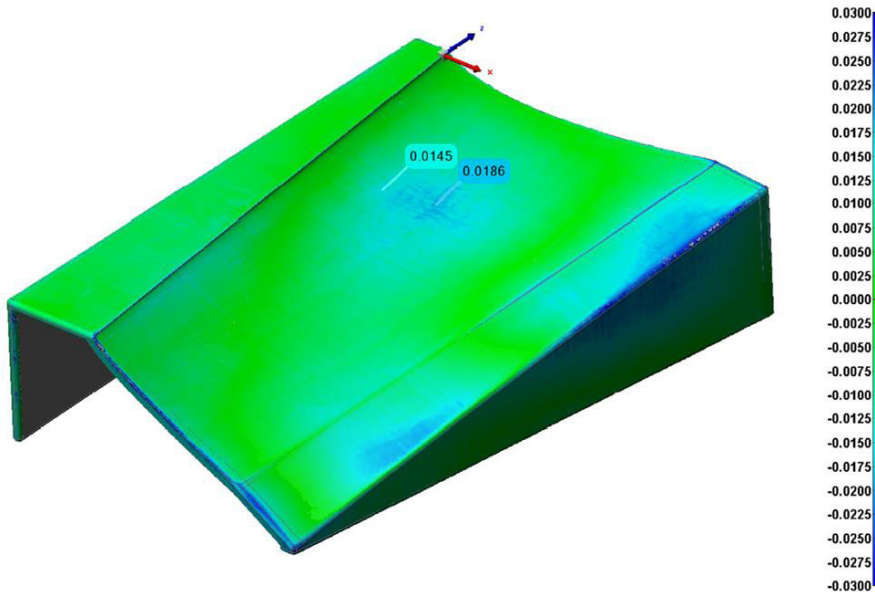
Figure 5-2: 3D scan data for a UAV fan-blade tool (shell style) with color map comparison to the original CAD model (no thermal cycling). Dimensions are in inches.
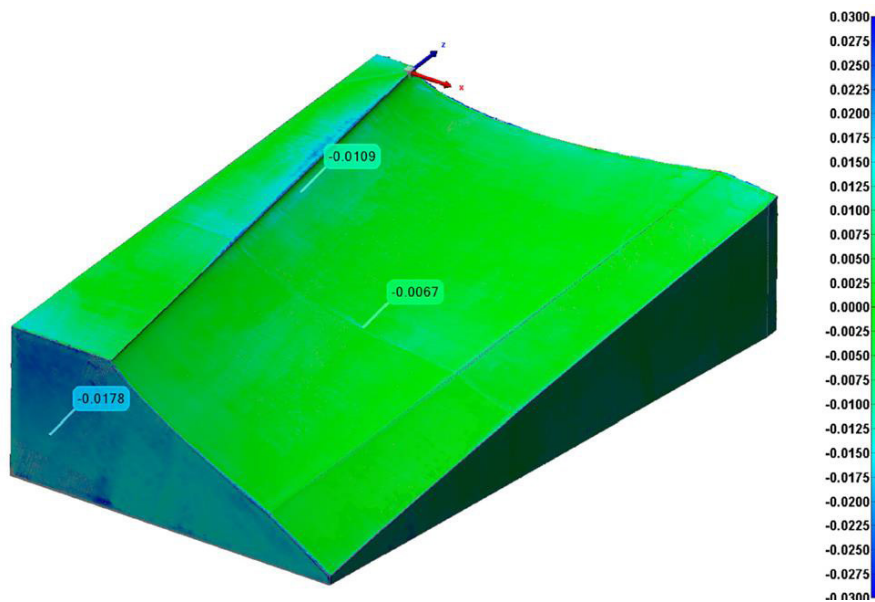

Figure 5-3: 3D scan data for a UAV fan-blade tool (sparse style) with color map comparison to the original CAD model (no thermal cycling). Dimensions are in inches.

Figures 5-4 and 5-5 show the scan data for the same two tools detailed above after 10 thermal cycles. As can be seen, there is negligible dimensional change, particularly given the ± 0.006 inch (0.152 mm) accuracy limit. For the shell tool, over 95% of the tool surfaces are within that limit and over 90% for the sparse tool. Additional investigation into tool accuracy is planned, including the use of a more accurate inspection device (e.g., CMM), and will be included in future design guide releases. Additional thermal cycling is underway for the tools used for the evaluation. Additionally, for the sparse tool in particular, the scan data showed the majority of variation on the vertical surfaces of the tool ends. Whether this is legitimate variation or related to the limitations of the scanning equipment has yet to be verified.

Figure 5-4: 3D scan data for a UAV fan-blade tool (shell style) after thermal cycling with color map comparison to the 3D scan data for the same tool prior to cycling. Dimensions are in inches.
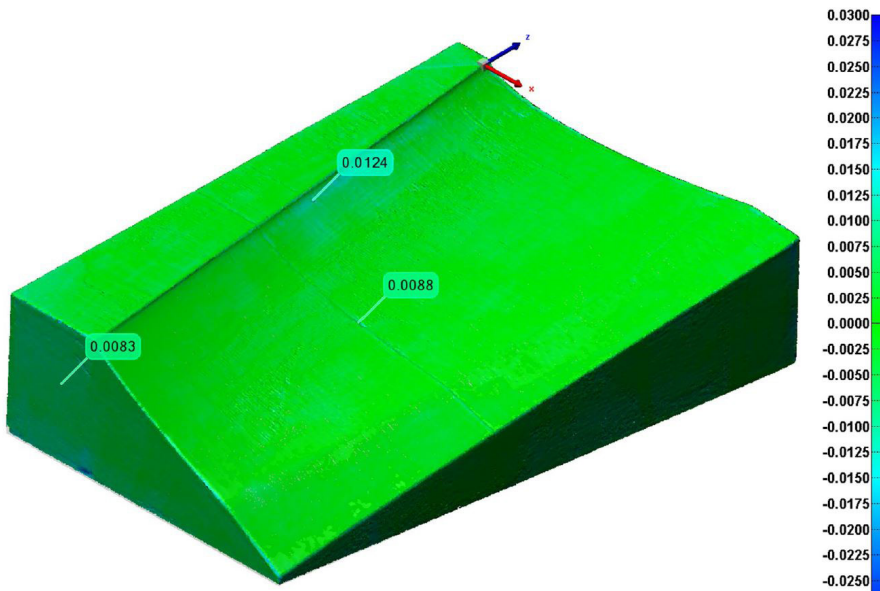

Figure 5-5: 3D scan data for a UAV fan-blade tool (sparse style) after thermal cycling with color map comparison to the 3D scan data for the same tool prior to cycling. Dimensions are in inches.

Elevated Temperature Mechanical Properties

The figures below present data for mechanical testing at critical elevated temperatures. Figure 5-6 shows the stress-strain curve for compression loading of samples built in the ZX orientation at various temperatures. The data set was generated by the Center for Aerospace Manufacturing Technologies (CAMT) at the Missouri University of Science & Technology. Compression testing was done in accordance with ASTM D695.

Figure 5-7 shows flexural modulus data at various critical temperatures for samples printed in the flat, on edge, and vertical orientations. Testing was performed at an external lab in accordance with ASTM D790, procedure A. The graph shows ULTEM 1010 resin reduces in stiffness by 20-30% depending on the print orientation at cure temperatures of 350 °F (177 °C).
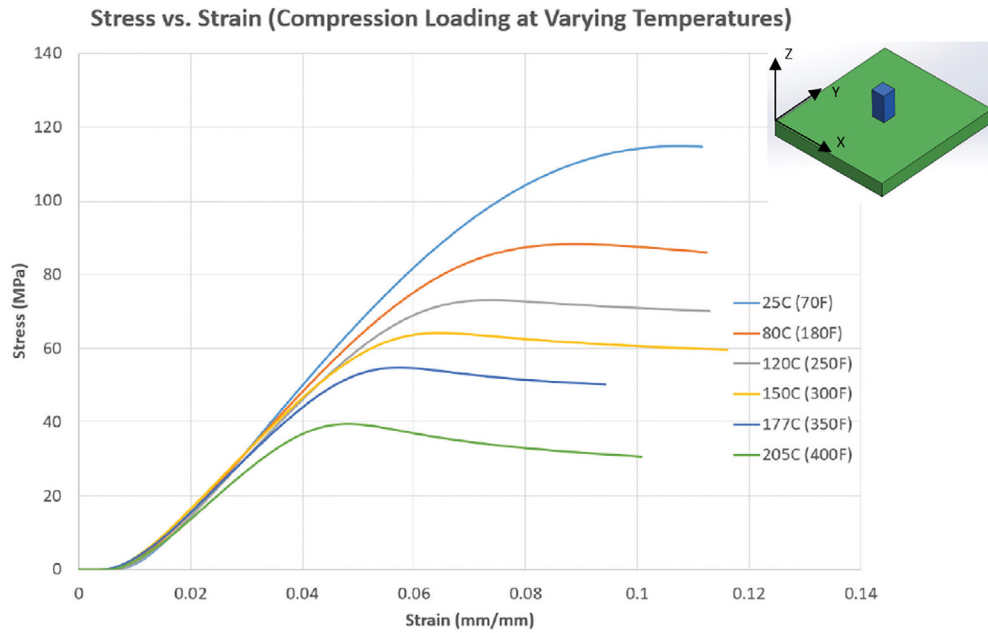

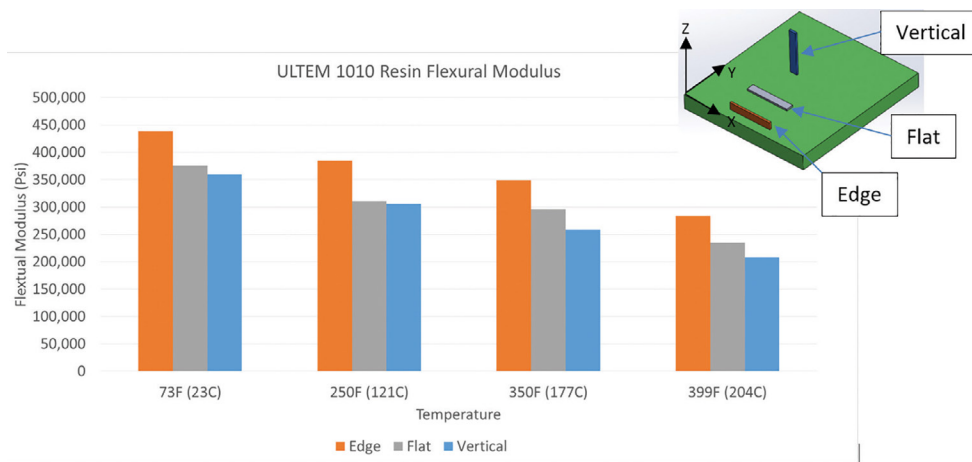Figure 5-6: Stress-strain relationship for ULTEM 1010 resin at various temperatures.

Figure 5-7: Flexural modulus at critical temperatures for various orientations.

FDM FOR COMPOSITE TOOLING 2.0

—

Moisture Sensitivity

Many polymeric materials absorb moisture to some extent over time and at various rates. Per the manufacturer (SABIC), ULTEM 1010 resin will absorb 0.7% when saturated at 75 °F/50% RH (24 °C/50% RH). Since moisture can be detrimental to composite laminate quality, relatively rudimentary testing was performed to ensure that such adverse effects can be prevented with basic precautions.

To ensure saturation and a "worst case" exposure scenario, four tools (two each of shell and sparse construction) were placed in a humidity chamber at 140 °F/90% RH (60 °C/90%) for two weeks. After conditioning, two tools were subsequently dried for 4 hours at 250 °F (121 °C). Eight-ply, quasi-isotropic carbon/epoxy laminates were then produced on each tool. The laminates were visually inspected after cure and then sectioned for microscopy to inspect for porosity, delamination, blistering and other indications of moisture-induced effects. The primary objective was to demonstrate that even in the most severe climates, if moisture absorption becomes a concern, oven drying tools before use is sufficient to prevent adverse effects on cured parts. In reality, most tools in a state of regular use are likely to be stored in environments far less harsh than those tested.

As expected, moisture exposure testing demonstrated that tools dried before use (4 hours at 250 °F (121 °C) produce laminates of acceptable quality (no significant porosity or other obvious issues).

Solvent Exposure

Solvent exposure testing was performed on ULTEM 1010 resin test coupons (unsealed) to verify general compatibility with the most common solvents used in composite fabrication facilities — isopropyl alcohol (IPA), acetone, and methyl ethyl ketone (MEK). During normal operations, most composite tools experience only brief exposure to such solvents, such as when being wiped clean prior to part fabrication. To demonstrate general compatibility, a scenario where a solvent was spilled on a tool and went unnoticed for the equivalent of a weekend (~48 hours) was evaluated to represent a likely worst-case scenario. Test specimens were built in ULTEM 1010 resin and then submerged in a solvent for 48 hours. After exposure, the specimens were removed from the solvent and oven dried for two hours at 250 °F (121 °C) to ensure residual solvent had evaporated. Flexural strength (3-point bend setup) was determined per ASTM D 790 and compared to baseline data (no solvent exposure).

The flexural strength of the exposed specimens after drying returned to full strength relative to the baseline specimens, confirming that if the solvent has evaporated from the tool, final performance is not impacted. During practical manufacturing use, tools will typically only be exposed to small quantities of solvent and briefly, in which case no performance changes are anticipated. The tools will also be sealed on the surfaces mostly likely to be exposed to solvent, which will add an additional level of protection and security.

Tool Life

A thorough understanding of the useful life of a non-metallic tool is critical, particularly for production tooling consideration or for any substantial part volumes beyond prototyping. It is also challenging information to obtain experimentally due to the time and resources involved. In working toward a preliminary baseline, both practical (empirical) and analytical data was gathered.

For empirical testing, the basic approach outlined for the accuracy and thermal stability testing described previously was followed, but extended to more thermal cycles. A single tool geometry (UAV fan blade), built in the two primary shell- and sparse-style constructions, was tested (the tools are shown in Figure 5-1). Tools were cycled for 30, 60, and 90 cycles at 350 °F (177 °F), full vacuum, oven only, followed by evaluation (inspection and 3D scanning) and laminate fabrication (eight-ply, quasi-isotropic carbon/epoxy) with subsequent inspection.

For the analytical portion, dynamic mechanical analysis (DMA) was used to evaluate creep in flexural specimens (3-point bend configuration). Isothermal testing was performed with a 100 psi (689 kPa) loading condition at multiple elevated temperatures (355 °F (179 °C), 385 °F (196 °C), and 400 °F (204 °C)) and then time-temperature superposition (TTS) principles were used to form an understanding of long-term behavior. The basis for use of TTS comes from the demonstrated principle that viscoelastic behavior for a given temperature can be superimposed on data for a different temperature by shifting the curves along the time/frequency axis. Note that the majority of loading applied to composite tooling is not flexural in nature, but rather compressive. Thus, evaluating flexural properties represents a "worst case" loading condition and ensures results are conservative, albeit slightly less directly applicable. An evaluation of compressive creep would be ideal, but such an apparatus was not available at the time of testing.

stratasys

FDM FOR COMPOSITE TOOLING 2.0

Tool Life – Results

For the practical evaluation of cycling tools for 30-90 cycles (in an oven, vacuum only), it was found that the TC-1614 two-part epoxy material used to seal the tools was beginning to break down around 30 cycles and had heavy oxidation and discoloration by 60 cycles. Despite the evidence of the epoxy sealer degrading, laminates were fabricated on tools at both cycle levels with no issues. One tool was also continued to 90 cycles, but the epoxy sealer was no longer capable of consistently adhering to the tool and was pulled from the surface during laminate fabrication, damaging the tool in the process. The tools themselves in unsealed areas had a slight color change, but showed no signs of damage or degradation. Evaluation of alternative sealing materials to address this issue is in work.

For the analytical evaluation of ULTEM 1010 resin, the flexural creep data measured by DMA and shifted to 355 °F (179 °C) using TTS principles is shown in Figure 5-8, along with stress versus time relationship at 355 °F (179 °C) master curve. Again, it is important to note that this data was obtained under flexural loading conditions and is expected to be a significantly harsher loading condition than the actual cyclic compressive loading that composite tooling experiences in reality. That said, the results support that an ULTEM 1010 resin composite tool is capable of performing well beyond the requirements of prototyping volumes. The flexural strain at failure was determined to be 3.5%, tested per ASTM D790. Selecting a strain limit such as 0.5% is not predicted to occur until more than 200 hours of exposure at 355 °F (179 °C) and 100 psi (689 kPa).
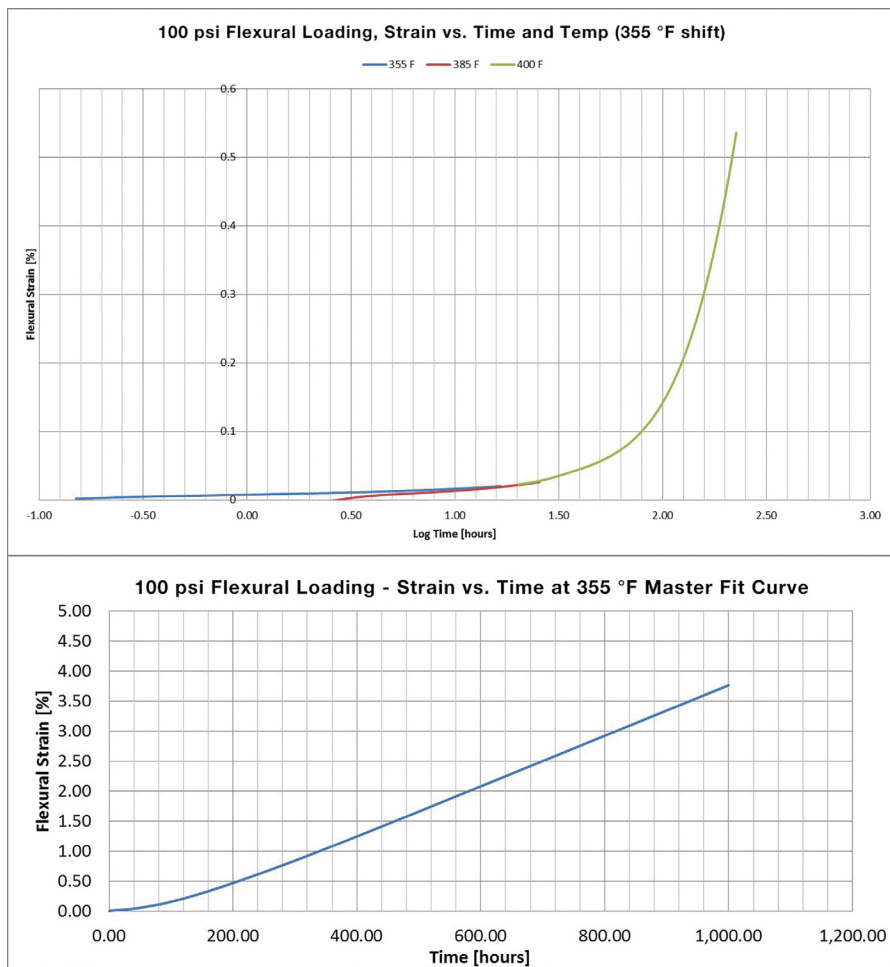
Figure 5-8: Flexural creep data for ULTEM 1010 resin test coupons – shifted to 355 °F (179 °C) using TTS (top) and 355 °F (179 °C) data only (bottom).

FDM FOR COMPOSITE TOOLING 2.0

—

ULTEM 1010 resin demonstrates the ability to perform under harsher loading conditions (flex) for the equivalent of dozens of high-temperature, high-pressure autoclave cycles, perhaps over 100 cycles. And of course, use of lower pressure and/or lower temperature cure cycles will only extend the usable life. This data also suggests that for use with the relatively low loading produced in vacuum bag-only cycles, tool life is not a significant concern for typical aerospace industry part volumes (at least from the perspective of creep-induced tool deformation). Empirical testing highlighted the limitations of the particular sealing material used, but supports the capability of tools built from ULTEM 1010 resin material. Further testing is necessary to more definitively confirm long-term capabilities and develop a more comprehensive understanding. Additional tool life characterization continues to be point of emphasis. Future development data will be included in subsequent design guide release.

Tool Repair

During normal manufacturing operations, the potential for minor tool damage due to handling and routine use is quite high. FDM composite tooling offers several advantages regarding tool damage. First, the tools tend to weigh a fraction of equivalent metal tools. This weight reduction allows for much simpler handling and storage since the need for cranes and forklifts is eliminated for most modest sized tools. For example, the weight for a male tool measuring approximately 21 inches (533 mm) in diameter and 16 inches (406 mm) in height was less than 22 pounds (9.5 kg). If damage does occur, FDM thermoplastics are highly repairable. This can be accomplished printing a new subsection of the tool and attaching it using the same approaches mentioned previously for joining large segmented tools such as thermal welding or structural adhesives, depending on the application and size of the damage. And finally, since the cost and lead time of FDM tools tend to be significantly less than traditional tooling, in the unlikely event of severe damage, it is often feasible from a cost perspective to 3D print a completely new tool. A basic procedure for repairing FDM tooling is provided in Appendix D.

FDM FOR COMPOSITE TOOLING 2.0

SECTION 6 – USE CASES AND EXAMPLES

Customer Success Story – Dassault Falcon Jet Interior Panel

Dassault Falcon Jet is a leader in the business aviation sector. Through a relentless pursuit of technological innovation, Dassault creates the worlds most advanced, efficient and comfortable jets to own and fly. Falcon Jet's desire to innovate, continually simplify the supply chain, reduce time to market and enable dynamic response to production and development environments led them to pursue new manufacturing solutions. The potential for FDM to address many key pain points led to a collaboration between Falcon Jet and Stratasys to investigate the operational and economic advantages of FDM tooling for low to moderate volume composite structures.

The teams at Falcon Jet and Stratasys worked closely together to re-design tooling for a composite structure for an aircraft interior panel to optimize the design for the FDM process. Figures 6-1 and 6-2 show the significant transformation of the lay-up tool and machining fixture for the composite panel.

The lay-up mold for the panel was designed as a sparse style tool with an exposed support structure (i.e. open / uncapped ends) to optimize air flow and minimize material use, Figure 6-1. Although trapezoidal cut-outs were left in the tool to accommodate transportation with a device such as forklift, the FDM tool weighs just 17 pounds (7.7 kg) and can be easily transported by hand. The Falcon Jet composite panel requires a moderate cure temperature of 250 °F (121 °C) at 1 atm (vacuum only), which is well within the performance range of the ULTEM 1010 resin used for the lay-up mold as it can easily withstand cure cycles over 350 °F (177 °C).

The machining fixture for the composite panel was designed with a similar style to the lay-up mold, incorporating an exposed internal structure to reduce material use. Additionally, the machining fixture was designed with printed bracketry to interface with the CNC machining centers and lift points to easily work into the existing workflow and was produced in ASA material to provide additional cost savings.

For comparison, the conventional Fiber Reinforced Polymer (FRP) lay-up tool and machining fixture for the interior composite panel each require 10-16 weeks and $20-30K to procure, Table 6-1. The FDM technology demonstrates the ability to significantly reduce lead time, cost, and weight for each tool by 70-80% or more. By qualifying FDM composite tooling for low to moderate volume production, Falcon Jet is enabling dynamic, cost effective response to demand fluctuation and customization in their production environment.
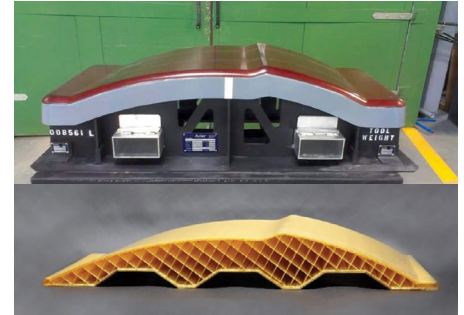
Figure 6-1: Views of the tradition FRP lay-up mold (top) and FDM lay-up mold (bottom).

Figure 6-2: Views of the traditions FRP machining fixture (top) and FDM machine fixture (bottom).

Cost comparison data (based on customer data)

TOOL FUNCTION	TOOLING MATERIAL	COST	LEAD TIME	BUILD TIME	FDM MATERIAL USAGE
Existing Lay-Up Tool	Fiberglass/Epoxy	~$25,000	~16 weeks	----	----
Existing Machining Fixture	Fiberglass/Epoxy	~$30,000	~16 weeks	----	----
FDM Lay-Up Tool	FDM – ULTEM 1010	*$5,600	1-3 weeks	37.5 hrs	382 in³ (6,260 cm³)
FDM Machining Fixture	FDM – ASA	*$4,300	1-3 weeks	48.5 hrs	535 in³ (8,767 cm³)

Table 6-1 *Cost includes material, machine build time, hardware, and labor for preparation and inspection.

stratasys

FDM FOR COMPOSITE TOOLING 2.0

Customer Success Story – Aileron Mandrels

In the development of innovative, next-generation composite structures, a leading business jet OEM approached Stratasys seeking to validate the use of FDM composite lay-up tooling. Using ULTEM 1010 resin mandrels, highly successful proof-of-concept articles for a patent-pending "single shot" (one-piece, single-operation construction) composite aileron were developed and built.

The sub-scale aileron was full length (~24 inches (610 mm)) and thickness, but reduced span (~24 inches) (610 mm) relative to the full 96-inch (2,438 mm) span of the production version. Layup tooling consisted of 21 ULTEM 1010 resin mandrel segments in seven sections. Each section had a main middle section and two short end-caps for assembly as shown in Figure 6-3. The segments were built vertically to minimize material use and optimize surface finish. Total build time for all segments was less than six days on one Fortus 900mc, using less than $3,600 in material. (Refer to Table 6-2 for additional information.) The mandrels were also used for a proprietary initial step in the fabrication process that is critical for proper laminate consolidation and overall dimensional control. Refer to Figures 6-4 and 6-5 for in-process and final aileron images.

With this approach, the customer took advantage of the ULTEM 1010 resin's higher CTE (relative to conventional tooling materials) to enable trouble-free mandrel removal after the part was cured and cooled. Overall, FDM tooling provided increased functionality while reducing both lead time and cost.
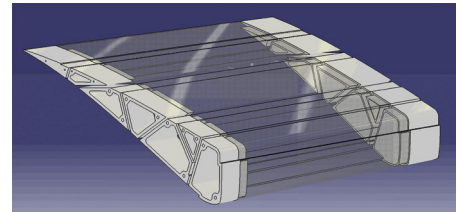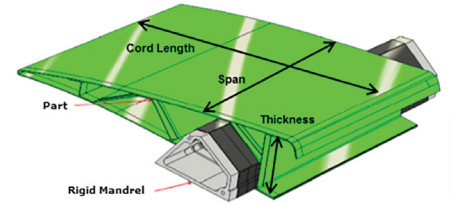
Figure 6-3: Single shot composite aileron schematic (left) and model of seven mandrel sections (right).
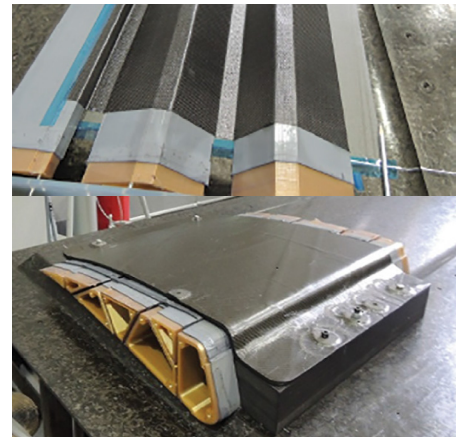
Figure 6-4: Aileron in-process lay-up and final configuration (vacuum bagging not shown).
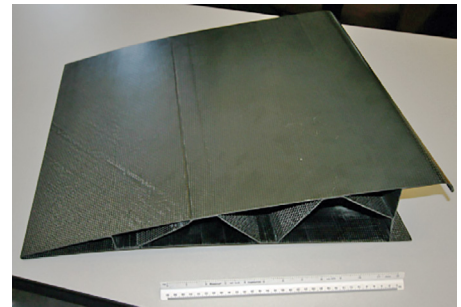
Figure 6-5: Final single shot composite aileron sub-scale demonstrator.

Build, cost, and lead time data

TOOLING MATERIAL	FDM MATERIAL USE	BUILD TIME	COST *	LEAD TIME
ULTEM 1010 resin	480 in³ (7,866 cm³)	234 hours**	$6,950	<2 weeks (single machine)

Table 6-2 * Based on build time and material for a Fortus 900mc (cost amortized over five years, operating at 65% utilization). ** Build time could be cut by 40-50% using the 0.020 inch (0.508 mm) slice height (not available at the time of this project). Resulting tooling cost would be ~$5,600. Material use includes both model and support.

stratasys

FDM FOR COMPOSITE TOOLING 2.0

—

Customer Success Story – Aurora Flight Sciences
Multi-Piece Fairing Tool

Aurora Flight Sciences (AFS) is a recognized leader in aviation and aeronautics research that specializes in designing and constructing special-purpose aircraft. AFS and Stratasys have partnered to evaluate and implement FDM composite tooling, ancillary manufacturing tooling (jigs, fixtures, trim tools, etc.) and flight parts during the development and production of multiple manned and unmanned aircraft structures. AFS was called upon by a key customer to design and produce a large belly-pod fairing (approximately 9 feet (2.74 m) x 24 inches (0.61 m) x 30 inches (0.76 m) [L x W x H]) for a modified Centaur aircraft in a very short timeframe. After receiving multiple external quotes for traditional composite tooling, AFS turned to Stratasys for support.

The size of the fairing required the tool design to be segmented to fit the build chamber of the Fortus 900mc (36 inches (0.91 m) x 24 inches (0.61 m) x 36 inches (0.91 m) [W x D x H]). Additionally, as can be seen in Figure 6-6, the optimally sized part design results in a trapped-tool geometry (i.e. the cured part cannot be removed from the rigid tool without disassembly or destruction). The flexibility of FDM enabled segment designs that allowed the critical, trapped cylindrical section of the tool to drop down out of the part easily after lay-up and curing.

Since the fiberglass/epoxy fairing used low temperature curing [<200 °F (93 °C)], out-of-autoclave materials, the tool was built in PC to save cost. The impact of the higher CTE of PC relative to ULTEM 1010 resin (40% lower than PC) was lessened as a result of the relatively low cure temperature. Sections were built in two construction styles – sparse and hollow shell. The sparse sections were built on a Fortus 900mc in combination with Xtend™ 500 material canisters [500 in³ (9,194 cm³) of material per canister] to reduce build time and material change-overs. After the build, the hollow sections were filled with high-temperature expanding foam to further improve tool rigidity with minimal cost and fabrication time.

Taking full advantage of FDM capabilities, AFS was able to meet the demanding timeline of their customer due to a 60-80% reduction in lead time, while also providing a 60-75% cost savings, compared with traditional tooling (refer to Table 6-3). In addition to the significant savings in cost and time, FDM enabled trouble-free segmentation of the design, permitting the use of a trapped-tool configuration.

This example of FDM composite tooling was featured in a June 2015 article in Composites World magazine (Sara Black, "A growing trend: 3D printing of aerospace tooling," Composites World June 2015: 22-31), also online: http://www.compositesworld.com/articles/a-growing-trend-3d-printing-of-aerospace-tooling

Figure 6-6: Dry fitting the tool after FDM build (top) and tool with cured fiberglass/epoxy fairing (bottom).

Figure 6-7: Composite belly pod fairing installed on Centaur aircraft (before and after painting).

Cost comparison data (based on customer data)			
TOOLING MATERIAL	COST	LEAD TIME	FDM MATERIAL USAGE
Aluminum	$65,000	7 weeks	--
Carbon / Epoxy	$95,000	12-14 weeks	--
FDM – Polycarbonate	$25,000	2-3 weeks*	4,864 in³ (79,707 cm3)

Table 6-3 * Based on parallel build of tool segments on multiple machines using Stratasys Direct Manufacturing, demonstrating a case of using a support bureau to manage excess build capacity.

stratasys

FDM FOR COMPOSITE TOOLING 2.0

Customer Success Story – RockWest Composites Radome

Rock West Composites is a full-service supplier for composite products offering services from initial engineering and product development to prototyping as well as low- and high-volume production. They utilize a wide variety of manufacturing techniques and leverage decades of industry experience to deliver customized, economical, high-performance solutions.

After working with Stratasys on various development projects that showcased the unique lead time benefits of FDM tooling, Rock West was approached by a customer needing to produce a 20-inch (508-mm) diameter, low dielectric (quartz) radome for flight testing in less than 4 weeks. Rock West recognized the tight timeline and low volume nature of the project as a perfect opportunity to leverage the unique strengths of FDM for the radome tooling.

Stratasys and Rock West collaborated closely to design a mold and drill guide for the radome. The lay-up mold was constructed with ULTEM 1010 resin for high temperature [350 °F, (177 °C)] curing and the drill guide was constructed from a more cost-effective material, ASA. The lay-up mold incorporated a combination of the shell and sparse style designs by utilizing a two-inch (51-mm) thick shell with a 0.5 inch (12.7mm) sparse double-dense fill pattern, Figure 6-8. The design freedom afforded by the additive manufacturing process allowed for this optimization of the design which provided a rigid structure that also minimized material use and cost. The lay-up mold and drill guide were both produced in three days enabling production and installation of the composite part on the aircraft in less than four weeks. The use of FDM tooling enabled Rock West to produce a precision product under strict time constraints. Figure 6-9 shows views of the lay-up mold and drill guide.

Figure 6-8: View of the composite part (white) on the FDM lay-up mold (amber).
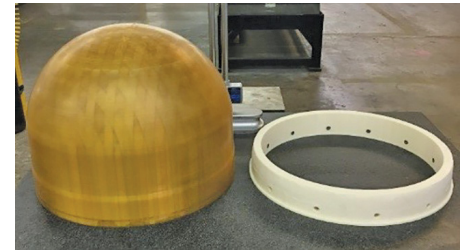
Figure 6-9: View of the FDM lay-up mold (left) and ASA drill guide (right).

Cost comparison data (based on customer data)

TOOL FUNCTION	TOOLING MATERIAL	COST	FDM BUILD TIME	LEAD TIME	FDM MATERIAL USAGE
Lay-Up Tool	Epoxy Board	$5,000	----	4-6 weeks	----
Drill Guide	Aluminum	$2,500	----	4-6 weeks	----
Lay-Up Tool	FDM – ULTEM 1010 resin	*$4,500	57 hrs	1 week	470 in³ (7,702 cm³)
Drill Guide	FDM – ASA	*$550	10 hrs	1 week	92 in³ (1,508 cm³)

Table 6-4 *Cost data accounts for raw material and machine run time assuming ownership of a Fortus 900mc.

FDM FOR COMPOSITE TOOLING 2.0

Customer Success Story – Penske Race Team: High-Temperature Masters and Sacrificial Mandrels

Team Penske is one of the most successful teams in the history of professional sports. With more than 450 major race wins, they have a history of advancement and innovation. Team Penske is now looking to FDM technology to speed development and enable dynamic response and fabrication of composite structures.

Team Penske needed to rapidly manufacture a fuel probe body out of a carbon fiber composite material for an upcoming race. The fuel probe body is part of a larger assembly used in the pit to rapidly refuel the race car, Figure 6-10. Traditionally, the fuel probe body is manufactured from aluminum sheet metal and tubing. The race team wanted to move to a composite fuel probe to allow better incorporation of feedback sensors, reduce weight, and improve cosmetics. However, the tight time constraints of the race season and long lead times for traditional composite tooling methods made production of a composite fuel probe body nearly impossible. Team Penske chose to manufacture the master pattern and cores for the fuel probe body with FDM technology to dramatically reduce their tooling lead times and enable production for the upcoming race.

Figure 6-10: Views of the fuel probe body.
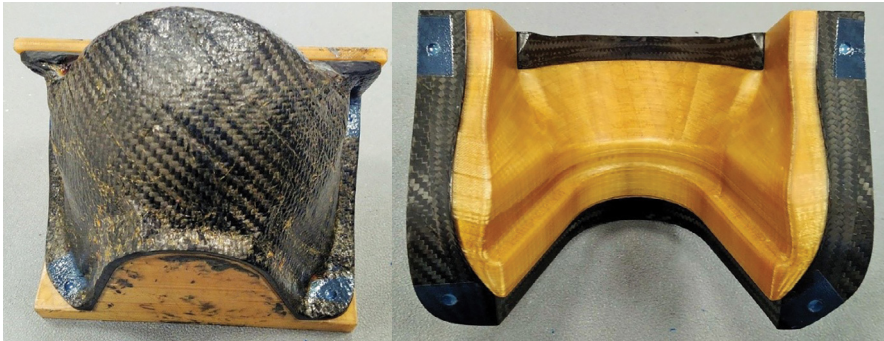
Figure 6-11: Views of the composite mold master pattern during initial fabrication.

Cost comparison data (based on customer data)

TOOL FUNCTION	TOOLING MATERIAL	COST	FDM BUILD TIME	LEAD TIME	FDM MATERIAL USAGE
Master Pattern	Epoxy tooling board	$8,500	----	3-4 weeks	----
Wash-out Core	Soluble ceramic	$1,200	----	4-6 weeks	----
Master pattern	FDM – ULTEM 1010 resin	$870	30.5 hrs.	<1 week	53.5 in³ (879.7 cm³)
Wash-out Core	FDM – ST130	$200	7 hrs.	<1 week	18 in3 (295 cm³)

Table 6-5 *Cost includes material, machine build time assuming ownership of a Fortus 900mc.

FDM FOR COMPOSITE TOOLING 2.0

Customer Success Story – Swift Engineering, Inc. UAV Propeller Blade Compression Molding Tool

Swift Engineering, Inc. is a recognized leader in motorsport and aviation product development and manufacturing, with an extensive pedigree in open-wheel racing and a strong emerging presence in aerospace. While developing schedule-critical hardware for wind tunnel testing, Swift took full advantage of the time-saving advantages of FDM composite tooling to quickly produce a complex, matched mold for compression molding carbon fiber-reinforced epoxy UAV propeller blades.

The approximately 14 x 4 x 2 inch (356 x 102 x 51 mm) mold halves were built using a relatively small layer thickness [0.01 inch (0.25 mm), T14 build tip] in a horizontal build orientation, as shown in Figure 6-12. Due to the complex shape, there was no build orientation perfectly suited to eliminate stair-stepping; this orientation was chosen to minimize support material while still reducing stair-stepping in most areas of the tool. The tool was built in a solid construction in ULTEM 1010 resin, which provides the required temperature resistance and mechanical performance. No significant design-for-additive-manufacturing optimization was performed on the design due to time constraints, meaning potential remains to further reduce build time and material use. As designed, the two mold halves took 30 hours of build time, as shown in Table 6-6. For post-processing, the mold halves were manually abraded and sealed with a two-part epoxy, resulting in a final surface finish smoother than 16 μin (0.4 μm) Ra.

The tool has been used successfully to produce multiple sets of propeller blades, shown in Figure 6-13, for wind-tunnel testing. The specific processing details are proprietary, but the carbon/epoxy blades are cured at a temperature of approximately 250 °F (121 °C) and pressures exceeding 500 psig (3,447 kPa). The resulting blades meet all initial inspection requirements. Wind-tunnel testing will be used to evaluate performance of the blade design and validate the use of FDM for the molds.

Using FDM technology, Swift Engineering met aggressive timelines and all initial technical objectives for its innovative product development and evaluation process while realizing more than 50% cost savings on the complex compression mold tooling.
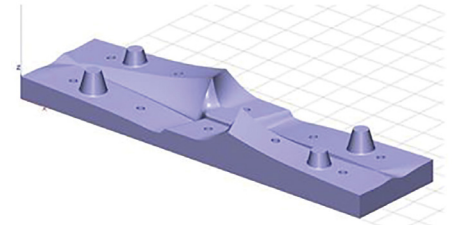
Figure 6-12: Propeller compression molding tool: horizontal build orientation (top), matched mold halves (bottom).

Figure 6-13: Carbon/epoxy composite propeller blade produced on a compression mold tool made from ULTEM 1010 resin.

FDM build and cost data for the propeller blade compression mold tools

TOOL GEOMETRY	~ SIZE	FDM MATERIAL USE	BUILD TIME	COST*
Upper mold	14 x 4 x 2.5 in. (356 x 102 x 63.5 mm)	52 in³ (852 cm³)	17 hours	$625
Lower mold	14 x 4 x 2 in. (356 x 102 x 51 mm)	36 in³ (590 cm³)	13 hours	$450

Table 6-6 *Based on build time and material for a Fortus 900mc (machine cost amortized over 5 years, operating at 65% utilization).

stratasys
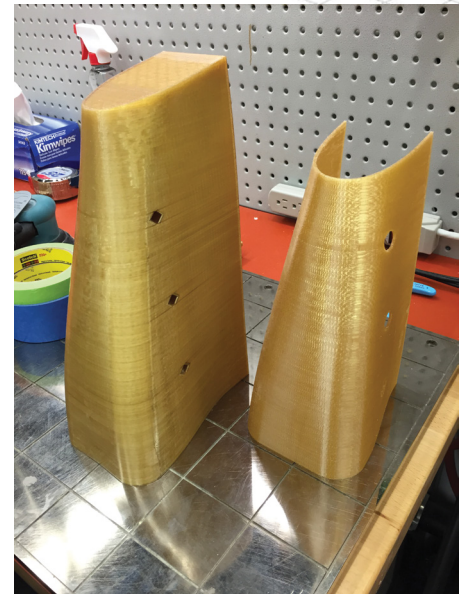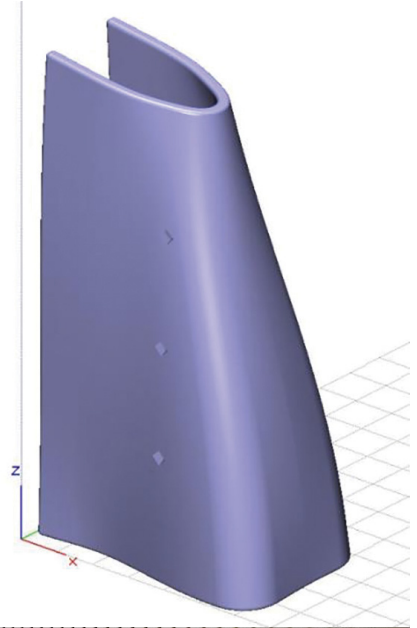
FDM FOR COMPOSITE TOOLING 2.0

Customer Success Story – Aerospace Repair Tools

Leading aerospace companies have collaborated with Stratasys to evaluate and establish ULTEM 1010 resin as a qualified composite repair tooling material. FDM provides great advantages in terms of tooling cost, and — most importantly for repair situations — timeliness. One organization set the lofty requirement of having repair tooling and the resulting composite repair laminate produced in less than 48 hours from the release of engineering documentation. FDM is one of the few technologies capable of consistently meeting this objective while delivering the equally important 350 °F (177 °C) cure temperature capability. FDM demonstrated the ability to meet all requirements while producing high-quality laminates in the process.

FDM composite tooling, and ULTEM 1010 resin in particular, was thoroughly characterized for outgassing, moisture sensitivity, solvent compatibility, and more, both on test panels and common repair tool shapes, such as those in the images shown at right.
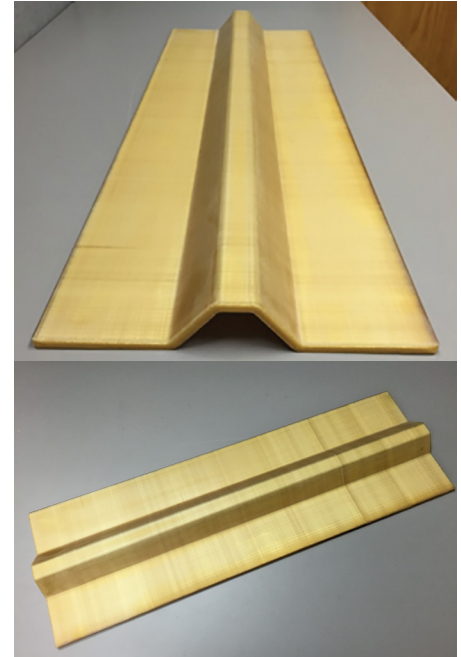
Figure 6-14: Common repair tool geometry – hat stiffener shape.

Repair tool build data (ULTEM 1010 resin)

TOOL GEOMETRY	~ SIZE	FDM MATERIAL USE	BUILD TIME	COST*
Hat Stiffener	28 x 9 x 2 in. (711 x 229 x 51 mm)	80 in³ (1,311 cm³)	13 hours	$780
Contoured Patch	25 x 25 x 2 in. (635 x 635 x 51 mm)	320 in³ (5,244 cm³)	40 hours	$2,930

Table 6-7 *Based on build time and material for a Fortus 900mc using 0.02 inch (0.508 mm) slice height (machine cost amortized over 5 years, operating at 65% utilization).
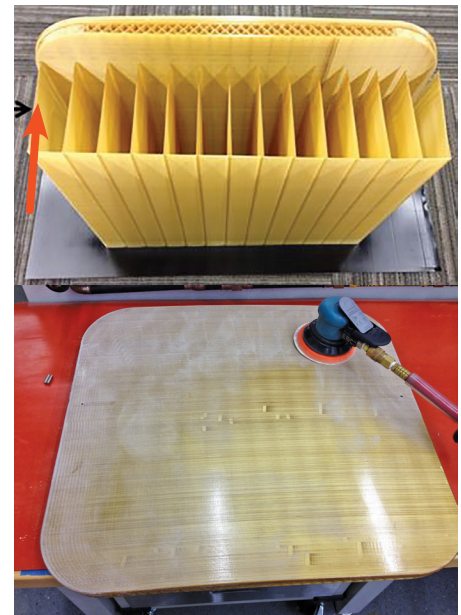
Figure 6-15: Complex contour patch repair tool as built (top, with arrow indicating stabilizer walls) and during post-processing (bottom).

stratasys

FDM FOR COMPOSITE TOOLING 2.0

UAV Shroud Tool

Background: The UAV shroud tool was developed with and used by a leading aerospace company to produce flight hardware, primarily for the purpose of evaluating FDM tooling and comparing/contrasting to traditional metal tooling. Both male and female (not shown – evaluation still in work) variants of the tool were evaluated, giving the flexibility to control either the inner mold line (IML) or OML surface of the part.

Approach: Both shell- and sparse-style tools were produced in ULTEM 1010 resin, permitting cure temperatures of >350 °F (177 °C), although this application required only 250 °F (121 °C). The male shell and sparse tools were produced to provide a comparison, as it was originally thought the shell tool would provide a time and cost savings. However, as can be seen in the table that follows, there is little difference in cost for a tool of this size and the build time is longer as a result of the increased number of surface contours required for the thicker shell tool surface (straight-line rasters are extruded more quickly than non-linear contours). The female tool was produced to provide the ability to control the exterior aerodynamic surface of the part and was built slightly larger in size to allow for material layup. As a result, build time and cost are slightly higher for that tool. All tools were built in a vertical orientation to minimize stair-stepping and support-material consumption. For post-processing, tools were manually abraded and sealed with a two-part epoxy, resulting in a final surface finish smoother than 16 µin (0.4 µm) Ra. The resulting part (produced on the male sparse tool) is also shown in Figure 6-16 (a proprietary coating is shown on the bag-side surface).

Additionally, a separate drill and trim tool (Figure 6-17) was created to supplement the lay-up tool. Typically, such tools are printed in lower-cost materials. In this case, due to the small tool size, they were produced in ULTEM 1010 resin in the same build as the lay-up mold. This ancillary tool is designed to nest on the resulting part and uses index holes to ensure accurate alignment and final trim profile.

Results – All tools were built and post-processed in less than 3 days at a cost less than $600 (each).*

* Costs are based on build time and material for a Fortus 900mc (machine cost amortized over 5 years, operating at a 65% utilization rate).
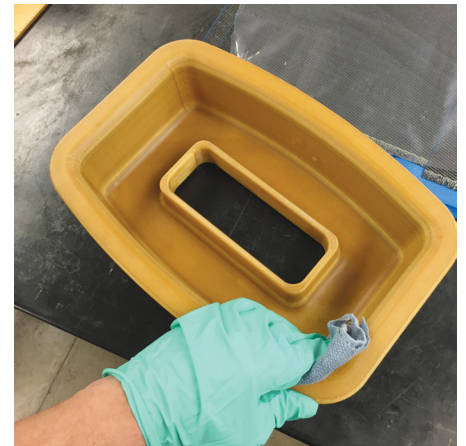

Figure 6-16: FDM tool in ULTEM 1010 resin (top) and resulting part (middle, bottom).

FDM FOR COMPOSITE TOOLING 2.0

UAV Shroud Tools

DESIGN STYLE	MALE SHELL	MALE SPARSE
Construction	0.3 inch thickness (7.6 mm)	0.25 inch (6.35 mm) sparse spacing, 0.1 inch (2.54 mm) shell thickness
Size (approximate)	11 x 4 x 6 inches (279 x 102 152 mm) [L x W x H]	
Build Orientation	Vertical	Vertical
Slice Height	0.01 inch (0.254 mm) (T14)	0.01 inch (0.254 mm) (T14)
Material Use	37 in^3 (606 cm^3)	48 in^3 (787 cm^3)
Support Material Use	0.1 in^3 (1.6 cm^3)	0.3 in^3 (4.9 cm^3)
Build Time	18.5 hours	15 hours
Tool Weight	2.2 pounds (1 kg)	4.9 pounds (2.2 kg)
Cost* – machine owner	**$535**	**$570**
Cost – service bureau	**$1870**	**$1790**
Intended Use	• 250 °F (121 °C) cure temp • Carbon/epoxy laminate for a UAV shroud • Low-volume part fab (shell) or production volumes (sparse)	
Comments and Highlights	• Design style – for a production tool, the sparse design is recommended. The shell tool provides an inexpensive option for very low volumes. • Vertical build orientation minimizes support material use. • Largest layer height selected to minimize build time; the design lacks steep contours or features that would drive the need for a finer resolution. • Tools are appropriate for use with either surface or envelope bagging methods. • Data for the female tool not shown (evaluation in progress) – 20 hr build time/40 in³ (655 cm³) material use/$580 cost.	

* Costs are based on build time and material for a Fortus 900mc (machine cost amortized over 5 years, operating at a 65% utilization rate).

Figure 6-17: Top – Shell-style shroud tool vertical build orientation (top); sparse-style lay-up tool and corresponding drill and trim tool (bottom).

UAV Bulkhead Tool

Background: The bulkhead tool is an example tool design based on a production component, modified to demonstrate the design of a deep-draft female tool for this guide. The layup surface represents the OML of the resulting part, for which dimensional control is critical to ensure proper fit-up and assembly within the aircraft structure.

Approach: Since the bulkhead tool shape is inherently rigid and there is little benefit to considering a sparse tool, a shell design was used. Due to the female shape and depth of the tool, thermal expansion was an important consideration to prevent the cured part from becoming tool-locked or damaged. This tool was intended for use at 250 °F (121 °C) cure temperatures. Despite the lower temperature requirement, ULTEM 1010 resin was the material of choice because it provides the lowest CTE of FDM materials. The tool was built in a flat ("horizontal") orientation with a fine-resolution build tip [0.010 inch (0.254 mm) slice height]. This is the optimal build orientation for this geometry to minimize stair-stepping in the internal radii of the part. The top flanges of the tool were included to assist in part extraction after cure. They provide areas to layup material outside the EOP that can be used for leverage during part removal and subsequently trimmed away. However, there is a tradeoff in that the optional flanges do require a significant amount of support material so alternative designs are worth considering to further optimize the tool. For post-processing, tools were manually abraded and sealed with a two-part epoxy, resulting in a final surface finish smoother than 16 µin (0.4 µm) Ra.

Results: A female tool design intended for 250 °F (121 °C) cure temperatures was achieved, built and post-processed in less than three days at a cost less than $980.*

* Costs are based on build time and material for a Fortus 900mc (machine cost amortized over 5 years, operating at 65% utilization).

Figure 6-18: UAV bulkhead tool.

Figure 6-19: UAV bulkhead tool being prepped for use.

FDM FOR COMPOSITE TOOLING 2.0

UAV Shroud Tools

DESIGN ASPECTS	SELECTION/DETAILS
Construction	0.3 inch (7.6 mm) shell thickness
Size (approximate)	13 x 9 x 3 inches (L x W x H)
Build Orientation	Horizontal (flat) with anchor pins
Slice Height	0.010 inch (T14 tip)
Material Use	57 in³ (934 cm³)
Support Material Use	15 in³ (246 cm³)
Build Time	31.5 hours
Tool Weight	2.6 pounds (1.2 kg)
Cost – machine owner	$980
Cost – service bureau	$3,400
Intended Use	• 250 °F (121 °C) cure temp, 15-100 psi (103-689 kPa) pressure • Carbon/epoxy structure for a UAV bulkhead
Comments and Highlights	• Design style – as designed, the tool is ready for use at low or production volumes • Regardless of cure temperature, ULTEM 1010 resin is recommended to minimize the impact of thermal expansion • Horizontal build orientation provides the shortest build time and least amount of support-material use • Fine build resolution selected to minimize stair-stepping in the tight internal radii, resulting in a slightly longer build time but improved surface finish and reduced post-processing labor • Tool is best suited to envelope bagging methods due to the complex shape • An anchor column was used to prevent part separation from the build sheet due to thermal shrinkage. More information is found in section 8 – FDM Insight Software for File Processing.
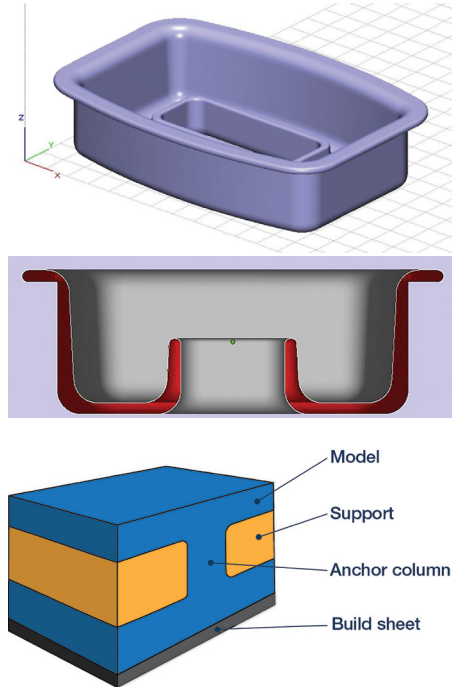
Figure 6-20: Horizontal build orientation (top); cross-section of tool design (middle); cross-section showing anchor column, used to prevent thermal distortion during the FDM process (bottom).

FDM FOR COMPOSITE TOOLING 2.0

—

SECTION 7 – INTRODUCTION TO FDM SACRIFICIAL TOOLING

Additive manufacturing for composite tooling has fundamentally changed the approach for creating complex, hollow composite parts. While current basic shapes with constant cross sections can be produced using traditional composite manufacturing techniques and FDM tooling, complex composite parts with hollow interiors (trapped-tool geometries) present unique challenges.

FDM provides multiple solutions for these challenges, depending on the requirements of the application. ST-130 is a soluble material with a cure temperature limit of 250 °F (121°C) that enables straightforward production of sacrificial tooling for hollow and highly complex composite parts using a thermoplastic material that dissolves in a basic (>7 pH) solution. This solution eliminates many of the design and manufacturing limitations for composites fabricated using eutectic salts, collapsible hard tooling, inflatable bladders, and other sacrificial tooling materials and methods. Stratasys has developed a comprehensive design guide, "Sacrificial Tooling for Composite Part Fabrication" (www.stratasys.com/landing/sacrificial-tooling), addressing the application of ST-130 for capable, cost-effective wash-out tooling.

Beyond the temperature limits of the ST-130 solution, ULTEM 1010 resin support material can be used to produce sacrificial tooling capable of withstanding cure temperatures exceeding 350 °F (177 °C). Unlike ST-130, ULTEM 1010 resin support material is not soluble. However, it does become very brittle with exposure to acetone and can be manually broken away after composite part fabrication. Additional information on the use of sacrificial tooling produced with FDM ULTEM support materials will be provided in subsequent versions of the sacrificial tooling design guide.

SECTION 8 – FDM INSIGHT SOFTWARE FOR FILE PROCESSING

The CAD file must be processed using Insight software before it is ready to print. This lets the user customize toolpaths to improve (or modify) final performance. This software comes with the installation of a Fortus 3D Printer. Formal training for Insight is available. Visit Stratasys.com/customer-support/training to register or for more information. The following sections are intended as a reference for the basic commands, and not as an alternative to the formal classes.

The workflow of preparing a file is:

1. Import STL file
2. Select printer, material and tip size
3. Slice part
4. Add support material
5. Generate toolpaths
6. Verify toolpaths
7. Estimate time
8. Send to printer

FDM FOR COMPOSITE TOOLING 2.0

—

Open Insight software.
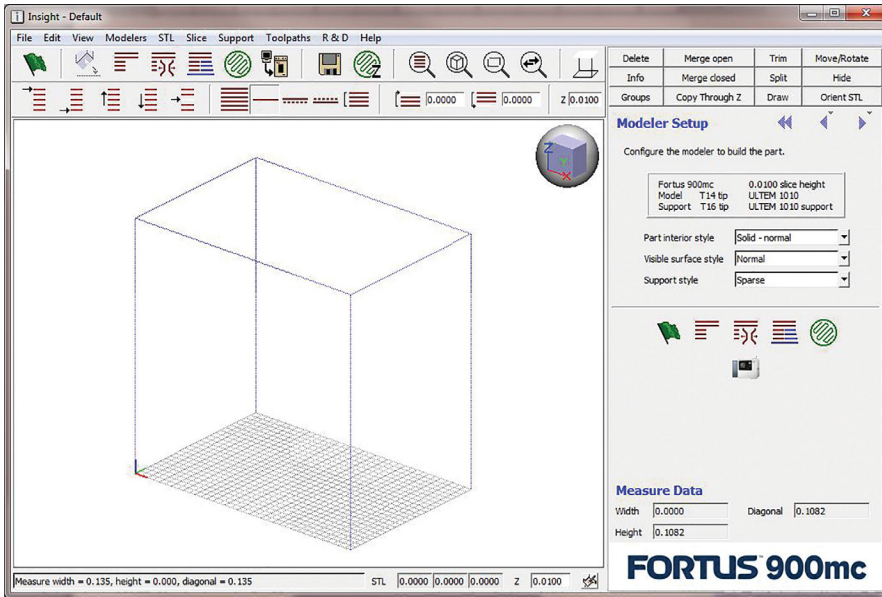
Figure 8-1: Home screen of Insight software upon opening.

The main commands can be found in the upper left part of the screen. Starting with **Orient Part** icon, move from left to right. The **Do-All** command is used for simple geometries and processes the entire file automatically. This function is not recommended for composite tooling because the performance depends on customized toolpaths.
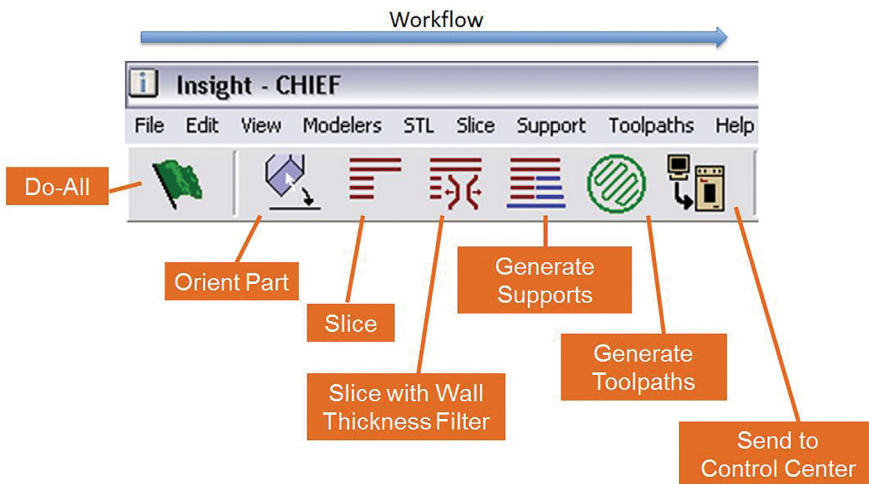
Figure 8-2: Workflow commands for Insight software.

stratasys

FDM FOR COMPOSITE TOOLING 2.0

Import STL

Insight only allows files with the extension ".stl" to be imported. Most CAD software is capable of exporting in STL format. Import the file by selecting, *File*, *Open* and *Select .STL file*.
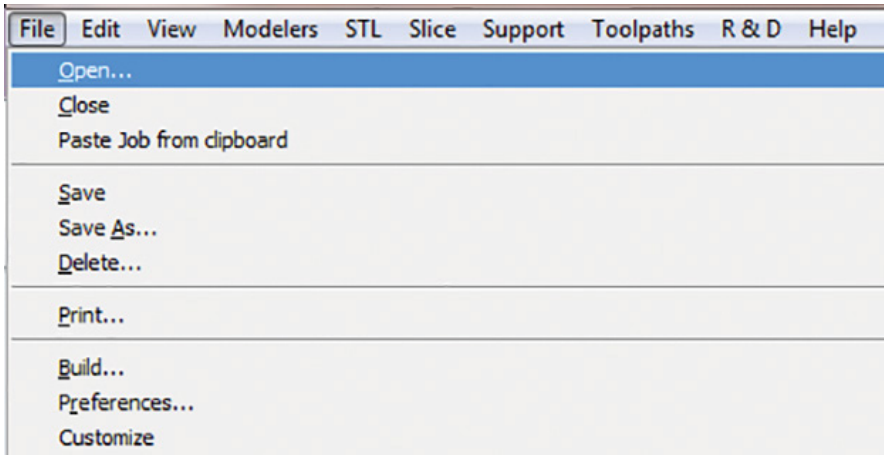
Select printer, material and slice height

1. Choose the printer, material and slice height by selecting the printer icon (shown circled in red below) on the right side of the screen.

stratasys
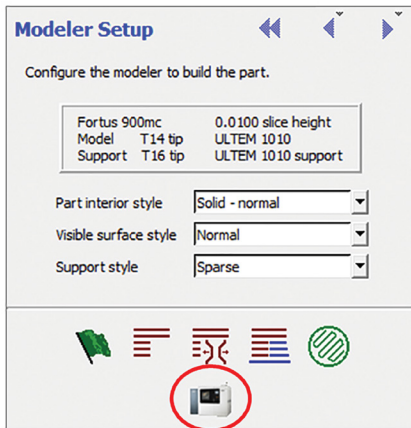
FDM FOR COMPOSITE TOOLING 2.0

—

A new menu will appear in the middle of the screen.
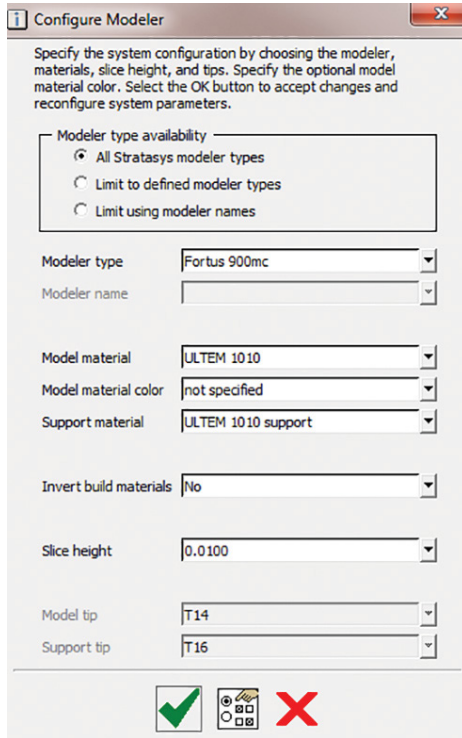
2. Begin by selecting **Modeler type** to choose the printer.

3. Click on **Model material** to choose the type of material.

4. Support material options are dependent on the chosen model material. In this case, only ULTEM support is available since the model material is ULTEM 1010 resin.

5. Verify that **Invert build materials** is selected as **No**. This feature is used for sacrificial tooling.

6. Finally, select **Slice height**, which will correlate to the tip size. In this case, a 0.010 inch (0.254 mm) slice will require a T14 tip. The table below lists the available tip sizes and corresponding slice heights and bead widths for ULTEM 1010 resin. NOTE – Tip size/slice height relationships shown in the table are for ULTEM 1010 resin only. Bead width can also be varied (within limits) to minimize internal porosity and ensure proper contact between contours.

Tip size and slice height specifications for ULTEM 1010 resin

TIP SIZE	SLICE HEIGHT (INCHES)	SLICE HEIGHT (MM)	BEAD WIDTH (INCHES)	BEAD WIDTH (MM)
T14	0.010	0.254	0.020	0.508
T20	0.013	0.330	0.026	0.660
T40	0.020	0.508	0.040	1.016

Table 8-1

7. Click the green check.

FDM FOR COMPOSITE TOOLING 2.0

Orient the part

The orientation of the part is an extremely important step that will impact the surface finish (stair-stepping), build time and amount of support material used. Determining the most critical surfaces of the part will help with orientation selection. Typically, parts should be oriented to minimize stair-stepping and provide the best surface finish. Additionally, overhanging features with angles greater than 45°, relative to the build platform, do not require support material. The following example will show how to correctly orient a part so that it prints with the best surface finish and least amount of support material.

For quick orientation, the user can select a surface on the tool and reference it as the top, bottom, left, right, front or back.
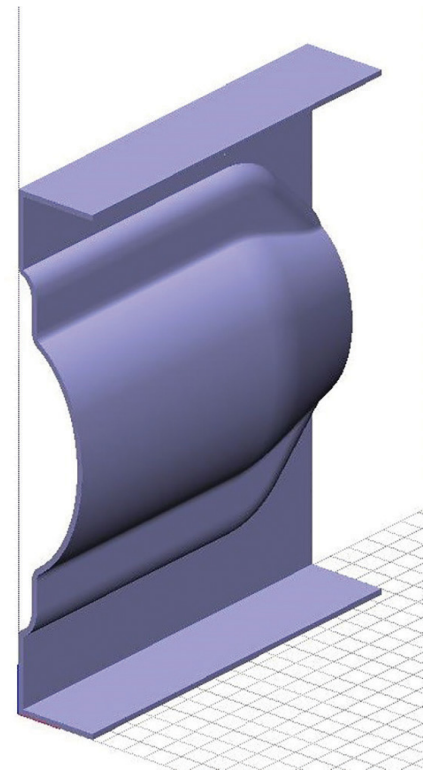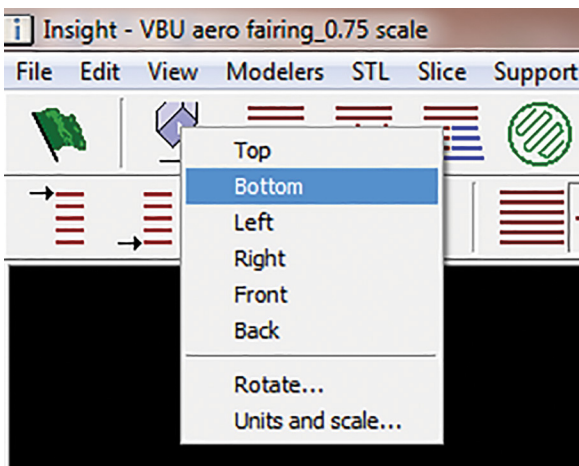
Figure 8-3: Orientation of part after being imported into Insight.

Insight also allows the user to orient the part at a specific angle. Select the **Orient Part** icon and then **Rotate**.
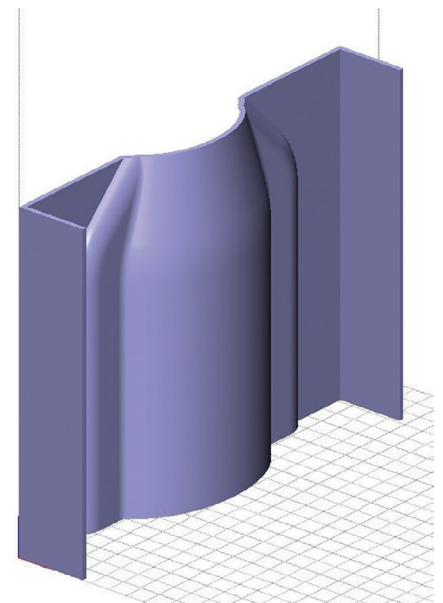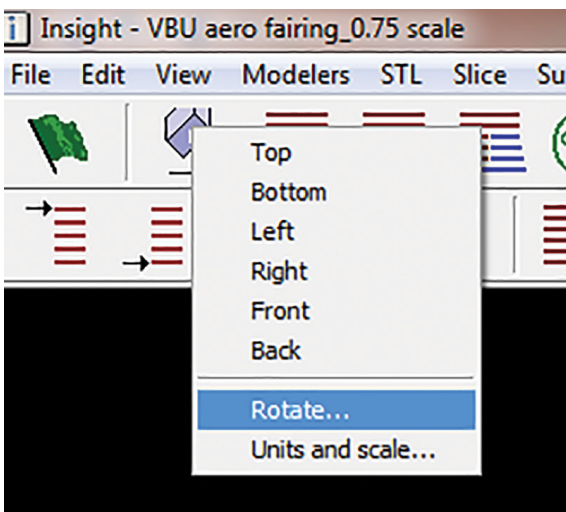
Figure 8-4: Orientation of part after selecting the bottom reference.

stratasys

—

The menu below will appear on the right side of the screen. Determine which axis to revolve the part around and by what increment. Enter a value from 1° to 180° for the Rotate increment and select one of the six axis orientations to rotate the part.
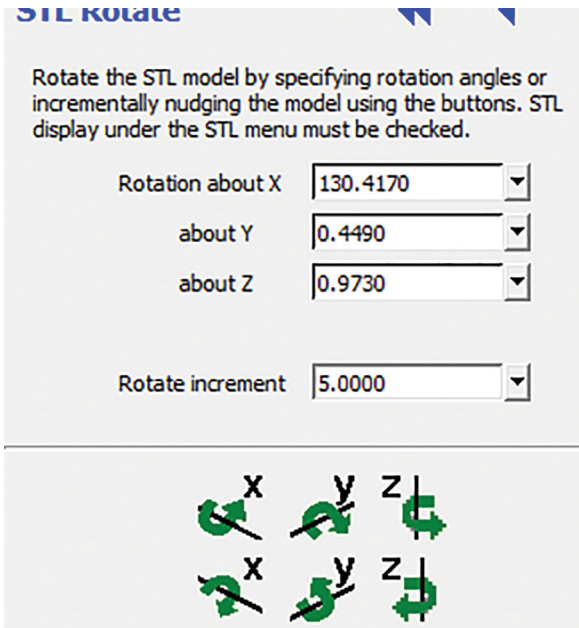
The part is now in the optimal orientation.
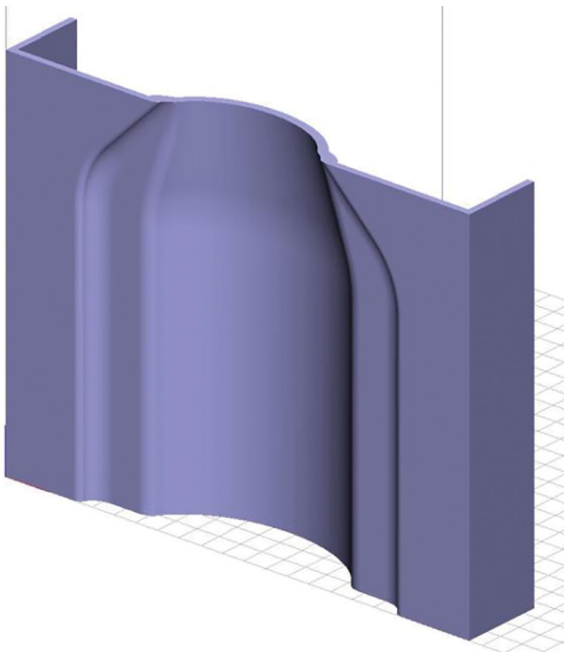
Figure 8-5: Part in optimal orientation after rotation.

FDM FOR COMPOSITE TOOLING 2.0

Slice the part

Slicing the part will section it into multiple layers. Each layer will have a specific toolpath that the printer follows to create the part. The slice height is determined by the tip size. To slice the part, select the icon indicated by the red circle.
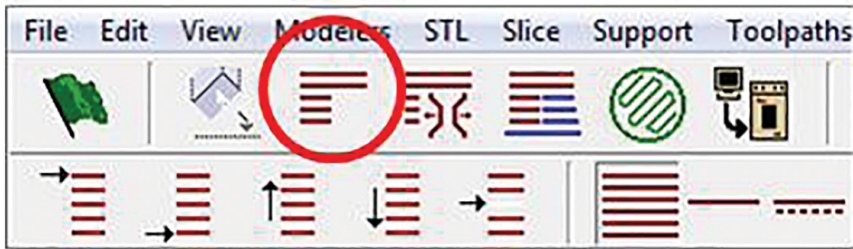
Generate support

Support generation is critical for a quality part as it prevents overhangs from sagging. Insight offers a variety of support options. Selecting the **Generate-Support** icon will create support according to default settings.
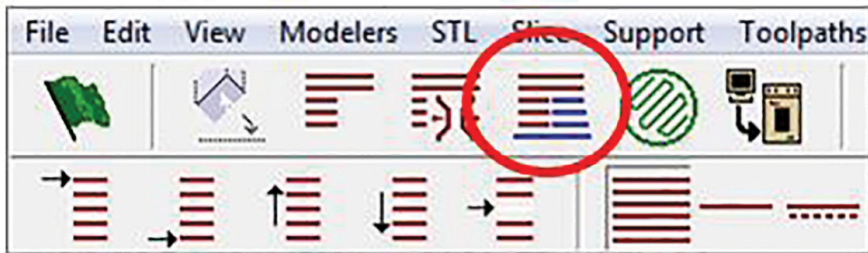
The support material can be modified to reduce the amount of support required and/or time to print. Click on **Support**, then **Setup**.

stratasys
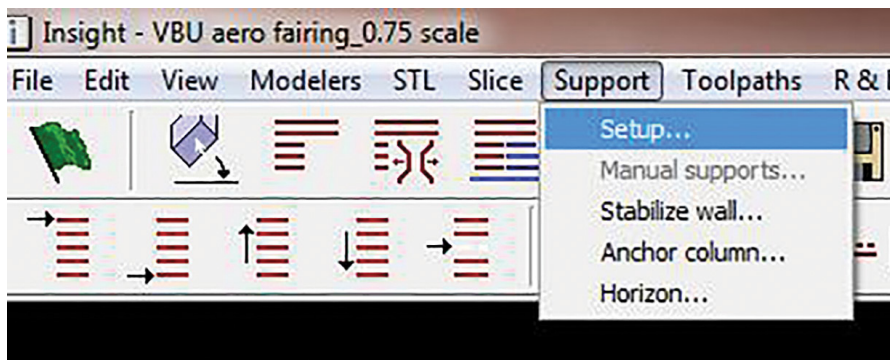
FDM FOR COMPOSITE TOOLING 2.0

—

The Support Setup menu will appear on the right side of the screen. Select the icon circled in red to go to the advanced settings.

The following menu will appear in the middle of the screen. It allows the user to set various parameters of support generation based on their desired outcome. In general, default support settings are recommend. Selecting the "use model material where possible" option can be useful to significantly reduce build time.

The following menu will appear in the middle of the screen. It allows the user to set various parameters of support generation based on their desired outcome. In general, default support settings are recommended. Selecting the "use model material where possible" option can be useful to significantly reduce build time.
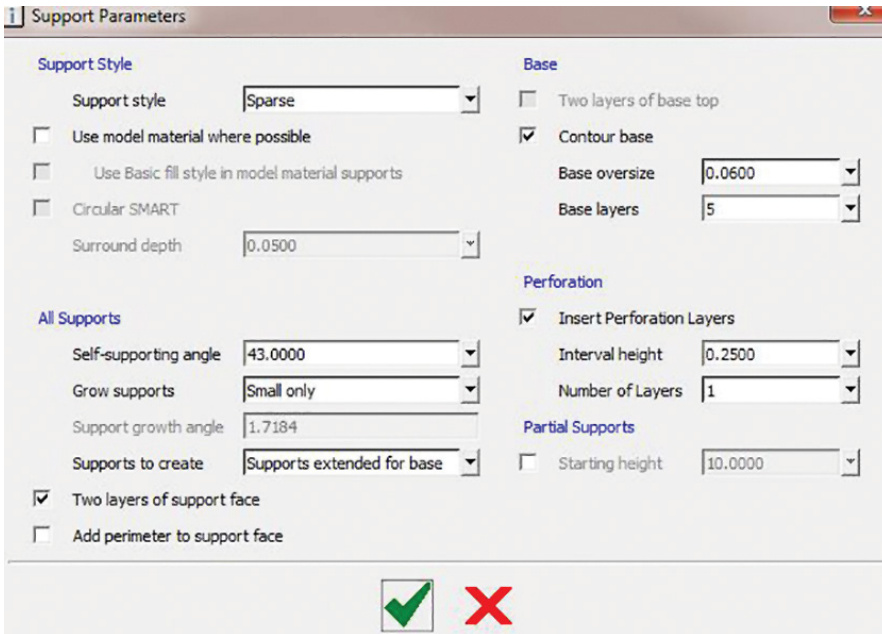
FDM FOR COMPOSITE TOOLING 2.0

Generate toolpaths

The diagram below shows the default "infill" parameters that can be selected for building the model. Additional fill pattern options can be accessed through the custom groups menu. The tradeoff among patterns is cost, strength and print time. Parts with a solid fill pattern will be stronger, but require more time and material to print. It is recommended to find a balance of these variables without compromising strength.

Cost ⟵ vs. ⟶ Structural Integrity

| Sparse | Sparse Double-Dense | Hexagonal | Solid |
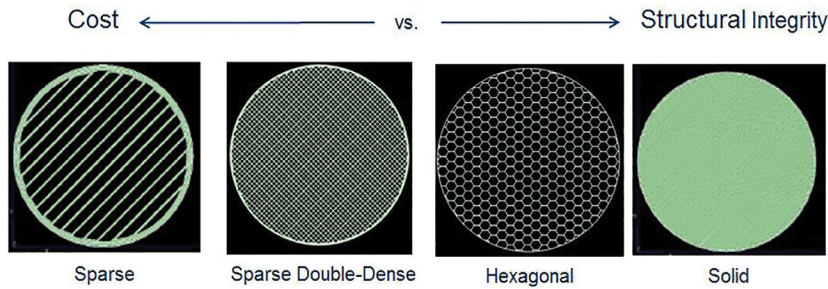
To generate a toolpath, select **Toolpaths**, then **Setup**.

FDM FOR COMPOSITE TOOLING 2.0

—
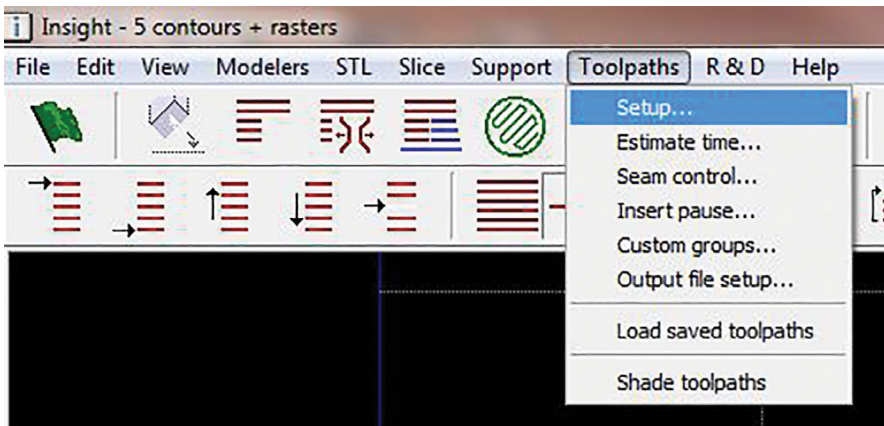
The following menu will appear on the right side of the screen and contains the relevant toolpath information. This menu lets the user change various aspects of the toolpath such as contour width, number of contours and fill pattern. Custom toolpaths can be created by selecting the icon indicated below.

FDM FOR COMPOSITE TOOLING 2.0

The advanced toolpath menu will appear in the middle of the screen. This menu will allow the user to specify various parameters of the toolpaths.

NOTE The tool path settings that are auto populated are recommended. Although contour and raster widths can be adjusted slightly to fill in small gaps and optimize fill at specific layers, default settings are recommended for optimal quality. Additionally, it is strongly recommend to NOT adjust shrink factors.

Select the green check to confirm the changes.

Shade toolpaths

Toolpaths can be viewed from the top orientation. The green line represents the centerline of the bead. In many cases, it is necessary to view the entire bead profile to look for porosity or poor bead contact. Right click on the work space and select *Shade toolpaths* to view the bead profile.
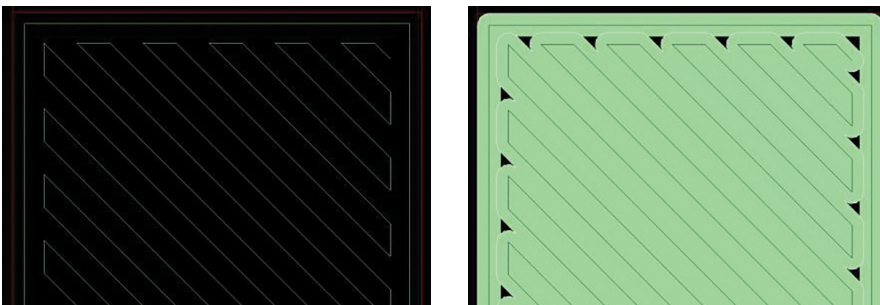
Figure 8-6: Before and after toolpaths have been shaded.

FDM FOR COMPOSITE TOOLING 2.0

Time estimation

Insight lets the user estimate the print time and material use after the support and toolpaths have been generated. To estimate the build time, click on *Toolpaths*, then *Estimate time*.
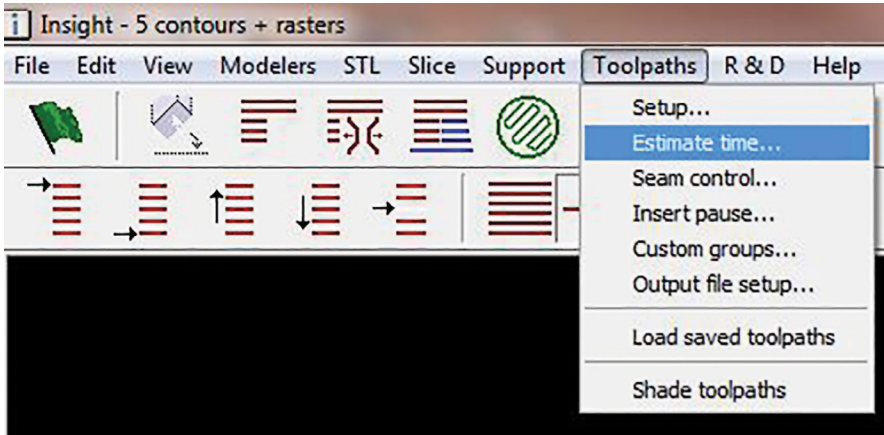
The following menu will appear.
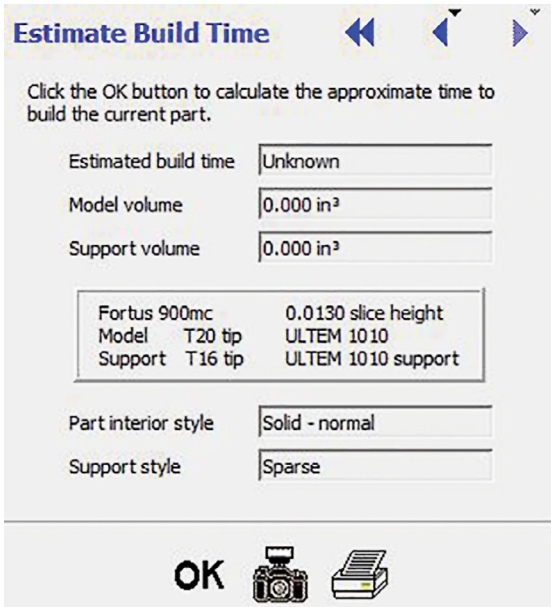
Select *OK* to run the estimation. The build time, model volume and support volume is displayed. A standard canister of FDM thermoplastic contains 92 cubic inches of material, which is helpful for estimating the amount of canisters required. Stratasys also offers 500 in³ (8,194 cm³) and 184 in³ (3,015 cm³) canister sizes for specific materials which can be helpful for larger tool builds.

stratasys

FDM FOR COMPOSITE TOOLING 2.0

Stabilizing walls

Vibrational forces within the machine can affect large, thin parts and may lead to dimensional inaccuracies. Adding stabilizing walls can prevent this. Stabilizing walls are sacrificial support columns, made out of model material, that help brace and anchor the part to the build sheet. Insight enables customization of stabilizing walls and the user can select which layer they will print up to.
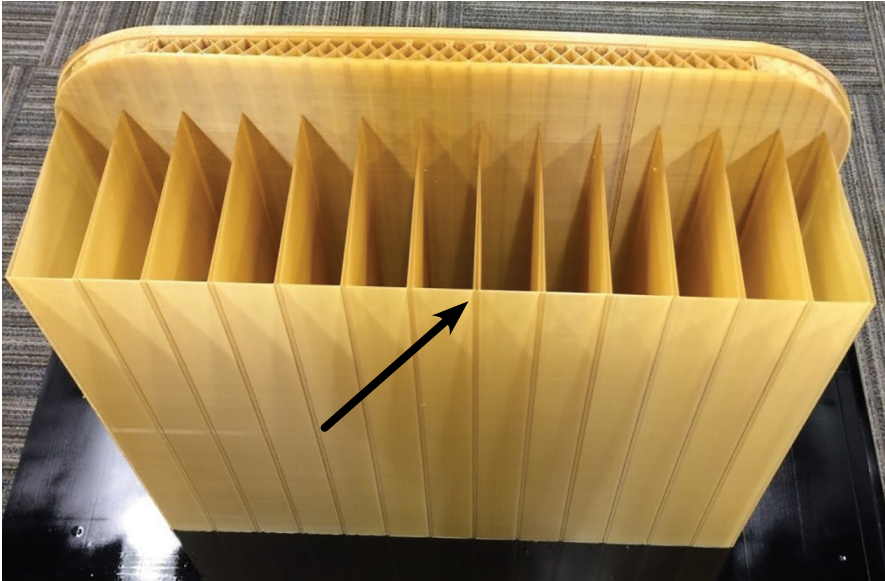
Figure 8-7: Composite tool printed with stabilizing walls indicated by the black arrow.

To add a stabilizer wall to a part, select **Support, Stabilize wall**.
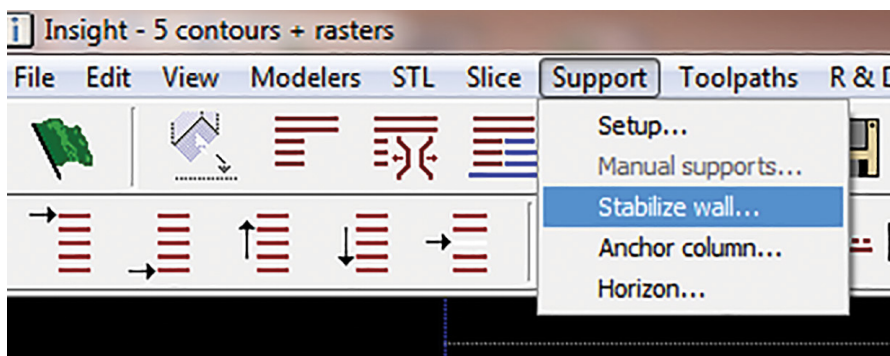
FDM FOR COMPOSITE TOOLING 2.0

The following menu will appear on the right side of the screen. This menu allows the user to select the various features of the stabilizer wall such as *Separation*, *Contact* interval and *Layer interval*.
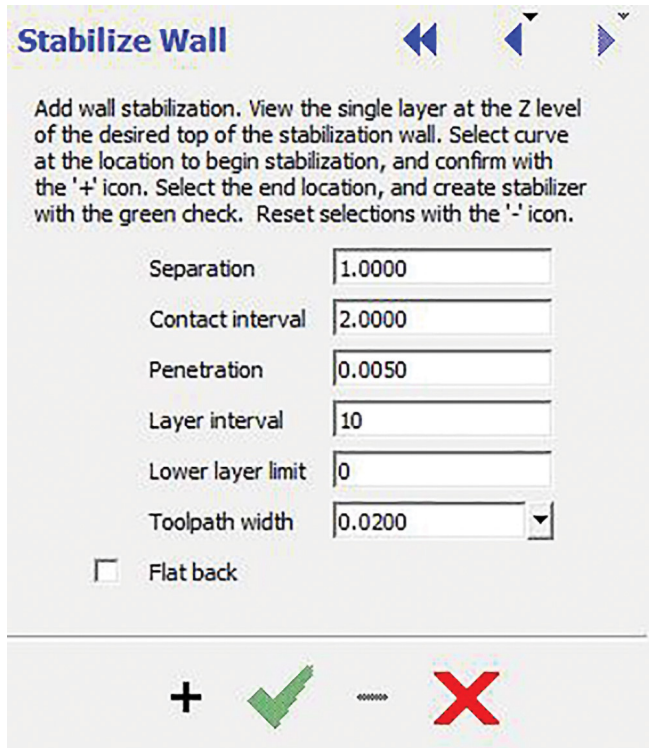
Once the features of the stabilizer wall have been established, select the layer that the stabilizer walls will go up to by entering a layer number. In the example below, the walls will reach layer 500 (see the red circle).
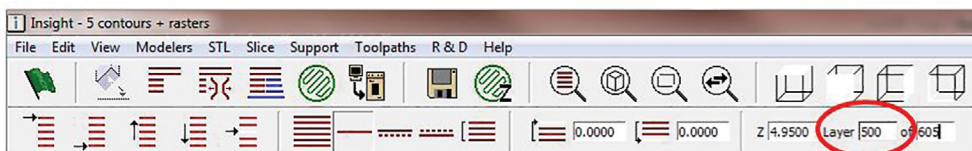
Select a point near the edge of the part as indicated below by the circled arrow. This will be the starting point of the stabilizer wall.

stratasys
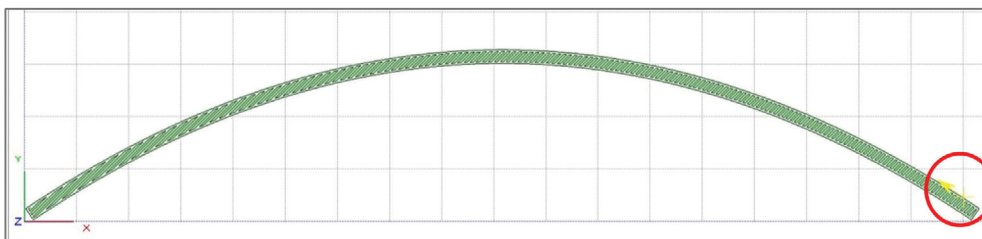
FDM FOR COMPOSITE TOOLING 2.0

—

Click on the "plus" icon to establish the first contact point of the stabilizer wall. Select the second point on the part to establish the end of the stabilizer wall. Then click on the green check mark.
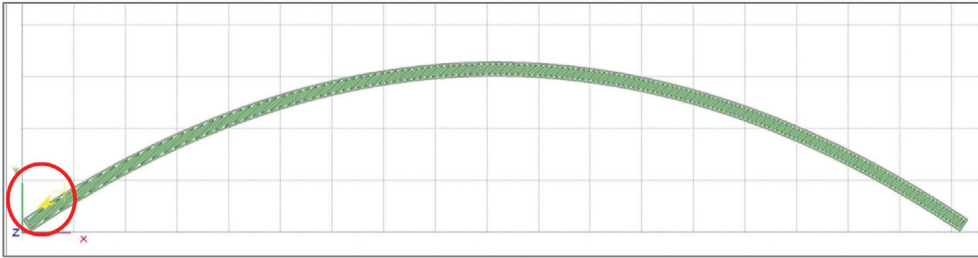
Select the green check mark. The stabilizer wall will appear and should look similar to the image below. The spacing and number of contact points may be different based on the chosen features.
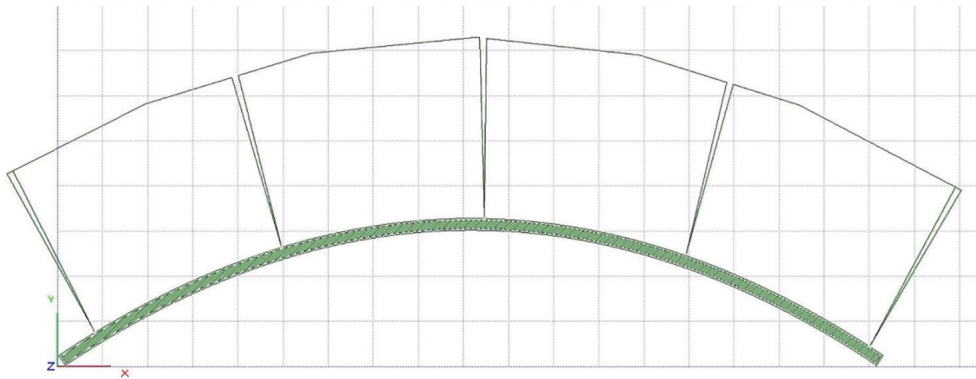
Seam control

When the printer applies a bead of material, the place where it starts and stops is called the seam. In certain cases the seam can cause a slight blemish on the surface of the part, which can lead to an unacceptable composite tool surface. Solve this by moving the seam to a non-critical surface (typically the back of the tool or at a corner). The blue arrow in the figure below indicates the current location of the seam and the red arrow indicates where the seam will be moved to.
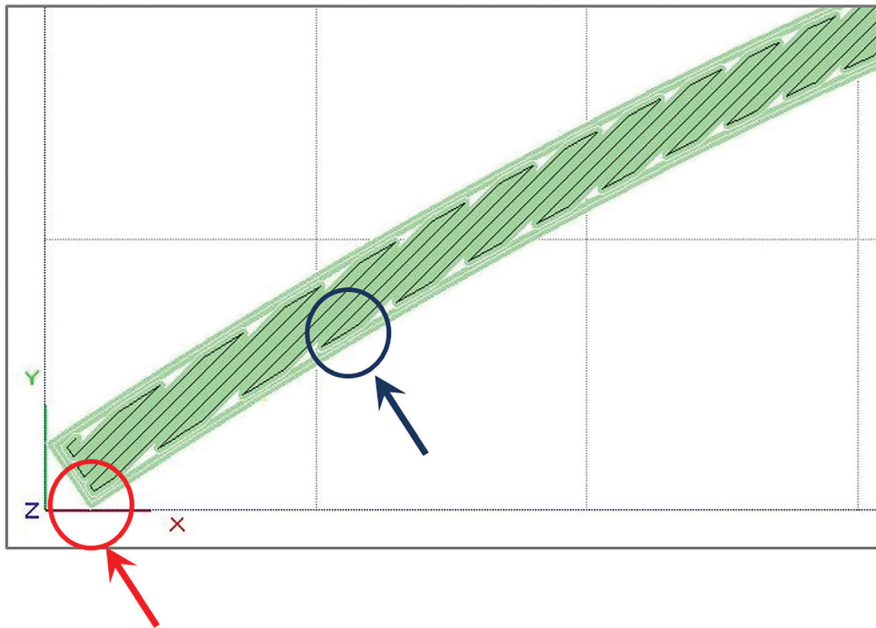
1. Move the seam by selecting **Toolpaths**, **Seam Control**.
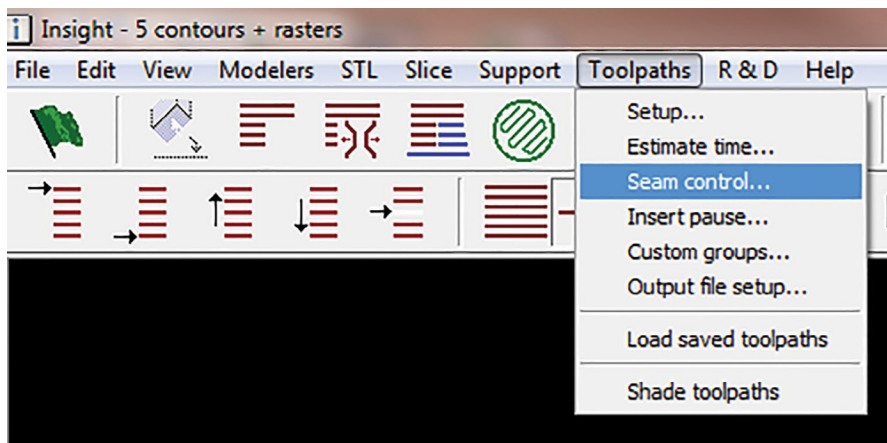
Select a point in space outside of the part to ensure that the seam on all layers is at the same spot. The red arrow below indicates where to select if the seam is to be placed on the corner.
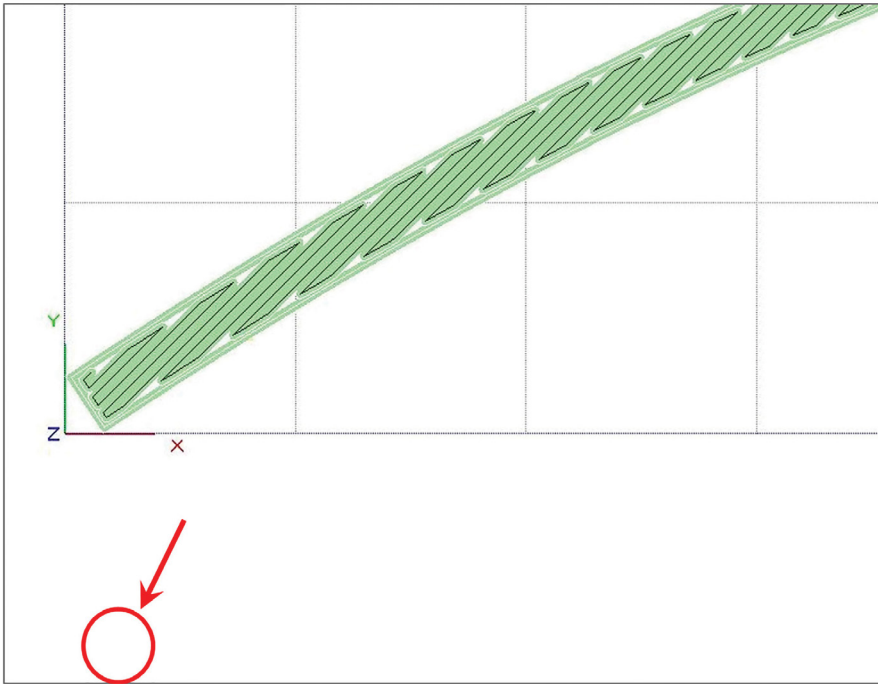
Select **Align To Nearest** for **Seam placement method** under the menu on the right side of the screen.

—
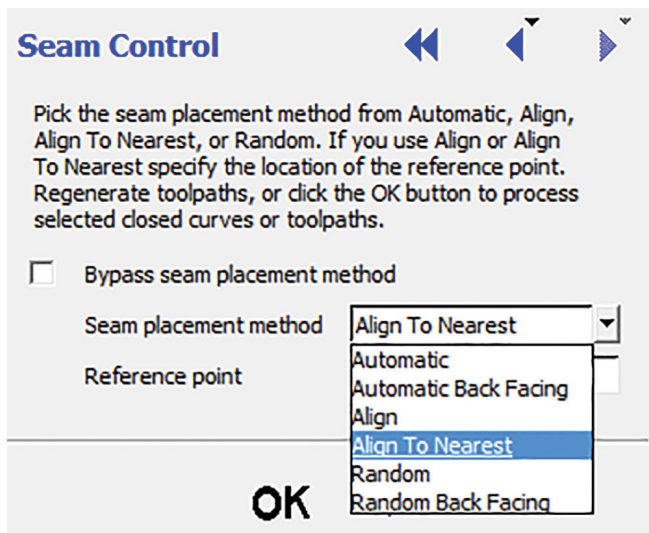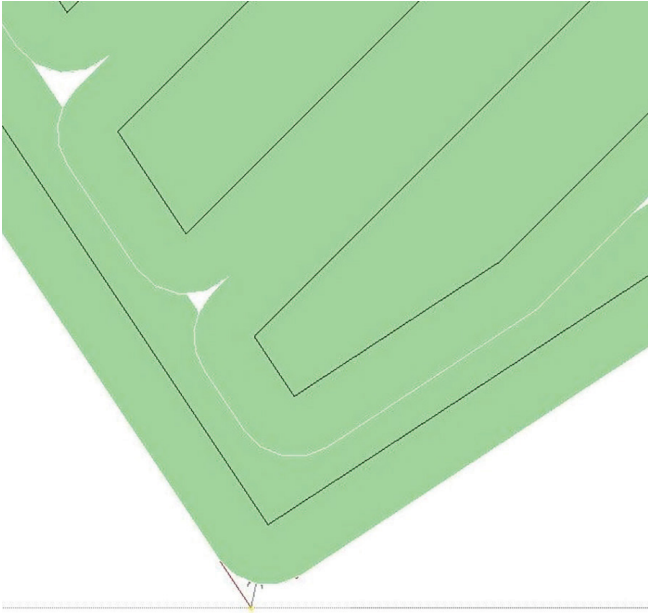
Using anchor columns to secure a part to the build sheet

To build a tool, a Fortus 3D Printer first applies a layer of model material, followed by several layers of support. The tool is then built upon that base of support material. High-temperature thermoplastics, such as ULTEM 9085 and 1010 resins, are susceptible to thermal shrinkage. In certain instances, parts can delaminate from the build sheet while building. This phenomenon is more prevalent among large, flat parts such as the bulkhead tool below.
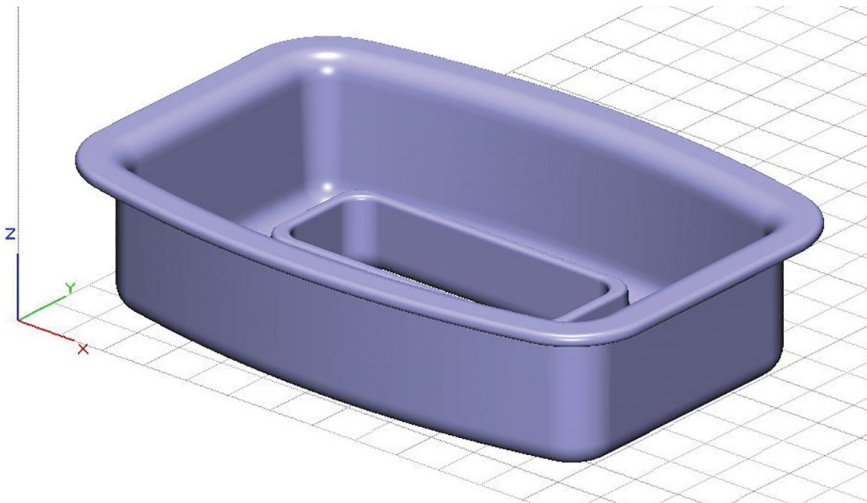
FDM FOR COMPOSITE TOOLING 2.0

—

The best method to prevent parts from delaminating is to add anchor columns roughly every 2-3 inches (50-76 mm) around the exterior edge of the part and at all corners. An anchor column is a column of model material that directly connects the build sheet to the part.
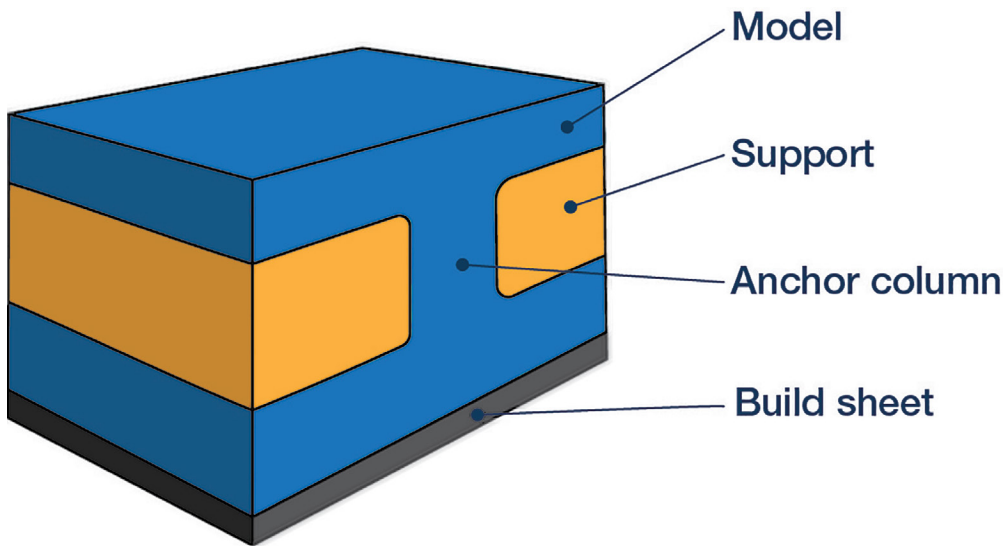
Figure 8-8: Cross-sectional view of an anchor column.

1. To add an anchor column, begin by slicing the part and selecting the top view.

2. Select the bottom layer of the part.

3. Select **Support, Anchor Column**
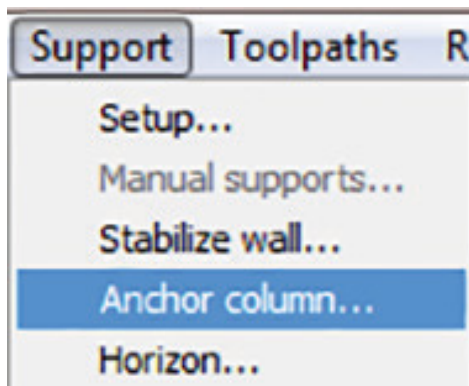
4. Select the diameter of the anchor column. 0.1 inch (2.54 mm) is the default and does an adequate job of securing the part to the build sheet. Make sure that the top and bottom sizes match, otherwise the column will have a conical shape.
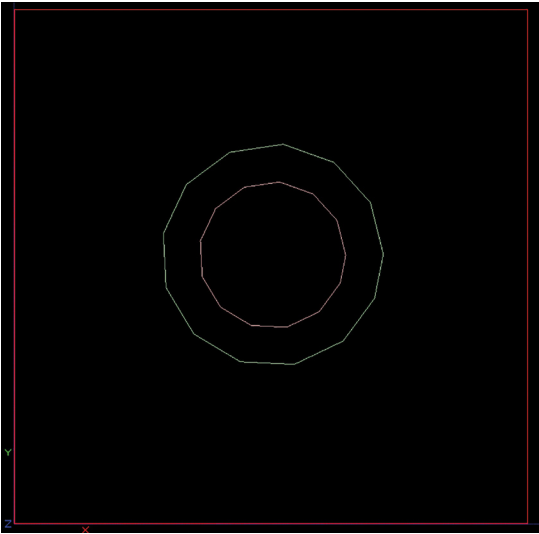

5. Select the location of the anchor column. A white circle will appear. Click OK. Note: The circle will disappear when the user clicks OK, but it has not been deleted.
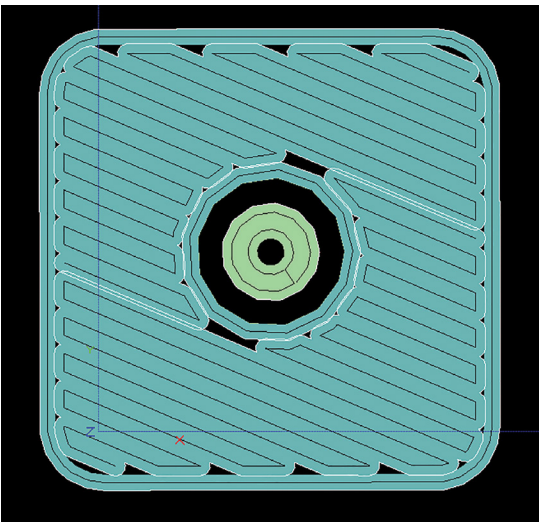
—
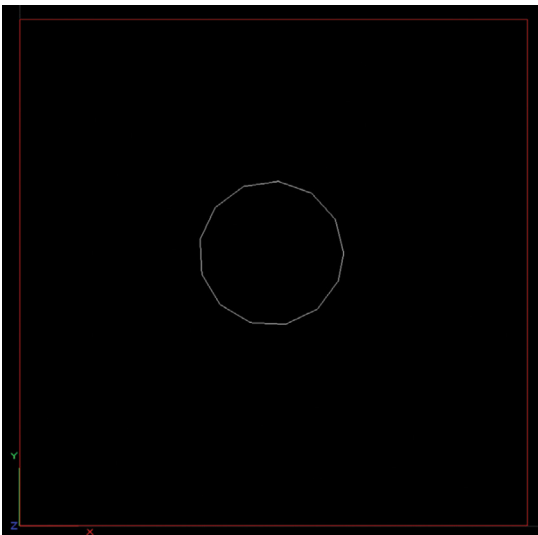
6. Generate support and toolpaths.

7. Shading the toolpaths will show that an anchor column has now been incorporated into the part.

FDM FOR COMPOSITE TOOLING 2.0

—

APPENDIX A – THERMAL WELDING EXAMPLE PROCEDURES

Large FDM parts can be bonded together with either an adhesive or by hot-air welding. Hot-air welding fuses thermoplastic parts together using heated air to melt a plastic filament in the joint between the parts. If done properly, the bond can be as strong as an adhesive bond. A heating element is required to melt the joint and the filament. There are many thermal welding devices; the following information shows how to use a Leister Hot Jet S hot-air hand tool.

Figure A-1: Leister Hot Jet S hot-air hand tool.

This particular tool requires a soldering nozzle to concentrate the heat. This example uses a model 107.148 (0.85 inch diameter) 0.11 x 0.06 inch, oval soldering nozzle, available on the Leister website.

Figure A-2: Soldering nozzle attachment.

FDM FOR COMPOSITE TOOLING 2.0

—

The temperature settings will vary based on the material being used. For ULTEM 1010 resin, set the fan (black knob) to 3 and heat (red knob) to 6.

Figure A-3: Fan and temperature settings for ULTEM 1010 resin if using a Leister Hot Jet S hot-air hand tool.

Designing a V-groove into the joint will increase the mating surface area. Stack layers of filament to fill the groove.
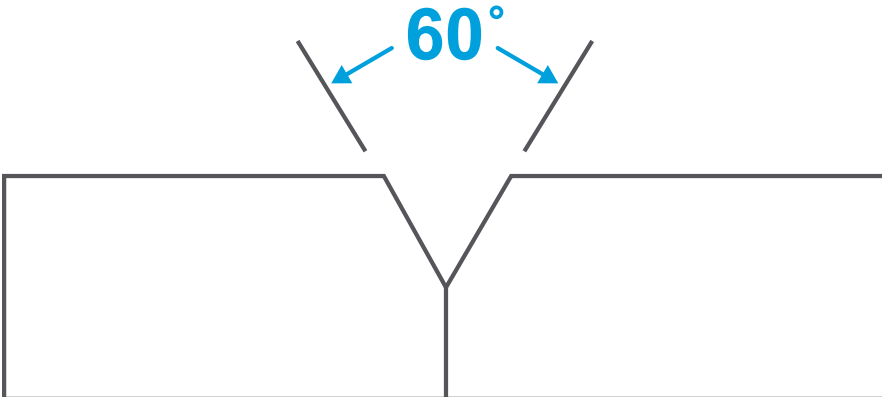

Figure A-4: V-groove channel for bonding two sections together.

Use a stick of filament from the canister to weld the two sections together. The hot-air welder should simultaneously heat the filament and the substrate until both become soft. Maximum bond depth will approximately equal the diameter of the welding filament

Figure A-5: Filament welding two sections together.

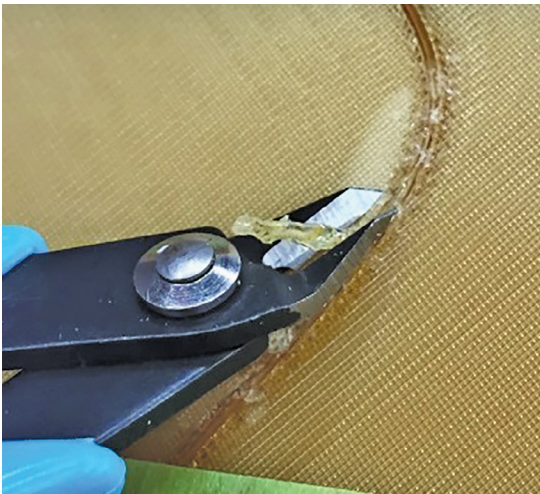The filament should be firmly secured to the bond. The figure below is an example of a poor bond because the pliers can easily remove it. When completed, sand the surface smooth.

Figure A-6: Example of a poor bond because the filament can be easily removed.

FDM FOR COMPOSITE TOOLING 2.0

——

APPENDIX B – SEALING PROCEDURES

Epoxy Sealer Application

Gather the following tools and materials:

- Dual-action orbital sander (electric or compressed air)
- Sandpaper in the following grits: 120, 220, 320, 400, 600 and 800
- Epoxy sealer
- IPA, acetone
- Squeegee or paper towel
- Clean, lint-free rags
- Oven that can accommodate the size of the tool

Procedure:

Note – The result of the process detailed below should be a smooth surface that provides vacuum integrity. The goal of this process is to first fill the voids in the tool surface that are created from the layered build process and then to simply remove any peaks created from the layering process. Ultimately the processor must be careful not to change to the tool surface geometry through excessive abrasion.

1. Set the oven to 200 °F (93 °C).
2. Wipe the tool using IPA or acetone (preferred) to remove dust and contaminants.
3. Begin by lightly sanding the tool using 120-grit sandpaper. **This abrasion is not meant to remove the layer lines**, but rather roughs up the surface and allows for a good bond between the epoxy and the tool.

Figure B-1: Initial sanding of tool with 120-grit sand paper.

stratasys

—

4. Wipe away excess dust and wipe the tool with IPA or acetone.

5. Place the tool in the oven set at 200 °F (93 °C) for 10-15 minutes. This helps increase the penetration and infiltration of the epoxy sealer.

6. Thoroughly mix the epoxy according to the manufacturer's recommendations.

Figure B-2: Weighing the epoxy.

Figure B-3: Mixing the epoxy.

7. Remove the tool from the oven and allow it to rest at room temperature for 1-5 minutes.

8. Apply the epoxy by pouring it onto the tool surface. Wipe the tool using a squeegee or paper towel to create a thin film across the entire surface.

9. The goal here is to fill in the valleys and or voids on the lay-up surface.
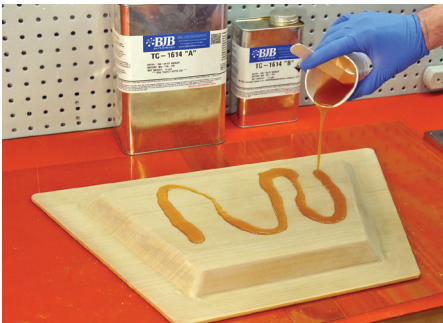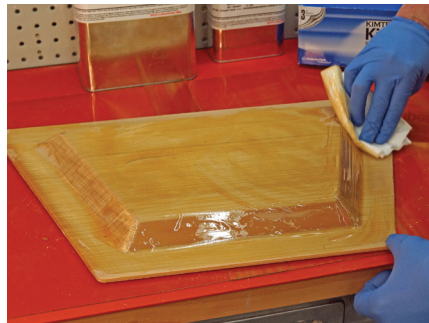

Figure B-4: Pouring epoxy on the tool.

Figure B-5: Wiping the tool.

10. Place the tool into the oven and cure the epoxy per the manufacturer's recommended procedure.

11. Remove the tool and let it cool until it is cool enough to touch (approximately 30 minutes).

12. Sand the tool using 120-grit sandpaper.

13. Apply a second coat of epoxy by repeating Steps 4-10.

14. Sand the tool using 120-grit sandpaper.

15. Wipe the tool using acetone to remove dust and contaminants.

16. Sand the tool using the following progressively finer sandpaper grits: 220 and 320. Wipe the dust off with a dry cloth between each sanding step.

Figure B-6: Sanding the tool.

17. Polish the tool by wet sanding with 400 and 600 grit sandpaper. Wet sanding with 800 grit can also be performed if surface roughness below 16 µin (0.4 µm) Ra is desired.

Figure B-7: Polishing the tool.
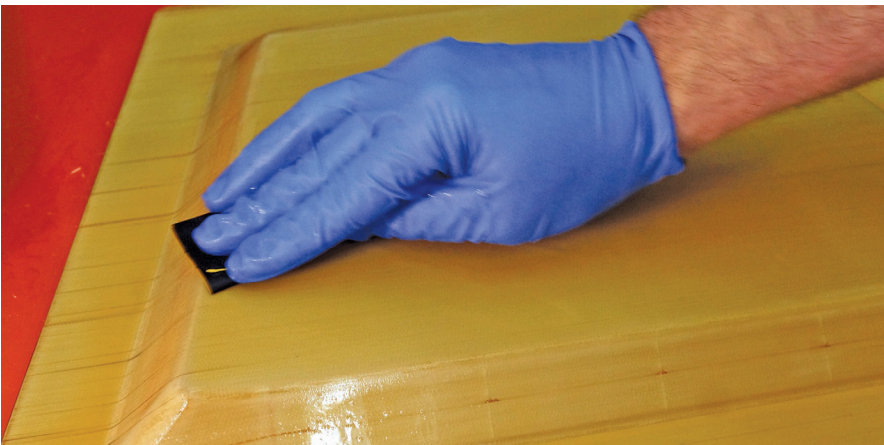
—

18. Check the surface roughness using a profilometer (optional).

Figure B-8: Checking surface roughness with a profilometer.

FDM FOR COMPOSITE TOOLING 2.0

—

APPENDIX C – COMMON TERMS

The following is a list of common terms related to FDM composite lay-up tooling.

additive manufacturing: The process of creating objects from a CAD file by depositing layers of material. Also known as "3D printing."

bead width: The width of the thermoplastic bead. This is not always the same as the tip size.

build chamber: The internal portion of the 3D printer where the part is built.

build platen: The platform inside of the build chamber that the material is deposited onto.

build sheet: A thin, disposable plastic sheet that is attached to the build platen to ensure the part doesn't tip over during the build process.

build volume: The maximum dimensions of the space inside a 3D printer where the part is built.

composite tool The tool used to layup, form, and cure composite materials in the production of composite parts. In this guide it is used synonymously with the terms layup tool, mold, mandrel and die.

Control Center™: Software that allows the user to send a part that has been processed using Insight Software, to the printer.

contours and rasters: Terms used to describe the two types of toolpaths that make up FDM parts. Contours outline the periphery of the part and rasters fill the internal space between contours.
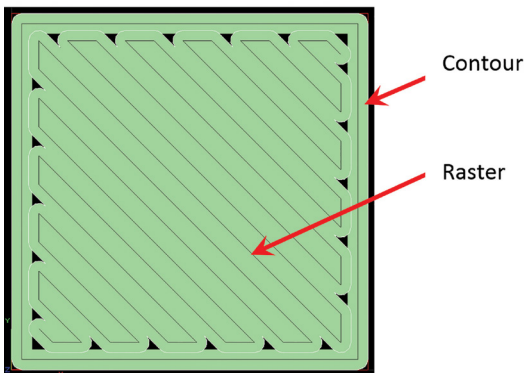

Figure C-1: Contour and rasters.

cure cycle: The process of curing the resin system within a composite laminate to create a rigid structure. Cure cycles can vary based on manufacturing recommendations. Many require heat and pressure from an autoclave.

extruder head: The assembly on the 3D printer's X-Y gantry that contains the extruding tips, liquefiers, drive blocks, and hardware necessary for proper deposition of model and support material.

filament: The form of the thermoplastic and support material as it enters the 3D printer.

stratasys

FDM FOR COMPOSITE TOOLING 2.0

Fortus 3D Printers: Production Series 3D Printers manufactured by Stratasys and driven by FDM Technology. The Fortus 900mc is the largest FDM 3D Printer, with a build volume of 3 feet wide x 2 feet deep x 3 feet high (914 x 610 x 914 mm). The second-largest offering in this family is the Fortus 450mc, with a build volume of 16 inches wide x 14 inches deep x 16 inches high (406 x 356 x 406 mm).

Figure C-2: Fortus 900mc 3D Printer.

Figure C-2: Fortus 450mc 3D Printer.

FDM Technology: One type of additive manufacturing that 3D prints a part by applying beads of thermoplastic in successive layers.

Insight software: Software used to specify the build parameters when a part will be produced in a Fortus 3D Printer.

liquefier: The part of the extruder head that liquefies the thermoplastic filament before being deposited.

material canister: A container that houses the material used by a Fortus 3D Printer.

model material: Any thermoplastic extruded in an FDM 3D printer that forms the object.

porosity: The quality that results from air pockets and voids that occur between extruded beads as well as the designed-in gaps between build paths in a sparse built tool (FDM tools can be built solid or with varying degrees of porosity, by design). Also see "Sparse Build."
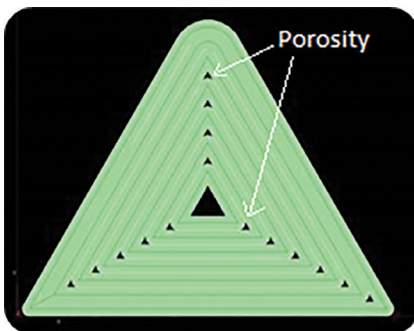

Figure C-4: Natural gaps between toolpaths resulting in porosity.

stratasys

post processing: Processes required after the tool is printed to prepare it for composite layup.

self-supporting angle: Angles on part features greater than 45°, relative to the build platform, that do not require support material.

Figure C-5: Self-supporting angles (left) require no support material.

slice height and build tips: Slice height defines the layer thickness of the part being fabricated. The build tip is the material-dispensing nozzle of the extruder. The slice height is related to the build tip size.

slicing: The act of dividing the .stl file into layers or "slices."

sparse build: A particular type of FDM build construction characterized by a sparse internal fill patterns intended for light weight, reduced build time and minimal material use. See the image below for a comparison between sparse build and other FDM fill patterns.

Sparse   Sparse Double-Dense   Hexagonal   Solid
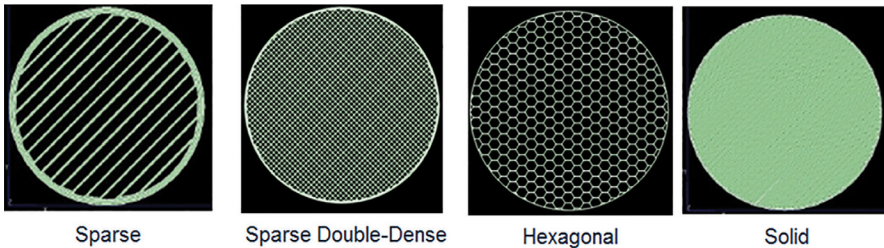
Figure C-6: Various fill patterns.

stair-stepping: A phenomenon where the slice height will create a stair-like pattern on curved surfaces of the part. This is due to the geometrical constraints of the bead profile. Stair-stepping can be minimized by changing build orientation, or decreasing the slice height.
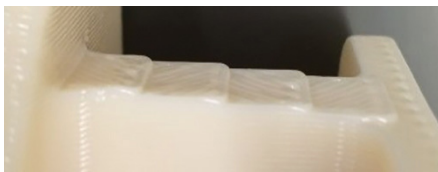

Figure C-7: Stair-stepping with a large slice height.

Figure C-8: Stair-stepping with a small slice height.

—

tip: A replaceable nozzle in the extruder head from which the material is deposited. The tip size will impact the size and profile of the bead.

Figure C-9: Build tip.

tip size: The diameter of the extrusion tip.

tool sealing: The process of applying a secondary material (adhesive, film or similar) to a printed tool to provide a smooth, continuous surface upon which to lay up material and provide vacuum integrity.

traditional tooling: Tools made with conventional materials such as machined metal or foam.

trim tools: Tools used to trim the excess material from a composite part after cure.

ULTEM 1010 Resin: Commonly known as polyetherimide (PEI), this is a high-performance thermoplastic, developed by SABIC, that can be used to build FDM parts offering excellent strength and thermal stability.

FDM FOR COMPOSITE TOOLING 2.0

—

APPENDIX D – ULTEM 1010 RESIN COMPOSITE TOOLING REPAIR PROCEDURE

Material Required:

- Digital Scale
- BJB TC-1614 two part epoxy
- Plastic or wax mixing cups (Dixie cups)
- Stirring sticks
- Lint-free towels (Kimwipes)
- Oven
- Plastic filament welder
- Cutting wheel
- Angle grinder
- 80-grit grinding disks
- Sandpaper: 180, 220, 320 grit
- Wet sandpaper: 400, 600 grit

Step 1: Measure and mark the damaged area of the part that will be cut away and replaced. Print the replacement part using the measurements taken.
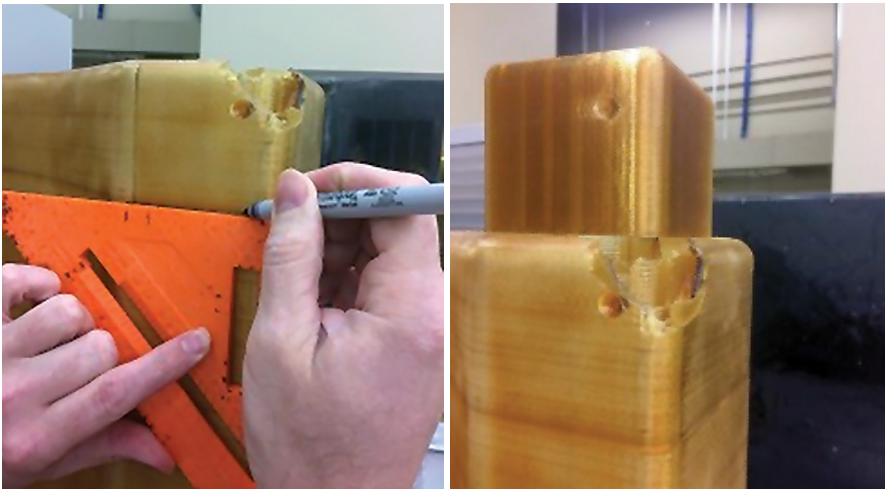
Step 2: Using the cutting wheel cut away the damaged area that was marked for removal. Cut on the inside of the marked lines so as to not cut away too much of the part.

Step 3: Use the Angle grinder with the 80 grit sanding disk to smooth out the edges of the part. This will give you a good surface to mount the replacement part too.
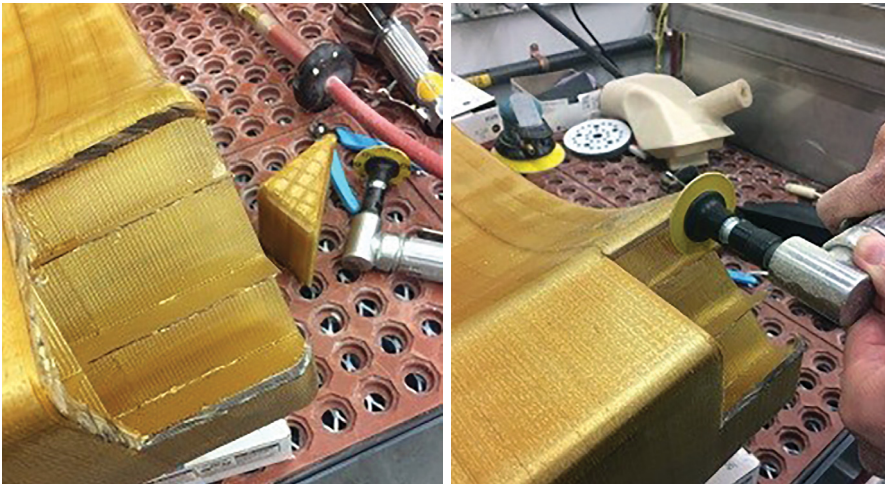
**Step 4:** Fit check the replacement part. Make any adjustment to ensure the part fits and will be flush with all critical surfaces.

Step 5: Align the replacement part and secure it in place with clamps. If unable to use clamps use tape to secure its position. Using the hot air welder and filament, spot weld all corners of the replacement part. Remove clamps or tape from the tool. Re-heat the spot welds to make any final adjustments to ensure a flush fit for all critical surfaces.
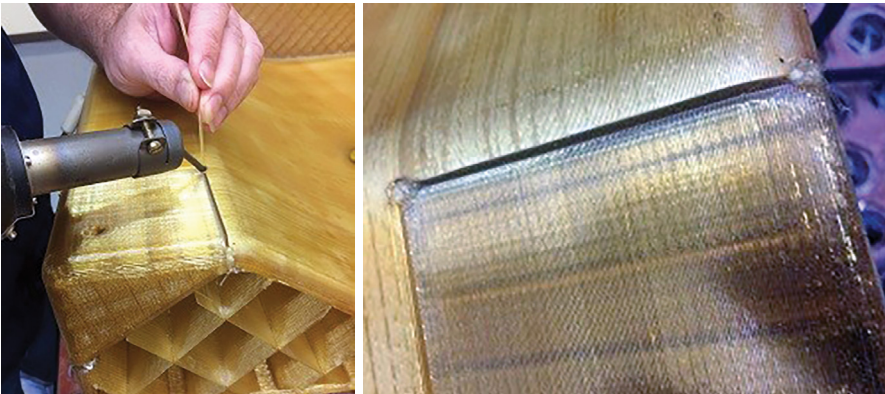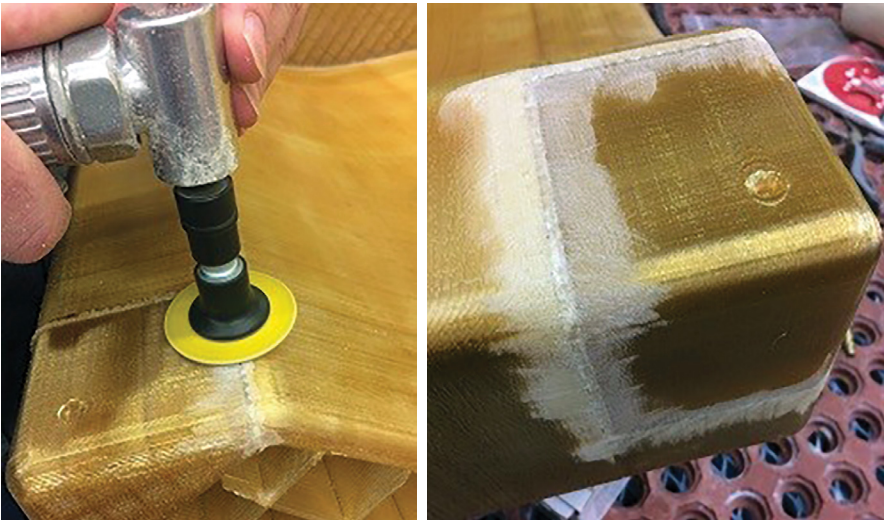
Step 6: After final adjustments have been made weld the rest of the replacement part in place.
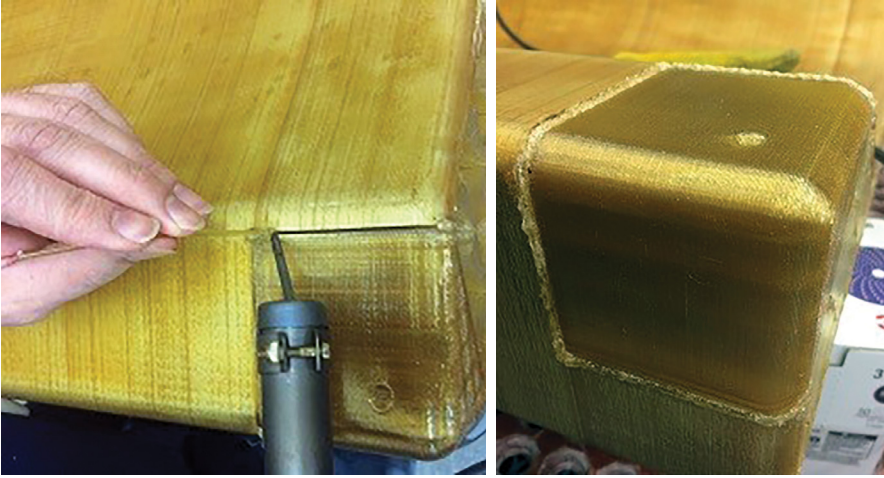
Step 7: Using the angle grinder with the 80 grit sanding disk sand down the welds flush with the rest of the tool's surface. Be careful not to over-grind the welds and create low spots in the overall surface of the part.

Step 8: DA (dual-action) sand the replacement part and weld with 120 grit sandpaper overlapping two to three inches on the rest of the composite tool.
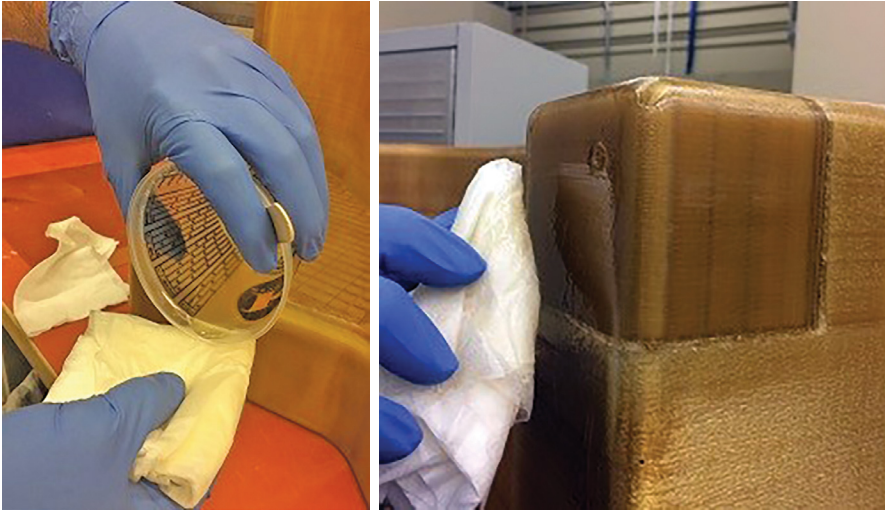

Step 9: Using the manufacturer's specifications of 5:1 (5 parts A to 1 part B) mix enough BJB TC-1614 epoxy to generously coat desired sanded surfaces.
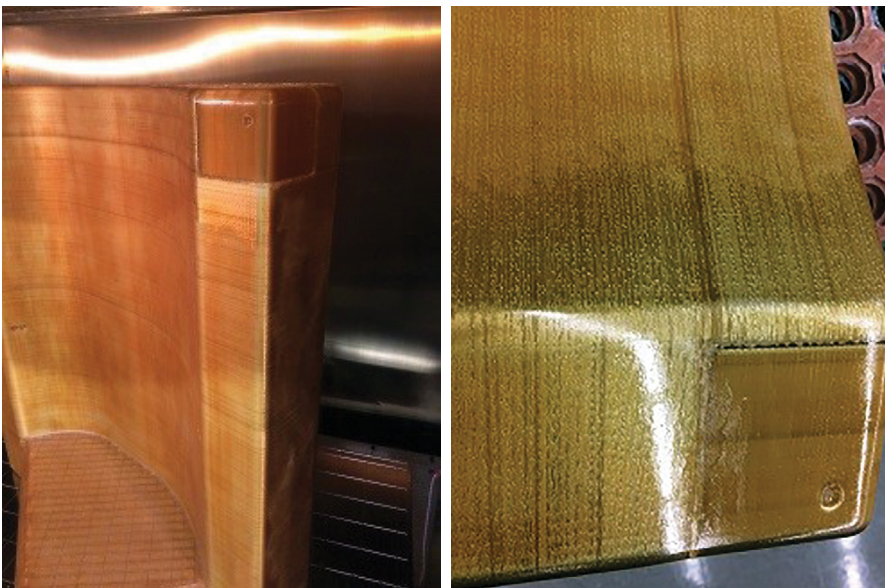
Step 10: Evenly coat the sanded surface with the mixed epoxy. Let the epoxy soak in to the part for roughly 5 minutes and wipe excess epoxy away with the Kimwipes.

Step 11: Place the composite tool into an oven and bake at 200° F (93° C) for 2-3 hours or until the epoxy is completely cured.

FDM FOR COMPOSITE TOOLING 2.0

—

Step 12: Remove part from the oven and allow it to completely cool to room temperature. Repeat steps 8-11 until the layer lines and stepping are completely filled with epoxy. Under normal circumstances this will take 2 to 3 coats.

Step 13: DA or hand sand all epoxy-coated surfaces with 220, 320 grit sandpaper. Now wet sand by hand with 400 and 600 grit wet sandpaper. Following this procedure will achieve a *Ra value of 32 µin (0.8 µm) or below, dependent on the overall ability and skill of the finisher.

*Ra is calculated as the Roughness Average of a surface's measured microscopic peaks and valleys. The industry standard Ra for molds are between 32 and 64 µin (0.8 and 1.6 µm).

FDM FOR COMPOSITE TOOLING 2.0

APPENDIX E – BUILD READINESS CHECKLIST

Preparation

☐ Material selected based on cure temperature requirements (ULTEM 1010 resin is recommended for nearly all layup tooling applications.)

☐ Verify anticipated cure pressure and vacuum bagging method – consider for tool style and build construction

☐ Build orientation established (to minimize material use, build time and, when important, stair-stepping)

☐ CTE impacts considered for the design

☐ Sanding and sealing method and material determined

☐ Required tool life is generally understood (10s versus 100s of parts)

Design

☐ Trim lines and non-essential features removed

☐ Determine desired build style (shell, sparse, hybrid)

☐ Self-supporting angles incorporated into any overhangs or internal features

☐ All sharp corners and edges rounded (particularly for envelope bagging)

☐ CTE scaling factor applied (when appropriate)

☐ Corresponding trim tools designed for final composite part

☐ For large tools, segmentation and joining method established and required features incorporated

Insight Processing

☐ Material, slice height and bead width selected

☐ Tool properly oriented

☐ Tool sliced

☐ Support generated

☐ Toolpaths generated

☐ Stabilizing walls and/or anchor pins added (if necessary)

☐ Seams moved away from layup surface

Post-Processing

☐ Tool sanded and sealed (as required)

☐ Cure temperature-compatible tool sealing material selected

☐ Tool joining procedures prepared (if multiple pieces)

☐ Mold release selected – water-based release agents are recommended

stratasys

STRATASYS.COM

ISO 9001:2008 Certified