FDM for Composite Tooling

DESIGN GUIDE

Photo courtesy of Aurora Flight Sciences

FDM for Composite Tooling

DESIGN GUIDE

FDM for Composite Tooling

DESIGN GUIDE

SECTION 1 - INTRODUCTION AND BACKGROUND

Application Overview

FDM® (fused deposition modeling) is becoming the technology of choice for rapid production of high-temperature (>350 °F [177 °C]), low-volume, composite lay-up and repair tools, as well as for moderate-temperature (<325 °F [163° 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.

FDM lay-up tools have many similar design and use considerations as traditional tooling, particularly those with higher coefficients of thermal expansion (CTE), such as aluminum and epoxy tooling board, although the technology provides greater design capability and freedom. To ensure high-quality surface finish and vacuum integrity, post-processing of FDM tools is typically required. 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. In fact, better than 16 µin. Ra can be consistently achieved. Sealing can be performed using a range of materials depending on the specific application, including high-temperature epoxy paste and film adhesives, as well as adhesive-backed FEP films and similar materials. Once the part is sealed, common mold-release agents can be applied in preparation for lay-up. Water-based released agents are recommended.

FDM tools can be vacuum bagged using either surface or envelope bagging methods. Envelope bagging tends to be preferred due to the lighter weight and more optimal sizes (no heavy support structure required). FDM tools are effective for autoclave, oven and electric heat-blanket curing and can be used with cure cycle parameters exceeding 350 °F and 100 psig, depending on the selected material.

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 (Section 3 and 8)
FDM Build (Section 3)	Post-Processing (Section 4)	Composite Part Fabrication (Section 4)

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, 100 psig)
- Provides low-hassle sacrificial and wash-out solutions for complex, trapped-tooling applications
- Permits trouble-free design changes and iteration

Background and Purpose

Traditional manufacturing methods for high-performance, fiber-reinforced polymer matrix composite structures require hard tooling for the mold or mandrel that dictates the shape 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 cure temperature required for aerospace and similar high-performance structures, and the absence of design knowledge and guidance.

Regarding materials limitations, FDM ABS-M30/ASA, polycarbonate (PC), and ULTEM™ 9085 resin are effective up to 180 °F, 270 °F, and 300 °F, 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 and pressures of 100 psig.

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

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

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 vworld-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 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. Support material is removed after the build.

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.

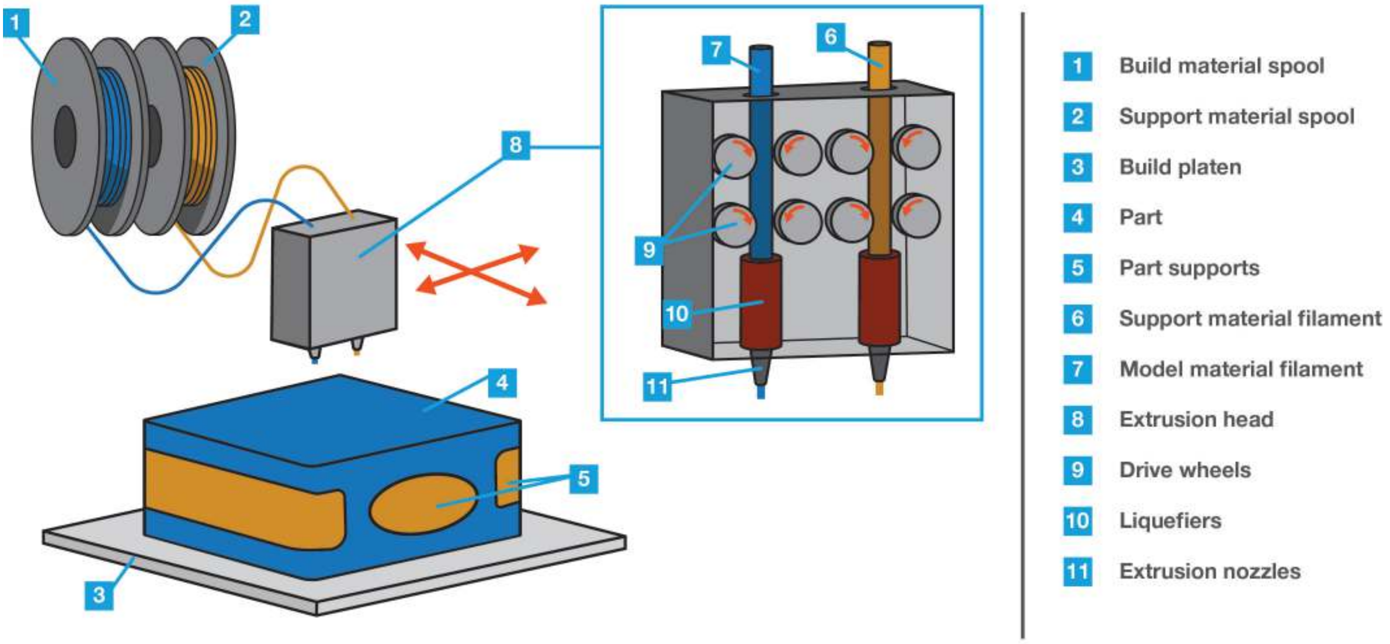


Figure 1-1: The main components of an FDM printer.

Application Best Fits

This application is most suitable in the following conditions:

- Lay-up and repair tools are required in days, not months
- Sacrificial tooling cures at moderate temperatures (<350 °F)
- 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 are also feasible
- Tool geometries can be adjusted to compensate for thermal expansion or benefit from higher CTE materials (e.g., male mandrels for increased part compaction)

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 below.

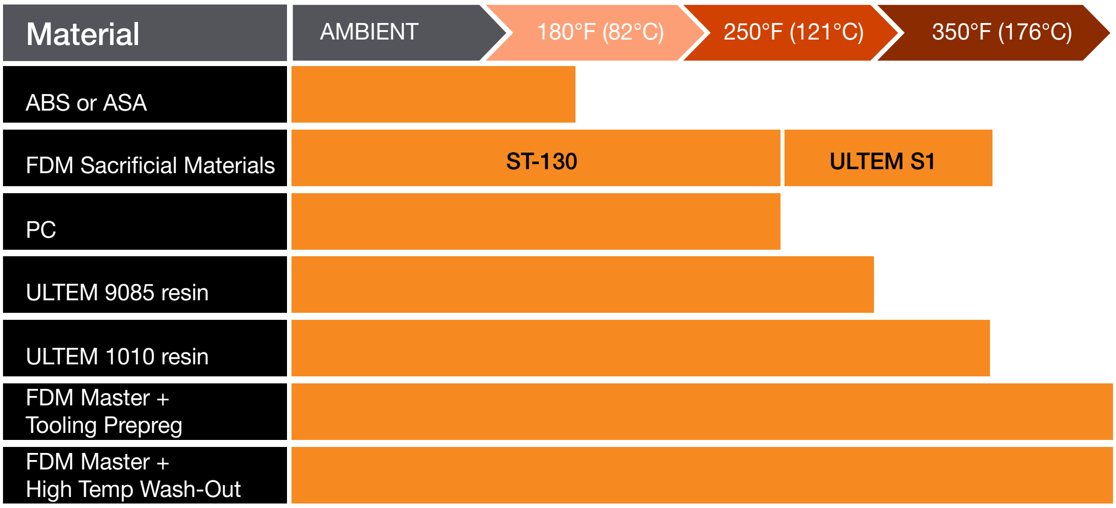


Figure 1-2: Approximate cure temperature capability for FDM tooling materials.

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-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.

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.

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.

Table 1-1: CTE comparison for FDM and traditional tooling materials.

FDM MATERIALS	μin / (in·°F)	μm / (m·°C)
ST-130 (SOLUBLE/ SACRIFICIAL)	59 (140-212 °F) 98 (212-266 °F)	106 (60-100 °C) 176 (100-130 °C)
ABS/ASA	49	88
PC	45	79
ULTEM 9085 RESIN	37	65
ULTEM 1010 RESIN	26	47
CONVENTIONAL TOOLING MATERIALS		
TOOLING BOARDS	20-40	36-72
ALUMINUM ALLOY (AL 6061-T6)	14	25
TOOL STEEL	6.5	12
CARBON/EPOXY	4.5	8
INVAR	0.7	1.2

Accuracy and Tolerances

FDM is capable of producing tools with accuracies of ±0.0035 inch or ±0.0015 inch/inch, 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 parts that require 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. 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. Additional information is provided in Section 3.

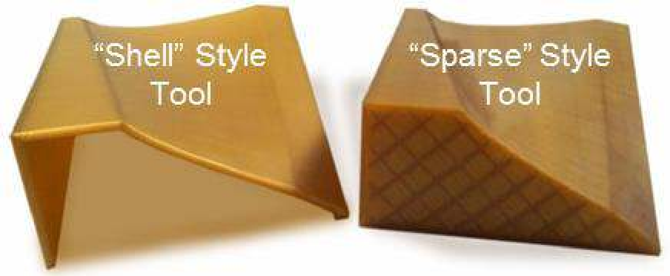


Figure 1-3: UAV Fan-blade tools showing examples of shell and sparse tools.

Shell-style tools are effective for most applications, able to withstand 100+ psig 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.

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.

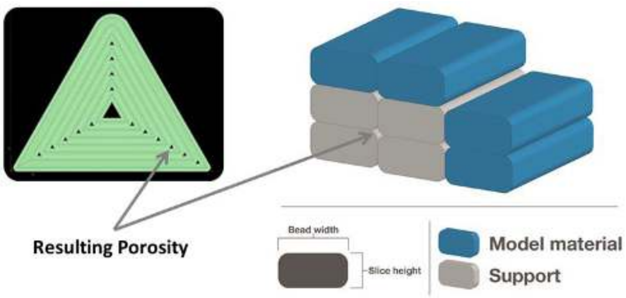


Figure 1-4: Top view of an example toolpath (left) and cross-section of bead profiles (right) showing inherent porosity in FDM parts.

Although requirements do vary somewhat across industries, a finish of 64 µinches Ra is generally considered acceptable. Using the standard procedure provided in Appendix B, a finish smoother than 16 µinches 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 paste 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 autoclave 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 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 materials can be found on www.sys-uk.com. Table 2-1 below provides guidance on the FDM materials most relevant for composite tooling applications.

Table 2-1: 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	• Moderate temp capability	• Higher CTE than ULTEM 1010 resin • Not suitable for 350 °F cure temps	367	186	37	--*	65	--*
PC	Lay-up tools for cure temps <270 °F	• 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) masters and patterns Trim/drill fixtures Other jigs and fixtures	• Lowest cost	• High CTE • Not suitable for even moderate cure temps	226	108	49	47	88	84
ASA	Low temp (<180 °F) masters and patterns Trim/drill fixtures Other jigs and fixtures	• Lowest cost	• High CTE • Not suitable for even moderate cure temps	226	108	49	46	88	82
ST-130	Sacrificial, wash-out tooling (<250 °F)	• Soluble material for trapped-tool applications	• Not suitable for high cure temps	269	132	59 (140-212 °F) 98 (212-266 °F)	--*	106 (60-100 °C) 176 (100-130 °C)	--*
ULTEM S1	Moderate temp (<330 °F) sacrificial tooling	• Higher temp capability vs. ST-130 • Can be embrittled with acetone	• Not soluble; must be manually removed	365	185	33	--*	59	--*

* - Denotes unavailable data.

SECTION 3 – TOOL DESIGN AND CONSTRUCTION

Design Considerations and Impact

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

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.

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 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.

Sparse-Style Tool

Although shell tends to be the most common tool-design approach, 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 small enough.

Figure 3-2: Test coupon for structural integrity testing of envelope-bagged, sparse style (sparse double dense in Insight), ULTEM 1010 resin. Raster spacing of 0.5 inch is shown.

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 fill pattern in combination with ULTEM 1010 resin only. Other patterns (e.g., hexagonal and custom fills) and materials will follow similar guidelines, but performance has yet to be verified. Results will also vary somewhat based on the specific tool geometry. It was determined experimentally that wall thickness (i.e., the number of contours) 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 (5 contours). The geometry of the test coupons is shown in Figure 3-2.

Table 3-1: Sparse double dense raster spacing for sparse tools when envelope bagging.

COMPACTION PRESSURE	WALL THICKNESS	MAXIMUM SPARSE SPACING	RELATIVE MATERIAL USE
VACUUM ONLY – 40 PSIG	0.1 inch	0.5 inch	1.0
<60 PSIG		0.25 inch	1.3
80 – 100 PSIG		0.1 inch	2.2

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 and the sparse raster spacing from 1.0 inch down to less than 0.08 inch. Results for coupons with 1.0 inch spacing are not shown as they had damage/crushing at even the lowest compaction pressures. All coupons were tested at 350 °F. All sparse construction refers to sparse double dense 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.

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.

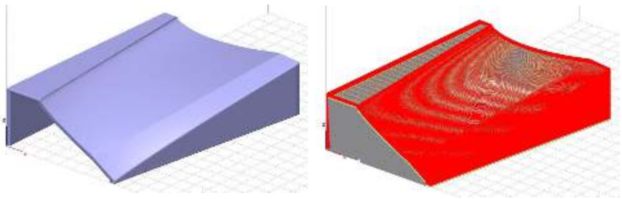


Figure 3-3: “Flat” build orientation.

Flat Build Orientation

A “flat” orientation, as shown in Figure 3-3, 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.

Sub-Optimal Vertical Build Orientation

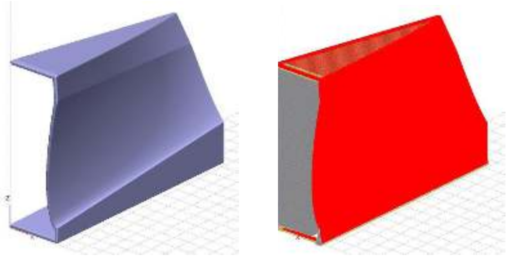


Figure 3-4: Sub-optimal “Vertical-A” build orientation.

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.

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.

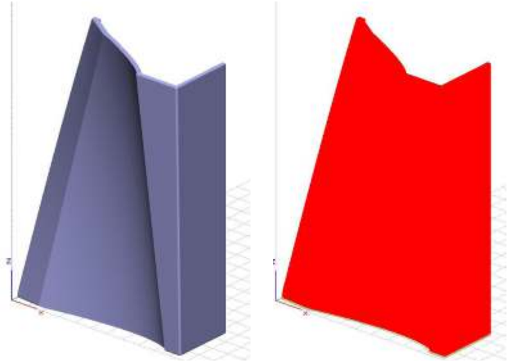


Figure 3-5: Optimal “Vertical-B” build orientation.

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).

Table 3-2: 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

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. It should be noted the following calculation does not factor in the CTE of the composite part itself (for the sake of simplification) since it varies by material and laminate configuration.

$$\text{Expansion factor} = (T_{cure} - T_{initial}) \times CTE$$

Subtracting the change factor from 1 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.

$$\text{Tool scaling factor} = 1 - \text{Expansion factor}$$

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

$T_{cure} = 350\ °F$

$T_{initial} = 75\ °F$

$CTE_{ULTEM\ 1010} = 26 \times 10^{-6}\ (in./in.\text{-}°F)$

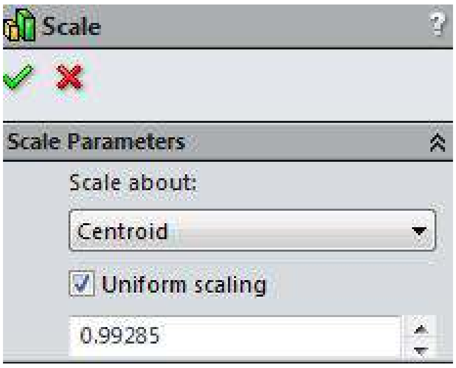
$$\text{Expansion factor} = (350\ °F - 75\ °F) \times 0.000026\ \frac{in.}{in. * °F} = 0.00715$$

Tool scaling factor $= 1 - 0.00715 = 0.99285$

This implies the tool will expand by 0.00715 inch per inch of tool length (at 350 °F) and the tool will need to be scaled by a factor of 0.99285 to compensate for that dimensional change and produce a composite part with the proper final geometry.

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



Scale Parameters

Scale about:

Centroid

☑ Uniform scaling

0.99285

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 obviously 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 tools for the 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).

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: Desired shape of composite part.

2. Create the trim area of the part by extending surfaces outside the EOP.

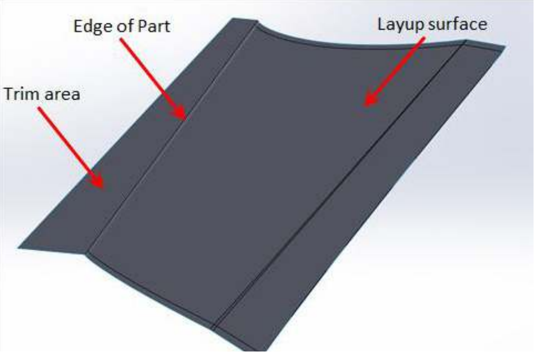


Figure 3-8: Adding the EOP and trim area.

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

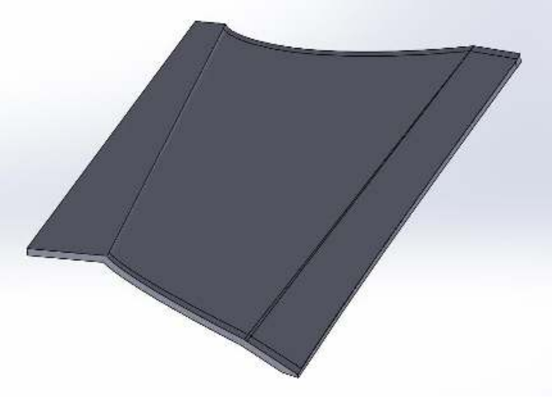


Figure 3-9: Thickening the layup surface.

4. Add stabilizing features such as legs, permitting the tool to sit flat on a table during layup. Stiffening features can also be added, if required. Identification features such as tool numbers can also be incorporated.

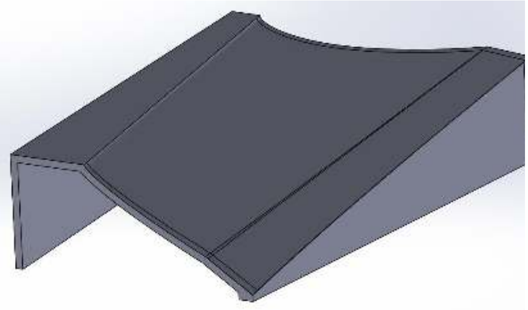


Figure 3-10: Adding support legs.

5. Add fillets at 90° angled corners to reduce stress concentrations and improve robustness.

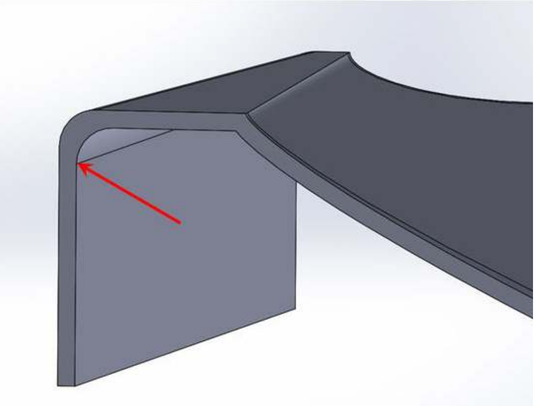


Figure 3-11: Addition of large-radii fillets to corners.

6. Round sharp corners and edges to prevent piercing of vacuum bagging materials, particularly when envelope bagging. A fillet radius of 0.25-0.5 inch is typically adequate.

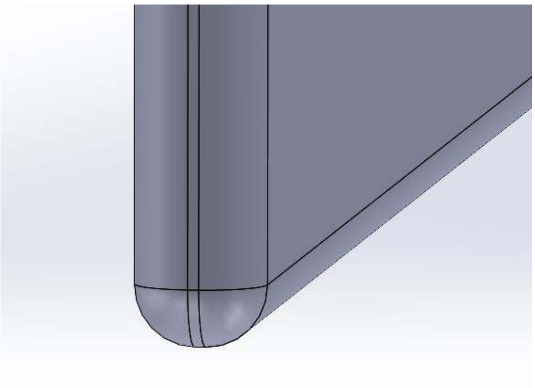


Figure 3-12: Rounding sharp edges and corners.

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.

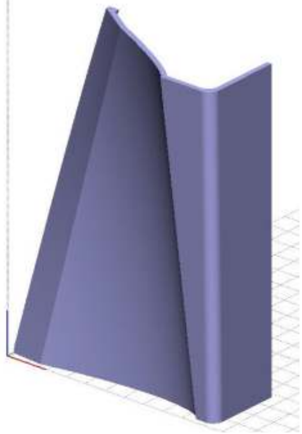


Figure 3-13: Vertical build orientation for the tool.

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

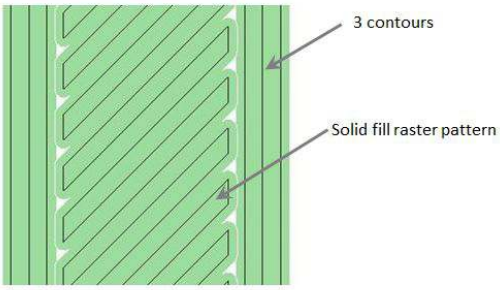


Figure 3-14: Cross-section of a solid fill toolpath.

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 can vary based on the consolidation pressure requirements of the application). 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 so that there is 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.

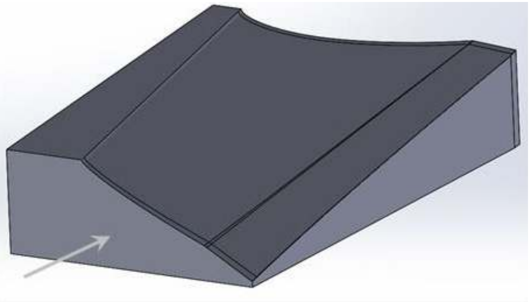


Figure 3-15: Sparse style tool (cavity indicated by the arrow).

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 sparse double dense fill pattern. The sparse double dense pattern provides 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.

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

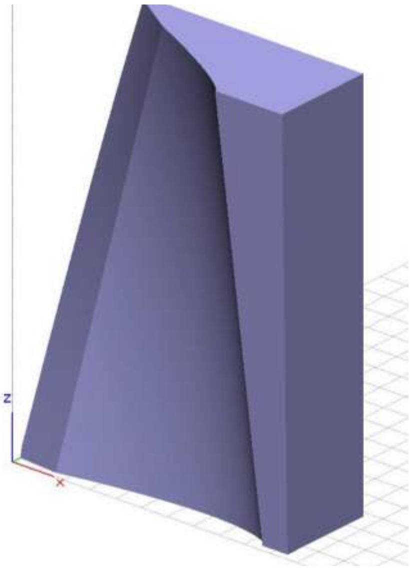


Figure 3-16: Recommended vertical build orientation for the tool.

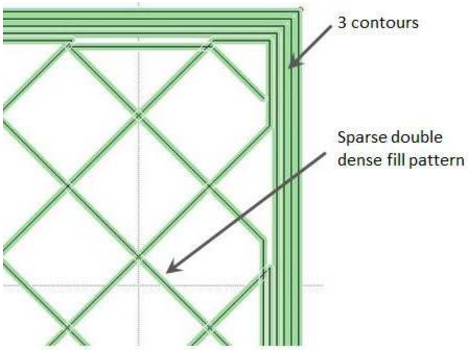


Figure 3-17: Top view of sparse double-dense fill toolpath.

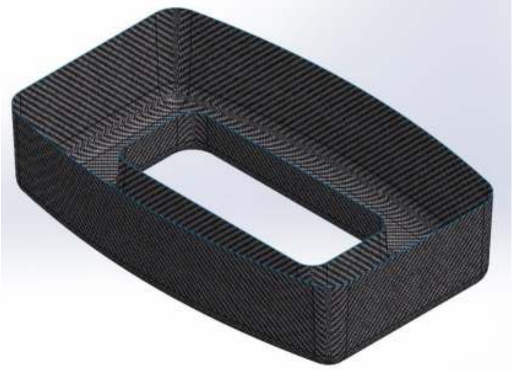


Figure 3-18: Example composite part.

1. Thicken the tool surface. A thickness of 0.3 inches is recommended. Also, extend the tool lay-up surfaces beyond the EOP to provide space for excess material.

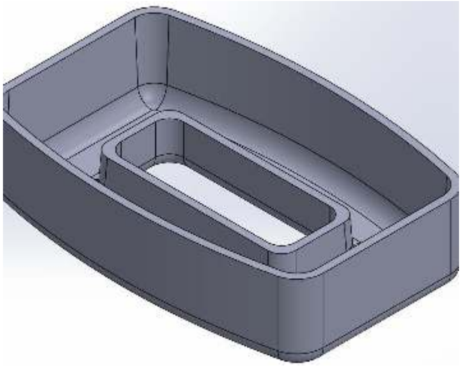


Figure 3-19: Thickening the tool surface.

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).

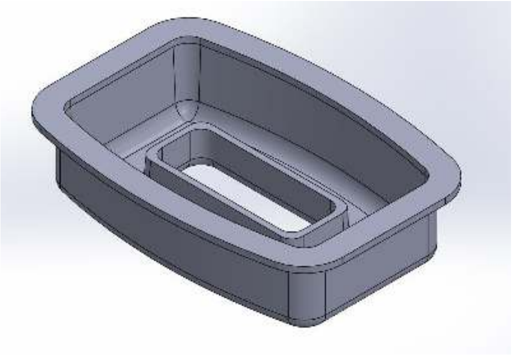


Figure 3-20: Addition of flanges to the top tool surfaces.

3. Remove sharp edges and corners by adding 0.1-0.5 inch fillets around the tool.

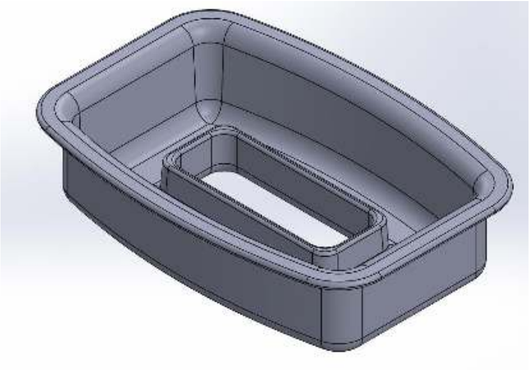


Figure 3-21: Rounding of tool corners and edges.

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).

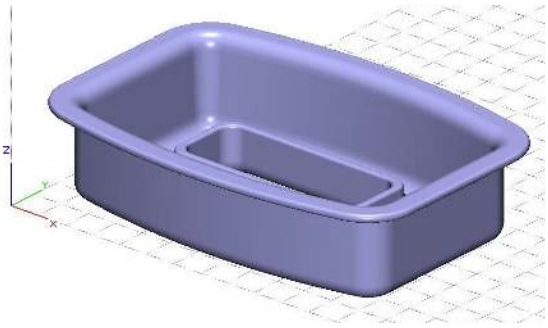


Figure 3-22: Recommended build orientation for the tool.

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. Figure 3-23 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.

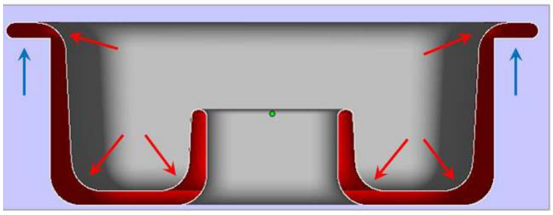


Figure 3-23: Cross-sectional view of tool design with red arrows indicating areas of anticipated stair-stepping and blue arrows indicating areas with required support material.

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.

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 fillets around the tool.

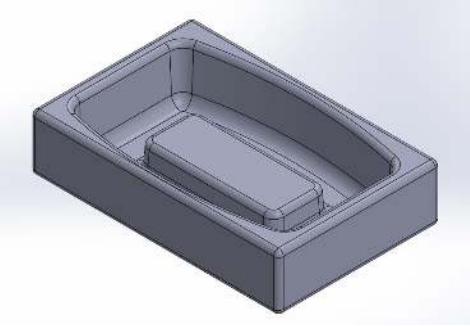


Figure 3-24: Sparse style female tool.

2. Process the file using Insight and use a sparse double dense 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).

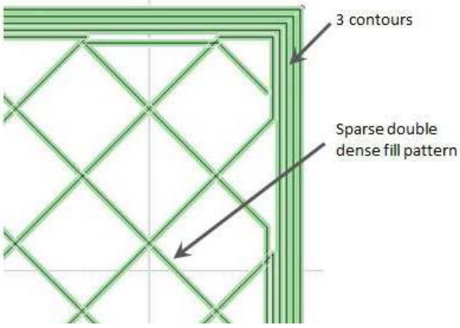


Figure 3-25: Top view of sparse double-dense fill toolpath.

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 lines (and similar) should be removed as they do not typically print well and are likely to be removed during sanding and sealing. 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 3-26, 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.

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 3-27 below show areas where excess material can be removed from the tool without harming performance.

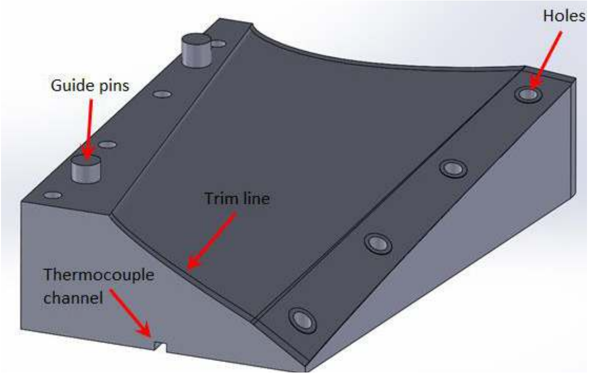


Figure 3-26: Tool features that typically must be removed or redesigned for FDM.

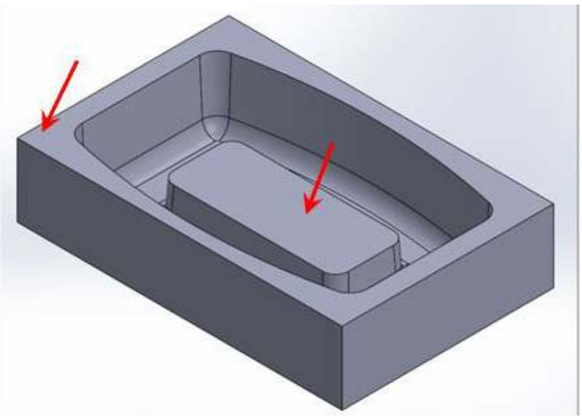


Figure 3-27: Excess material from a conventional tool design that can be removed.

Design of Trim Tools, Drill Guides and Similar Ancillary Tooling

Since the trim lines used on conventional tooling do not translate well to FDM tools, an alternative approach is to print separate tools to properly trim the final composite parts. Unlike an FDM composite layup tool, trim tools can generally be printed with any material and do not require any sanding or sealing. Such tools can be designed to print as quickly and inexpensively as possible. Examples are provided in the following procedures. Additional ancillary tooling for inspection fixtures, assembly aids, and other jigs and fixtures can also be designed and produced in a similar manner. It is also possible to incorporate such hardware as bushings and inserts when needed.

1. Begin by offsetting the tool surface by the thickness of the composite part to provide adequate clearance.

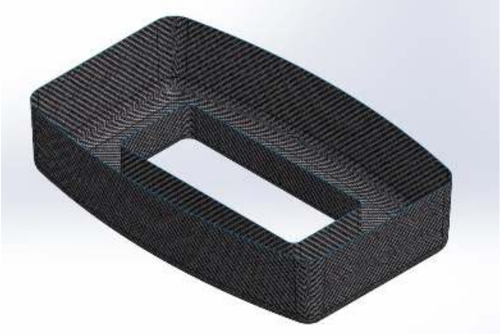


Figure 3-28: Example composite part.

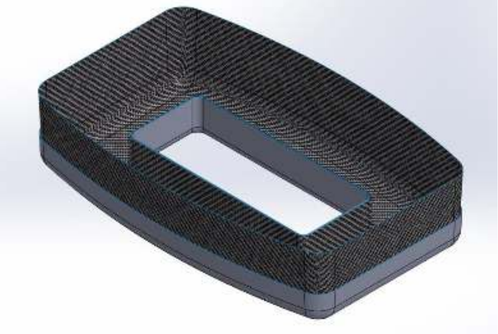


Figure 3-29: Offsetting the trim tool surface from the composite part.

2. Thicken the resulting surface by 0.2 inch (recommended) and enclose to form a solid.

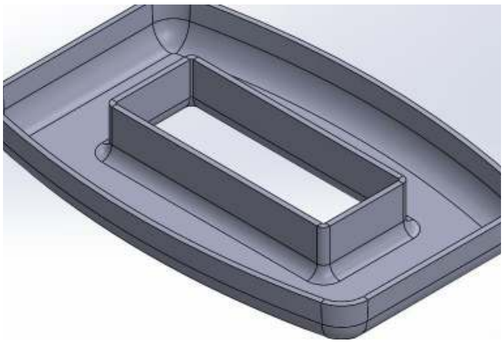


Figure 3-30: Trim-tool surface thickened.

3. Drill guides can be created as a separate tool or, in many cases, incorporated into the trim tool as shown in Figure 3-31. The tools can be printed with a single contour and solid raster fill pattern.

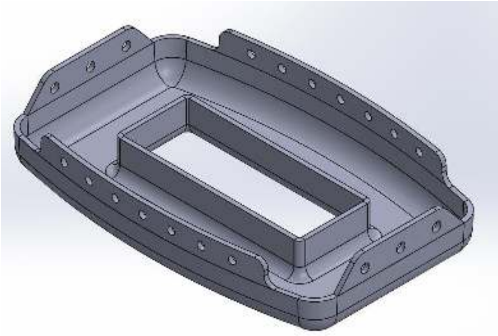


Figure 3-31: Trim profile with drill-guide features added.

Internal Feature Design and Modification

Internal features, such as holes or thermocouple ports within a tool, should incorporate self-supporting angles (>45°) into the tool design whenever possible to eliminate the need for additional support material. The examples in Figure 3-32 and 3-33 show how circular and rectangular holes and cavities require support material, whereas shapes that use self-supporting angles do not.

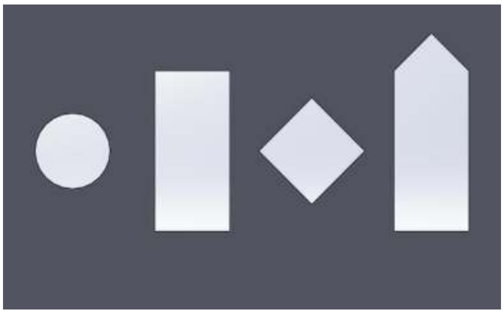


Figure 3-32: Example internal features for an FDM part.

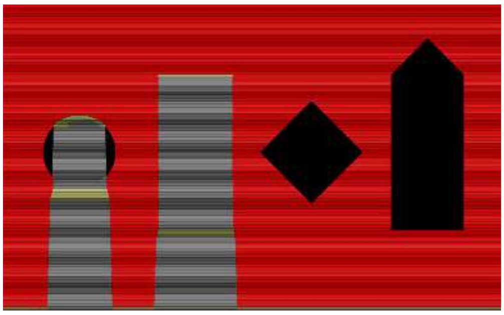


Figure 3-33: Example internal features with support material shown where required.

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 3-36). The example in Figures 3-35 and 3-36 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 3-37, a pin can be produced with no support material.

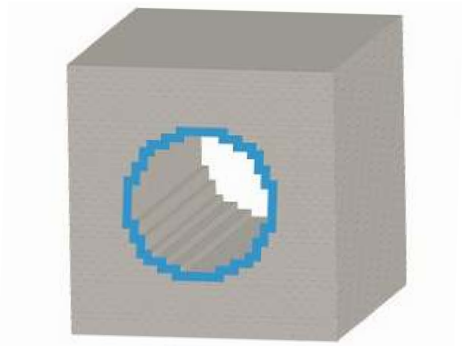


Figure 3-34: Resolution of a horizontally-oriented hole after printing (reaming recommended).

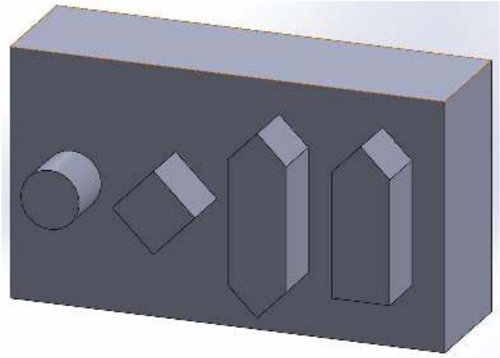


Figure 3-35: Example external features (typically used for guides or pins).

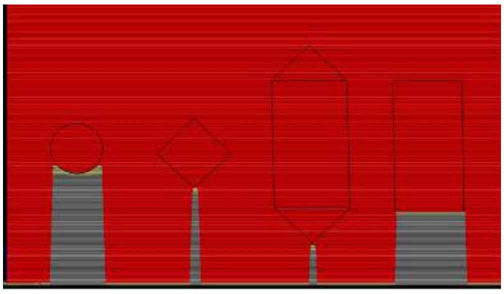


Figure 3-36: Example external features with support material added.

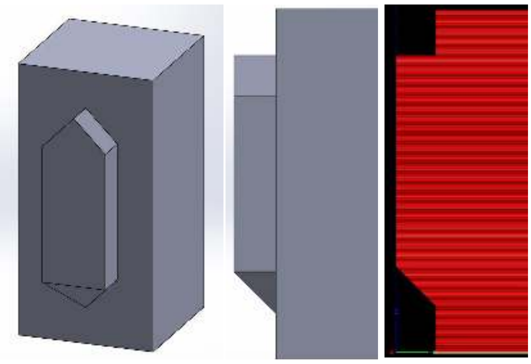


Figure 3-37: Guide feature designed with self-supporting angles in both horizontal axes, resulting in no required support material.

Tool Segmentation and Joining

For tools larger than the build chamber of the FDM machine (e.g., the Fortus 900mc build chamber is 36 x 24 x 36 inches), 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, mechanical features (overlaps, dovetails, saw-tooth patterns, etc.) can be easily incorporated to ensure proper fit and alignment. An example of a tool requiring segmentation is shown in Figure 3-38; additional information for this specific case is provided in Section 6.

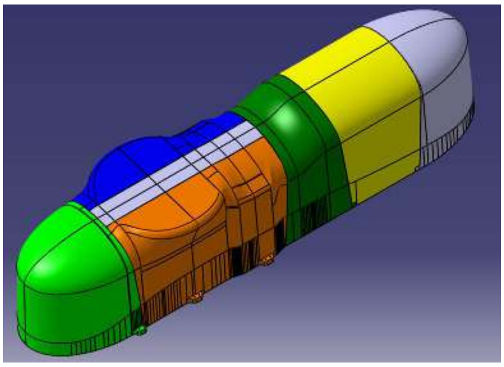


Figure 3-38: Aurora Flight Sciences fairing tool model, sectioned into seven segments (noted by the different colors).

For joining tool segments, bonding can be performed with compatible structural adhesives with the appropriate capability to withstand cure temperatures. Materials such as Hysol EA 9394 two-part epoxy paste adhesive have been effectively used for such bonding (although many other alternative materials will 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.

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 abrasions, 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 it is likely not capable of withstanding more than approximately 30 cure cycles at 350 °F before it begins to break down and oxidize. Evaluation of alternative epoxy resin systems better-suited to continuous exposure at 350 °F 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 from Airtech for years to provide an effective layup and release surface to tools made from traditional materials and processes. Such films are best-suited for relatively flat tool shapes with few or very gradual complex contours since they exhibit minimal elongation. They also are 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.

A variety of adhesive-backed films will be effective, provided they meet the temperature requirements and have enough elongation to conform to the tool shape without wrinkling. Additional higher-elongation materials, such as Toolwright from Airtech, are also being evaluated with more complex tool shapes and results will be provided in subsequent design guide releases.

Surface Finish Results

Figure 4-1 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. Ra). As can be seen, the common methods used to seal FDM composite tooling produces surface finishes considerably smoother than typical requirements.

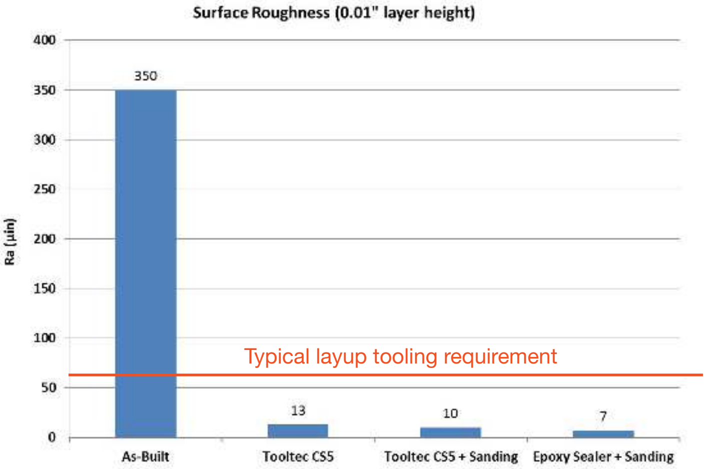


Figure 4-1: Resulting surface roughness for common FDM post-processing techniques.

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).

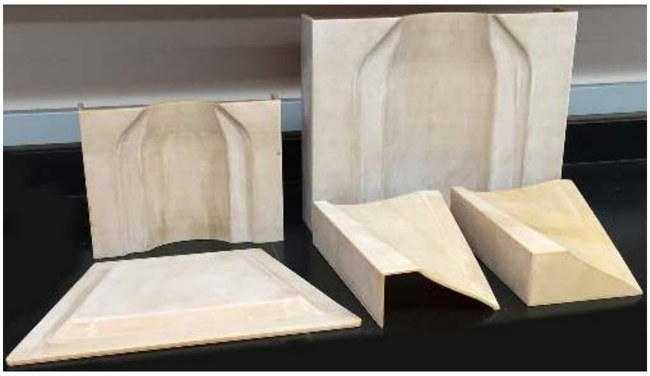


Figure 5-1: Tool designs used for thermal stability testing (shown sanded, but unsealed).

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, held at temperature for two hours (minimum) under full vacuum, and then ramped back down to below 150 °F between cycles for a total of 10 consecutive oven cycles.

The accuracy of the 3D scanner is ± 0.0015 inch and the accuracy of the FaroArm is also ± 0.0015 inch for a total accuracy of 0.003 inch. 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.

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 5-1, the scan data shows the shell-style tool has areas that vary from the model by as much as approximately 0.019 inch and over 92% of the tool is within ± 0.015 inch. 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 in size.

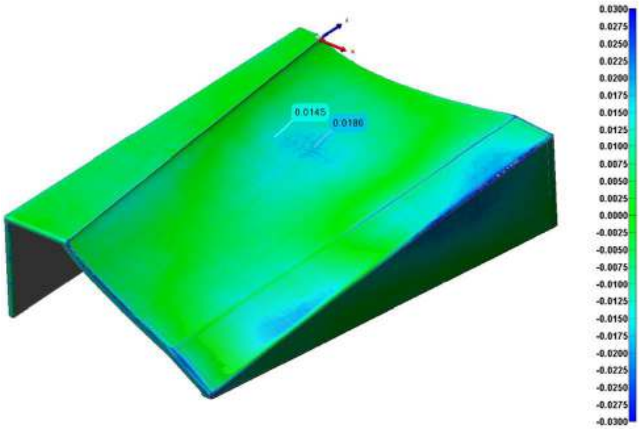


Figure 5-2: 3D scan data for a UAV fan-blade tool (shell style) with color map comparison to the original CAD model (no thermal cycling). Dimensions are in inches.

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, as shown in Figure 5-3. For this tool, over 90% of the tool is within ± 0.015 inch 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.

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 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.

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 (75 °F/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 for two weeks. After conditioning, two tools were subsequently dried for 4 hours at 250 °F. 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.

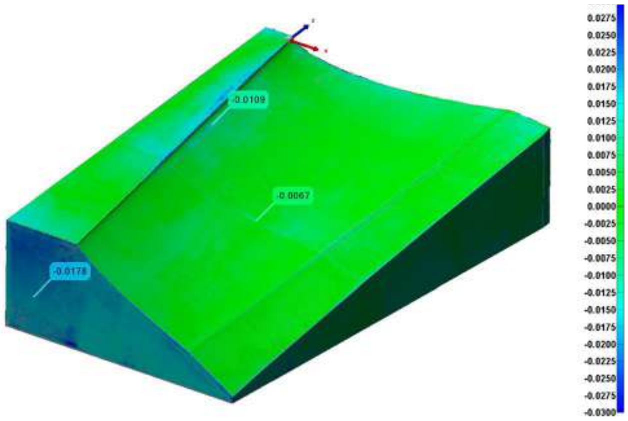


Figure 5-3: 3D scan data for a UAV fan-blade tool (sparse style) with color map comparison to the original CAD model (no thermal cycling). Dimensions are in inches.

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.

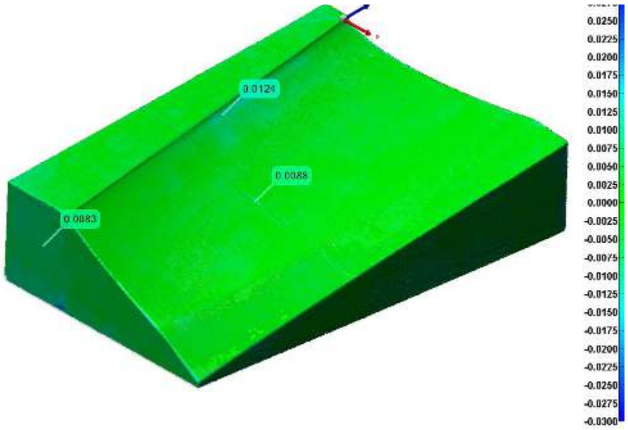


Figure 5-5: 3D scan data for a UAV fan-blade tool (sparse style) after thermal cycling with color map comparison to the 3D scan data for the same tool prior to cycling. Dimensions are in inches.

As expected, moisture exposure testing demonstrated that tools dried before use (4 hours at 250 °F) produce laminates of acceptable quality (no significant porosity or other obvious issues). Even the tools that were saturated and not dried produced laminates that have no visual indications of issues (final sectioning and review of those laminates was not yet complete at the time of publication; results will be included in subsequent versions).

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 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, 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 loading condition at multiple elevated temperatures (355 °F, 385 °F, and 400 °F) 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.

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. The cycled tools have not yet completed additional 3D scanning dimensional inspection to evaluate thermal stability, although that work is planned and results will be provided when available.

For the analytical evaluation of ULTEM 1010 resin, the flexural creep data measured by DMA and shifted to 355 °F using TTS principles is shown in Figure 5-6, along with stress versus time relationship at 355 °F 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 and 100 psi.

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 releases.

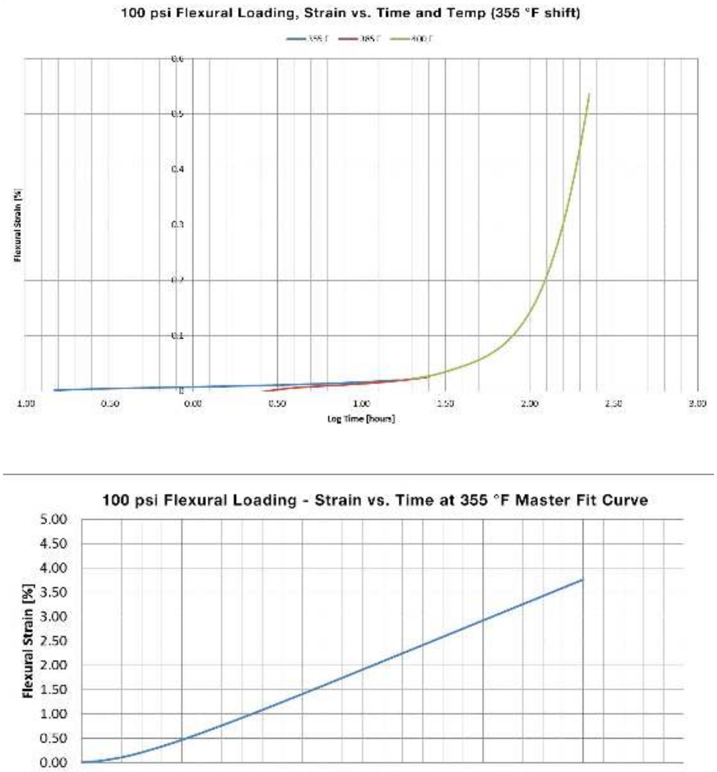


Figure 5-6: Flexural creep data for ULTEM 1010 resin test coupons – shifted to 355 °F using TTS (top) and 355 °F data only (bottom).

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 of tools since the need for cranes and forklifts is eliminated for most modest sized tools. For example, the weight for a male radome tool measuring approximately 50 x 14 x 8 inches was less than 35 pounds. If damage does occur, FDM thermoplastics are highly repairable. This can be accomplished 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.

SECTION 6 – USE CASES AND EXAMPLES

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 in.) and thickness, but reduced span (~24 in.) relative to the full 96-inch 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 6-1. 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 6-1 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 6-2 and 6-3 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.

Table 6-1: Build, cost, and lead time data.

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

* 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 0.02 inch slice height (not available at the time of this project).
Resulting tooling cost would be ~$5,600.
Material use includes both model and support.

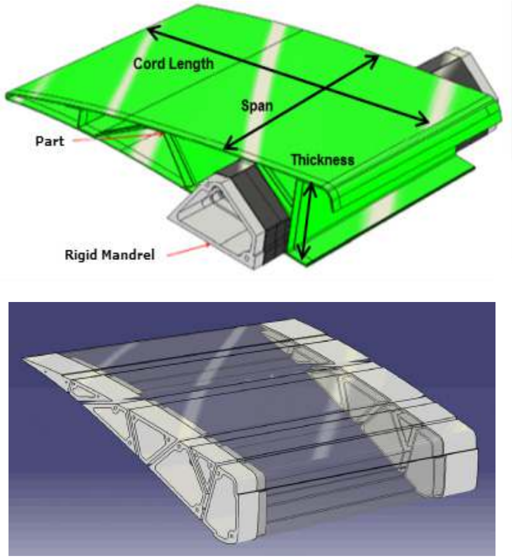


Figure 6-1: Single shot composite aileron schematic (top) and model of seven mandrel sections (bottom).

Figure 6-2: Aileron in-process lay-up and final configuration (vacuum bagging not shown).

Figure 6-3: Final single shot composite aileron sub-scale demonstrator.

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 x 24 inches x 30 inches [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 W x 24 inches D x 36 inches H). Additionally, as can be seen in Figure 6-4, 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), 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 in^3 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.

Figure 6-4: Dry fitting the tool after FDM build (top) and tool with cured fiberglass/epoxy fairing (bottom).

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 6-2). 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

Table 6-2: Cost comparison data (based on customer data).

TOOLING MATERIAL	COST	LEAD TIME	FDM MATERIAL USE
ALUMINUM	$65,000	7 weeks	--
CARBON / EPOXY	$95,000	12-14 weeks	--
FDM – POLYCARBONATE	$25,000	2-3 weeks*	4864 in^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.

Aurora
FLIGHT SCIENCES

Figure 6-5: Composite belly pod fairing installed on Centaur aircraft (before and after painting).

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 mold halves were built using a relatively small layer thickness (0.01 inch, T14 build tip) in a horizontal build orientation, as shown in Figure 6-6. 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 6-3. 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. Ra.

The tool has been used successfully to produce multiple sets of propeller blades, shown Figure 6-6, for wind-tunnel testing. The specific processing details are proprietary, but the carbon/epoxy blades are cured at a temperature of approximately 250 °F and pressures exceeding 500 psig. 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.

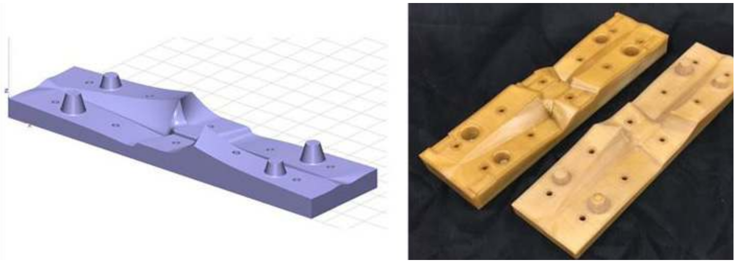


Figure 6-6: Propeller compression molding tool: horizontal build orientation (left), matched mold halves (right).

Figure 6-7: Carbon/epoxy composite propeller blade produced on a compression mold tool made from ULTEM 1010 resin

Table 6-3: 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.	52 in^3	17 hours	$625
LOWER MOLD	14 x 4 x 2 in.	36 in^3	13 hours	$450

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

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 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 below.

Table 6-4: Repair tool build data (ULTEM 1010 resin).

TOOL GEOMETRY	~ SIZE	FDM MATERIAL USE	BUILD TIME	COST*
HAT STIFFENER	28 x 9 x 2 in.	80 in³	13 hours	$780
CONTOURED PATCH	25 x 25 x 2 in.	320 in³	40 hours	$2,930

*Based on build time and material for a Fortus 900mc using 0.02 inch slice height (machine cost amortized over 5 years, operating at 65% utilization).

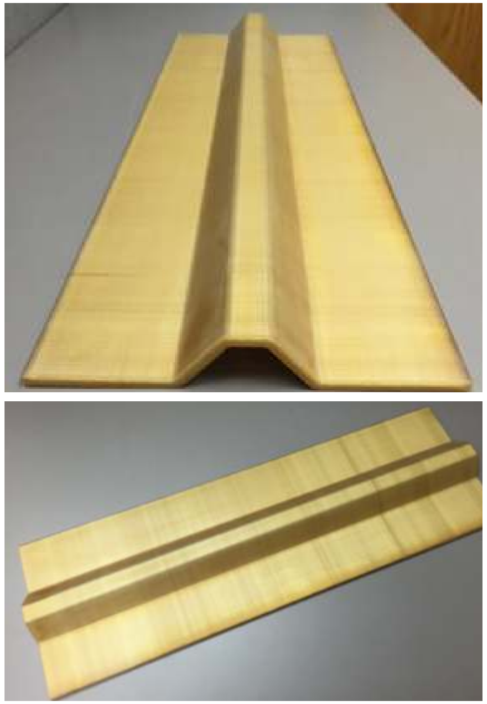


Figure 6-8: Common repair tool geometry – hat stiffener shape.

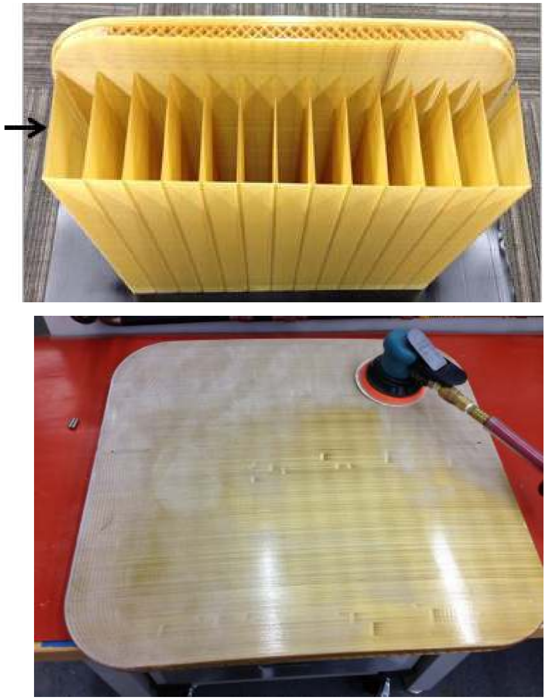


Figure 6-9: Complex contour patch repair tool as built (top, with arrow indicating stabilizer walls) and during post-processing (bottom).

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, although this application required only 250 °F. 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. Ra. The resulting part (produced on the male sparse tool) is also shown in Figure 6-10 (a proprietary coating is shown on the bag-side surface).

Additionally, a separate drill and trim tool (Figure 6-11) 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 layup 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 6-10: FDM tool in ULTEM 1010 resin (top) and resulting part (middle, bottom).

DESIGN ASPECTS	SELECTION/DETAILS	
Design Style	*Male Shell*	*Male Sparse*
CONSTRUCTION	0.3 inch thickness	0.25 inch sparse spacing, 0.1 inch shell thickness
SIZE (APPROXIMATE)	11 x 4 x 6 inches (L x W x H)	
BUILD ORIENTATION	Vertical	Vertical
SLICE HEIGHT	0.01 inch (T14)	0.01 inch (T14)
MATERIAL USE	37 in³	48 in³
SUPPORT MATERIAL USE	0.1 in³	0.3 in³
BUILD TIME	18.5 hours	15 hours
TOOL WEIGHT	2.2 pounds	4.9 pounds
COST* – MACHINE OWNER	$535	$570
COST – SERVICE BUREAU	$1870	$1790
INTENDED USE	• 250 °F 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 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).

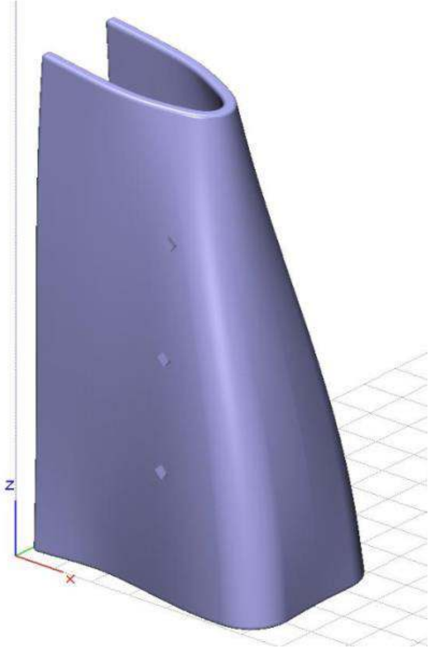


Figure 6-11: Shell-style shroud tool vertical build orientation (top); sparse-style lay-up tool and corresponding drill and trim tool (bottom).

Pan-Skin Tool

Background: The “pan-skin” shape is a common geometry in composite structures (sandwich structures in particular) for components such as doors, covers and access panels. The subject tool was produced for evaluation of a somewhat simplified blocker door design for an aircraft thrust reverser. A male tool is used for this application to control the IML of the part, which is the critical surface in this case since it is subsequently bonded to honeycomb core and the relatively flat back-side skin to create the final sandwich structure.

Approach: A shell tool was produced in ULTEM 1010 resin, permitting cure temperatures of >350 °F. Since the pan-skin shape has inherent rigidity, the shell style is appropriate versus the more robust sparse construction. The tool was built in a vertical orientation and since the ramp areas are designed with self-supporting angles (>45°, relative to the build platform), minimal support material is required. A fine-resolution build tip (0.010 inch slice height) was used to minimize stair-stepping on the ramp sections of the tool. The tradeoff with this selection is a longer build time, but for this type of ramp feature, the minimization of stair-stepping leads to less post-processing labor and improved accuracy in those areas. Stabilizer walls are used to provide stability during the build. For post-processing, tools were manually abraded and sealed with a two-part epoxy, resulting in a final surface finish smoother than 16 µin. Ra.

Additionally, a separate example trim tool (Figure 6-12) was created to supplement the lay-up tool and was printed in ASA to save additional cost (the trim tool cost was approximately $200). This ancillary tool is designed to nest on the lay-up mold-controlled IML surface to ensure accurate alignment and final trim profile.

Results: The tool was built and post-processed in less than 3 days at a cost less than $1000*.

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

Figure 6-12: FDM tool in ULTEM 1010 resin (top) and corresponding trim tool (bottom) for a blocker door, pan-skin tool concept.

Figure 6-13: Pan skin vertical build orientation (top); top view of sealed tool (middle); the unfilled bottom surface of the tool with no post-processing (bottom).

Pan-Skin Tool

DESIGN ASPECTS	SELECTION/DETAILS
Design Style	*Shell*
CONSTRUCTION	0.3 inch shell thickness
SIZE (APPROXIMATE)	21 x 13.5 x 1.1 inches (L x W x H)
BUILD ORIENTATION	Vertical with stabilizer walls
SLICE HEIGHT	0.010 inch (T14 tip)
MATERIAL USE	74 in^3
SUPPORT MATERIAL USE	0.2 in^3
BUILD TIME	32 hours
TOOL WEIGHT	<3.5 pounds
COST* – MACHINE OWNER	<$1,000
COST – SERVICE BUREAU	$2,900
INTENDED USE	• 350 °F cure temp, 15-100 psi pressure • Carbon/epoxy skin for a honeycomb core sandwich structure • Commercial aircraft nacelle component
COMMENTS AND HIGHLIGHTS	• For a production variant of this tool, consider a sparse style construction for increased rigidity • Vertical build orientation selected to minimize support material use • Fine build resolution selected to minimize stair-stepping on ramp sections – resulting in a slightly longer build time, but improved surface finish and reduced post-processing labor • Tool is appropriate for use with either surface or envelope bagging methods – recommended to increase area outside of EOP if using surface bagging (to allow room for bagging)

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

UAV Fan Blade Tools

Background: Stratasys worked with a leading aerospace company to develop design concepts for a UAV fan blade. The resulting geometry was used to evaluate ULTEM 1010 resin tools in the development of this design guide. The geometry represents a common laminate shape found in a wide array of aero structures. The layup surface for this design represents the OML or aerodynamic surface of the fan-blade assembly.

Approach: Both shell and sparse tools were produced in ULTEM 1010 resin to compare performance. Regardless of design, both tools were built in a vertical orientation to minimize stair-stepping on the contoured layup surface, as well as to minimize support-material consumption. The largest available build tip was used, delivering a 0.020 inch layer height to minimize build time while still delivering acceptable surface finish, given that the tools are post-processed before use. Stabilizer walls were not used. Post-processing consisted of manual abrasion and tool sealing with a two-part epoxy, resulting in a final surface finish smoother than 16 µin. Ra.

Characterization of accuracy and thermal stability, as described in Section 5, revealed no significant differences between the two construction styles. Additional testing and evaluation after more thermal cycles is underway. Both tools can be used with either surface or envelope vacuum bagging, although guidelines for raster fill spacing for the sparse tool should be followed to prevent tool damage.

Results: Both 350 °F cure temperature-capable tool styles were built and post-processed in less than 3 days at costs ranging from $480 (shell) to $1000 (sparse).*

* Costs are based on build time and material for a Fortus 900mc using 0.02 inch slice height (machine cost amortized over 5 years, operating at 65% utilization).

Figure 6-14: Sparse- and shell-style layup tools for UAV fan blades.

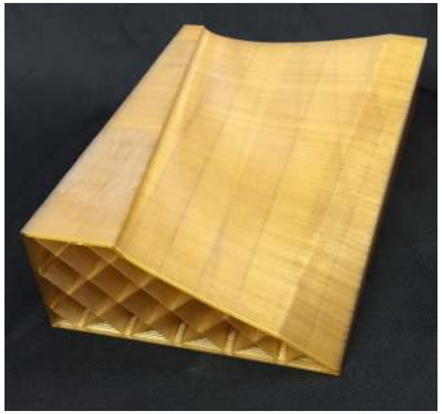


Figure 6-15: Sparse tool with large raster spacing and uncapped (open) ends.

UAV Fan Blade Tools

DESIGN ASPECTS	SELECTION/DETAILS	
Design Style	*Shell*	*Sparse*
CONSTRUCTION	0.3 inch thickness	0.25 inch sparse spacing, 0.1 inch shell thickness
SIZE (APPROXIMATE)	15 x 11 x 4 inches (L x W x H)	
BUILD ORIENTATION	Vertical – no stabilizer walls	Vertical – no stabilizer walls
SLICE HEIGHT	0.02 inch (T40)	0.02 inch (T40)
MATERIAL USE	47 in^3	106 in^3
SUPPORT MATERIAL USE	0.1 in^3	0.3 in^3
BUILD TIME	9 hours	15 hours
TOOL WEIGHT	2.2 pounds	4.9 pounds
COST* – MACHINE OWNER	$480	$1,000
COST – SERVICE BUREAU	$3,200	$6,950
INTENDED USE	• 350 °F cure temp, 15-100 psi pressure (verify proper sparse fill spacing when using envelope bagging) • Carbon/epoxy laminate for a UAV fan blade assembly • Low-volume part fab (shell) or production volumes (sparse)	
COMMENTS AND HIGHLIGHTS	• For a production tool, the sparse-style design is recommended. The shell-style tool provides an inexpensive option for very low volumes, but does have some slight perceptible flex/deflection during material layup. • Vertical build orientation used to minimize support 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	

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

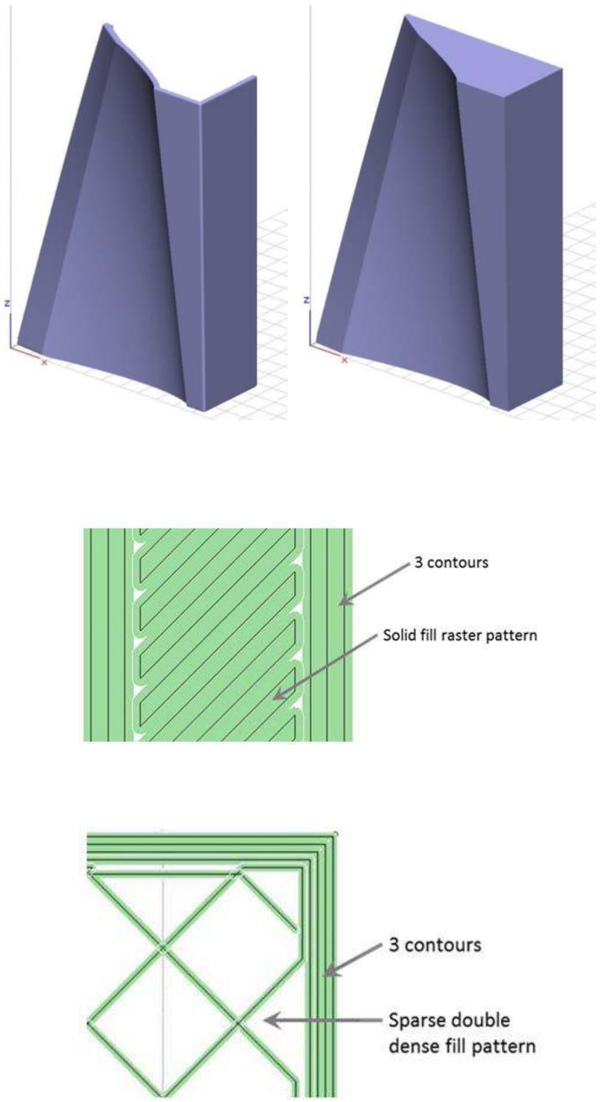


Figure 6-16: Vertical build orientation for both tool styles (top); toolpaths for shell tool (middle) toolpath and sparse tool (bottom).

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 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 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. Ra.

Results: A female tool design intended for 250 °F 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 6-17: UAV bulkhead tool.


Figure 6-18: UAV bulkhead tool being prepped for use.

UAV Bulkhead Tool

DESIGN ASPECTS	SELECTION/DETAILS
Design Style	*Shell*
CONSTRUCTION	0.3 inch 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
SUPPORT MATERIAL USE	15 in³
BUILD TIME	31.5 hours
TOOL WEIGHT	2.6 pounds
COST – MACHINE OWNER	$980
COST – SERVICE BUREAU	$3,400
INTENDED USE	• 250 °F cure temp, 15-100 psi 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.

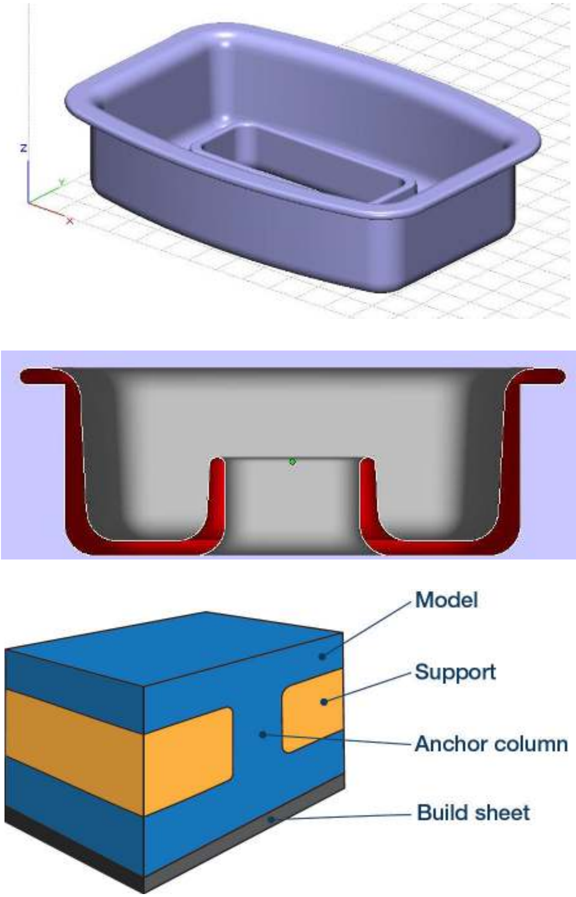


Figure 6-19: Horizontal build orientation (top); cross-section of tool design (middle); cross-section showing anchor column, used to prevent thermal distortion during the FDM process (bottom).

Aerodynamic Fairing Tool

Background: The aero fairing shape is a common geometry in composite structures ranging from missile structures to engine nacelle cowlings. The subject design was produced as a demonstration tool for the 2015 CAMX technical conference. A female tool is used for this application to control the OML of the part, which in practice would represent the critical aerodynamic surface.

Approach: A shell-style tool was produced in ULTEM 1010 resin, permitting cure temperatures of >350° F. This tool was intended for demonstration with very low part volumes so a more robust construction was not required. The tool was built in a vertical orientation to minimize stair-stepping on the contoured lay-up surface, as well as to minimize the amount of support material. The largest available build tip was used, delivering a 0.020 inch layer height to minimize build time while still delivering acceptable surface finish, since the tool was post-processed before use. For post-processing, the tool was manually abraded and sealed with a two-part epoxy, resulting in a final surface finish smoother than 16 μin. Ra.

The tool was used to demonstrate the use of integral heating to cure the composite skin, successfully enabling part production outside an oven or autoclave.

Results: The tool was built and post-processed in 3 days at a cost of $2450.*

* Costs are based on build time and material for a Fortus 900mc using 0.02 inch slice height (machine cost amortized over 5 years, operating at 65% utilization).

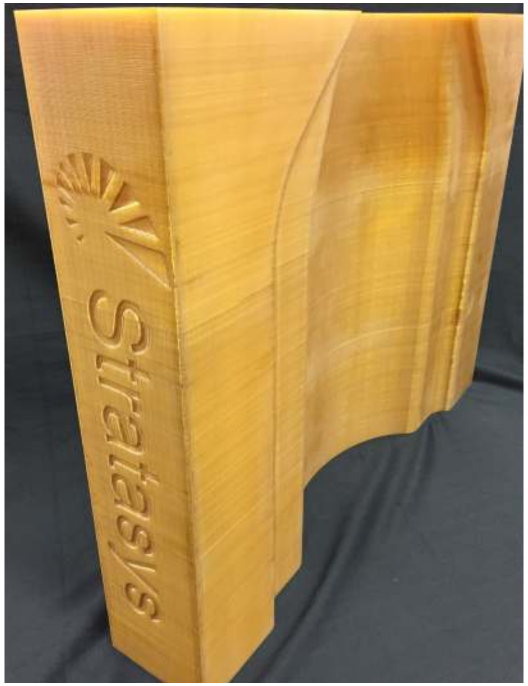


Figure 6-20: Aerodynamic fairing tool.

Figure 6-21: Composite fairing.

Aerodynamic Fairing Tool

DESIGN ASPECTS	SELECTION/DETAILS
Design Style	*Shell*
CONSTRUCTION	0.3 inch shell thickness
SIZE (APPROXIMATE)	29 x 22 x 5 inches (L x W x H)
BUILD ORIENTATION	Vertical
SLICE HEIGHT	0.020 inch (T40 tip)
MATERIAL USE	268 in³
SUPPORT MATERIAL USE	0.1 in³
BUILD TIME	30 hours
TOOL WEIGHT	12 pounds
COST – MACHINE OWNER	$2,450
COST – SERVICE BUREAU	$8,800
INTENDED USE	• 350 °F cure temp, vacuum bag only, integrally heated • Carbon/epoxy skin for an aerodynamic fairing or cowling, typically in a honeycomb core sandwich structure • Commerical aircraft nacelles, UAVs, missiles, automotive motorsports
COMMENTS AND HIGHLIGHTS	• For a production variant of this tool, consider sparse construction or integrated stiffeners for increased rigidity • Vertical build orientation selected to minimize 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 • Tool is most appropriate for use with surface bagging methods

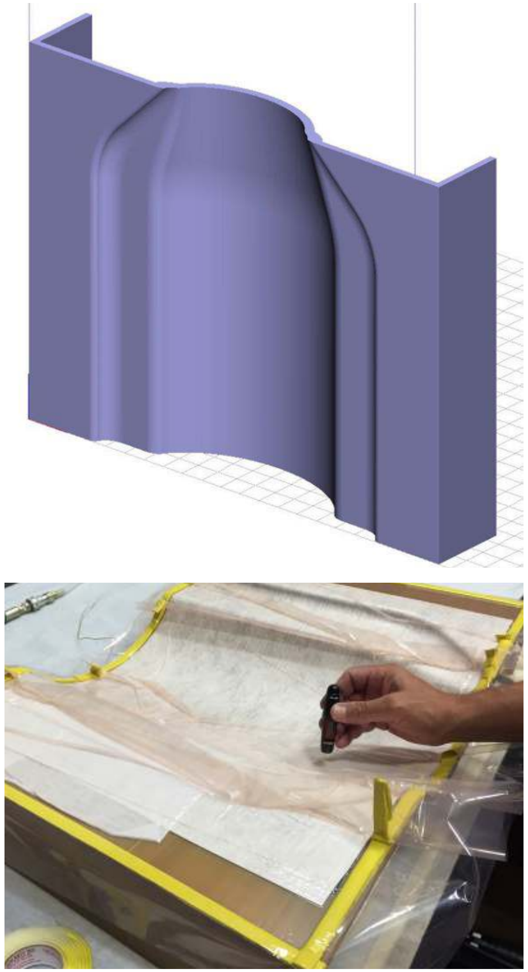


Figure 6-22: Vertical build orientation (top); vacuum-bagged part and tool during laminate fabrication (bottom).

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 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", available at Stratasys.com (stratasys.com/solutions/additive-manufacturing/tooling/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 support materials can be used to produce sacrificial tooling capable of withstanding cure temperatures up to 350 °F. Unlike ST-130, ULTEM support materials are not soluble. However, they do 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 this 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

Open Insight software.

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.

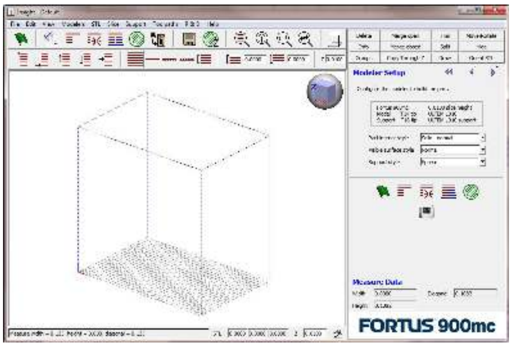


Figure 8-1: Home screen of Insight software upon opening.

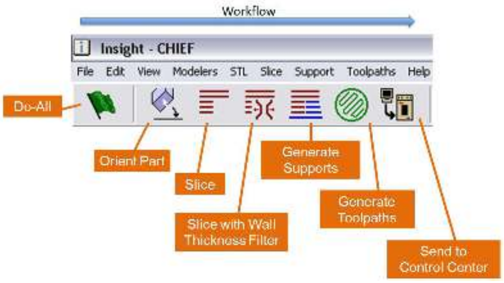
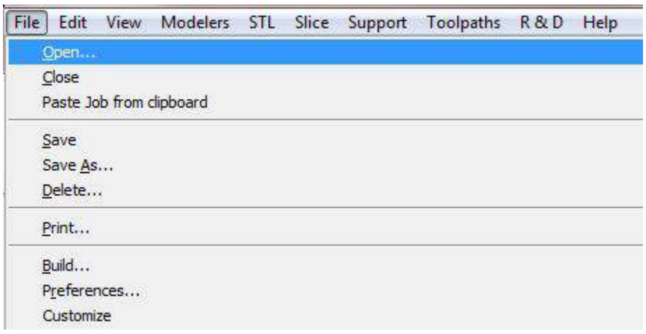


Figure 8-2: Workflow commands for Insight software.

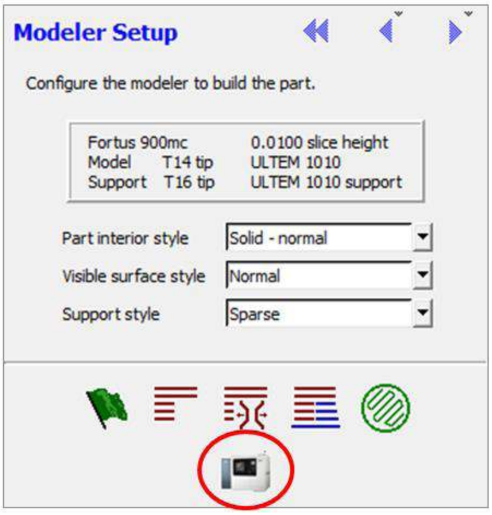
Import STL

Insight only allows files with the extension “.stl” to be imported. Any CAD software will work as long as it can export an STL. 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 (circled in red) on the right side of the screen.



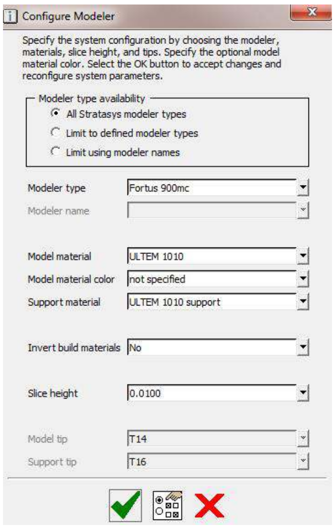
Modeler Setup

Configure the modeler to build the part.

Fortus 900mc	0.0100 slice height
Model T14 tip	ULTEM 1010
Support T16 tip	ULTEM 1010 support

Part interior style: Solid - normal
Visible surface style: Normal
Support style: Sparse

A new menu will appear in the middle of the screen.



Configure Modeler

Specify the system configuration by choosing the modeler, materials, slice height, and tips. Specify the optional model material color. Select the OK button to accept changes and reconfigure system parameters.

Modeler type availability
● All Stratasys modeler types
○ Limit to defined modeler types
○ Limit using modeler names

Modeler type: Fortus 900mc
Modeler name:

Model material: ULTEM 1010
Model material color: not specified
Support material: ULTEM 1010 support

Invert build materials: No

Slice height: 0.0100

Model tip: T14
Support tip: T16

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 in. 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	SLICE HEIGHT (INCHES)	BEAD WIDTH (INCHES)
T14	0.010	0.020
T20	0.013	0.026
T40	0.020	0.040

Table 8-1: Tip size and slice height specifications for ULTEM 1010 resin.

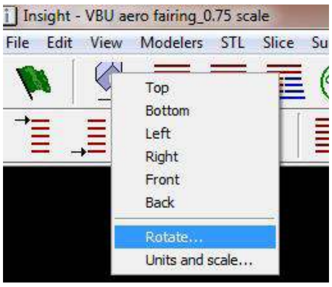
7. Click the green check.

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.

Insight also allows the user to orient the part at a specific angle. Select the ***Orient Part*** icon and then ***Rotate***.



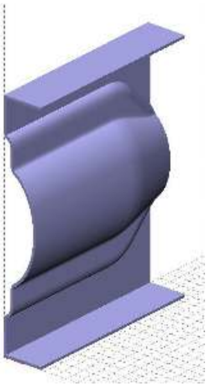


Figure 8-3: Orientation of part after being imported into Insight.

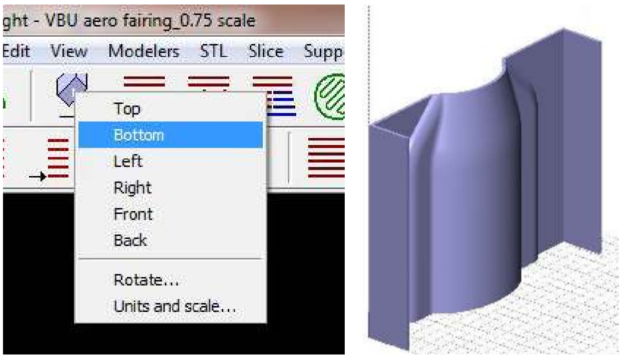
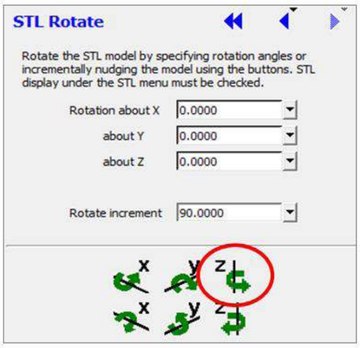


Figure 8-4: Orientation of part after selecting the bottom reference.

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.



STL Rotate

Rotate the STL model by specifying rotation angles or incrementally nudging the model using the buttons. STL display under the STL menu must be checked.

Rotation about X 0.0000

about Y 0.0000

about Z 0.0000

Rotate increment 90.0000

The part is now in the optimal orientation.

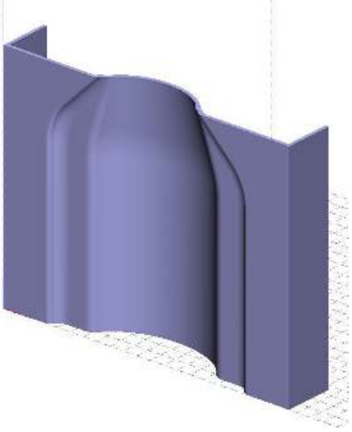
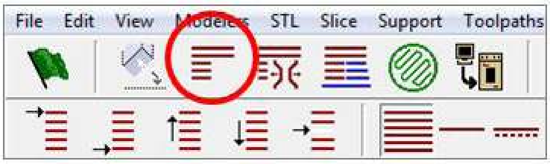


Figure 8-5: Part in optimal orientation after rotation.

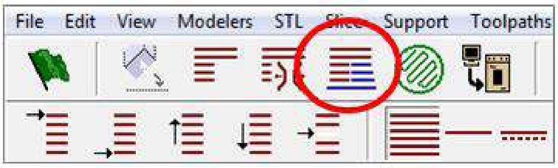
Slice the part

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.

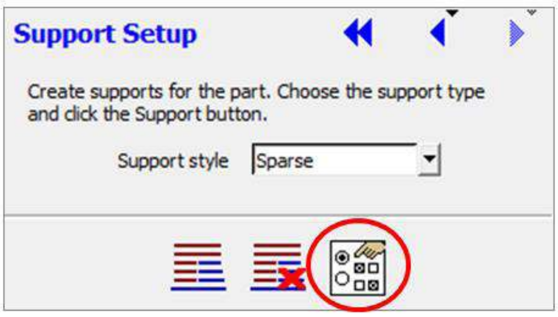


Generate support

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***.

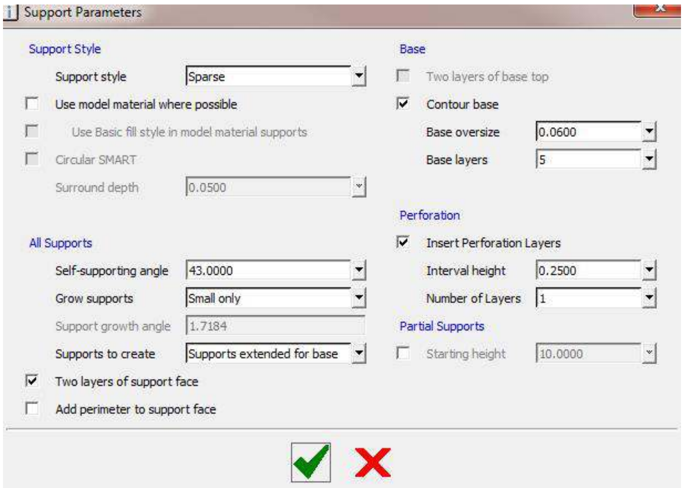


Support Setup

Create supports for the part. Choose the support type and click the Support button.

Support style: Sparse

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.



Support Parameters

Support Style
Support style: Sparse
☐ Use model material where possible
☐ Use Basic fill style in model material supports
☐ Circular SMART
Surround depth: 0.0500

All Supports
Self-supporting angle: 43.0000
Grow supports: Small only
Support growth angle: 1.7184
Supports to create: Supports extended for base
☑ Two layers of support face
☐ Add perimeter to support face

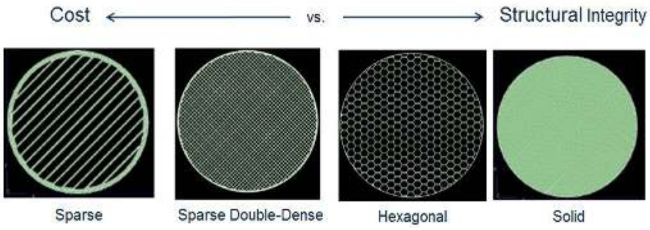
Base
☐ Two layers of base top
☑ Contour base
Base oversize: 0.0600
Base layers: 5

Perforation
☑ Insert Perforation Layers
Interval height: 0.2500
Number of Layers: 1

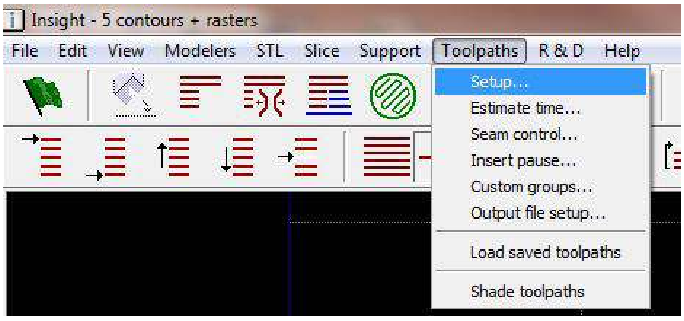
Partial Supports
☐ Starting height: 10.0000

Generate toolpaths

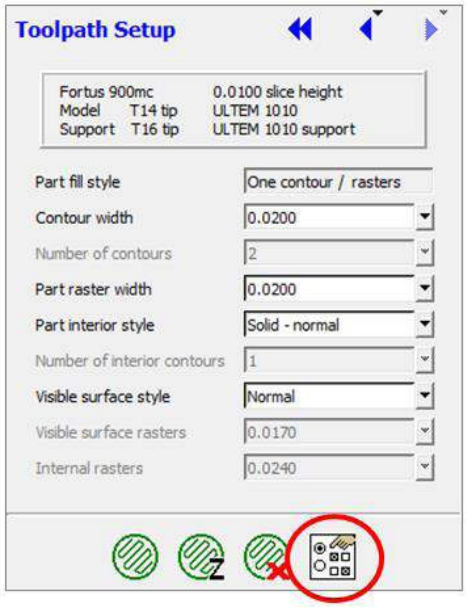
There are four different “infill” patterns as indicated in the diagram below. The tradeoff among these patterns is cost, strength and print time. Parts with a solid fill pattern will be stronger, but require more time and material to print. It is recommended to find a balance of these variables without compromising strength.



To generate a toolpath, select ***Toolpaths***, then ***Setup***.



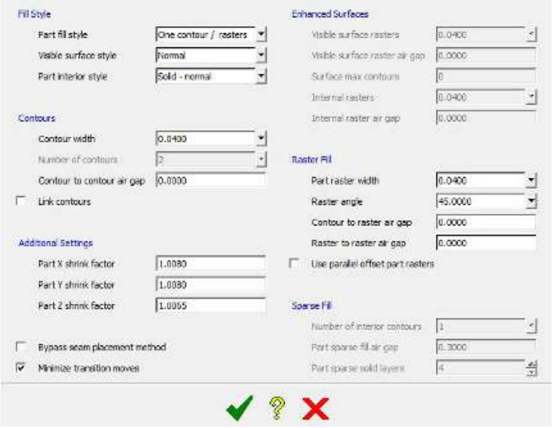
The following menu will appear on the right side of the screen and contains the relevant toolpath information. This menu lets the user change various aspects of the toolpath such as contour width, number of contours and fill pattern. Custom toolpaths can be created by selecting the icon indicated below.



Fortus 900mc 0.0100 slice height
Model T14 tip ULTEM 1010
Support T16 tip ULTEM 1010 support

Part fill style	One contour / rasters
Contour width	0.0200
Number of contours	2
Part raster width	0.0200
Part interior style	Solid - normal
Number of interior contours	1
Visible surface style	Normal
Visible surface rasters	0.0170
Internal rasters	0.0240

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.


Fill Style

Part fill style	One contour / rasters
Visible surface style	Normal
Part interior style	Solid - normal

Contours

Contour width	0.0400
Number of contours	2
Contour to contour air gap	0.0000

☐ Link contours

Additional Settings

Part X shrink factor	1.0080
Part Y shrink factor	1.0080
Part Z shrink factor	1.0055

☐ Bypass seam placement method
☑ Minimize transition moves

Enhanced Surfaces

Visible surface rasters	0.0400
Visible surface raster air gap	0.0000
Surface max contours	0
Internal rasters	0.0400
Internal raster air gap	0.0000

Raster Fill

Part raster width	0.0400
Raster angle	45.0000
Contour to raster air gap	0.0000
Raster to raster air gap	0.0000

☐ Use parallel offset part rasters

Sparse Fill

Number of interior contours	1
Part sparse fill air gap	0.3000
Part sparse solid layers	4

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.

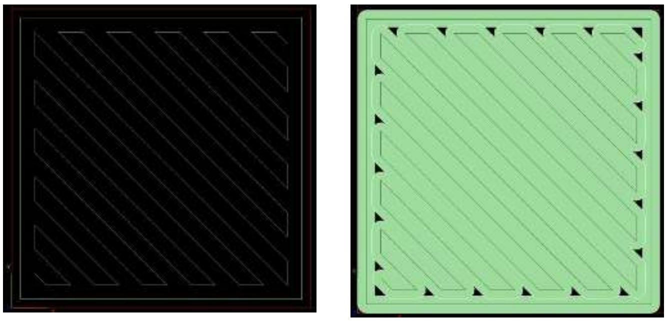
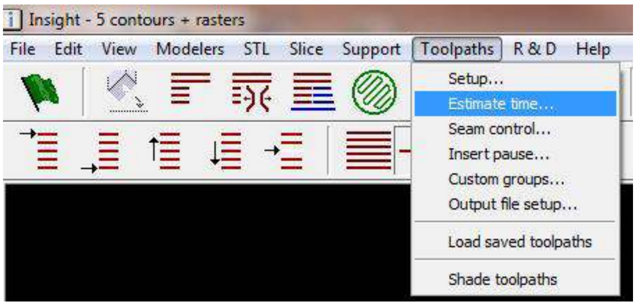


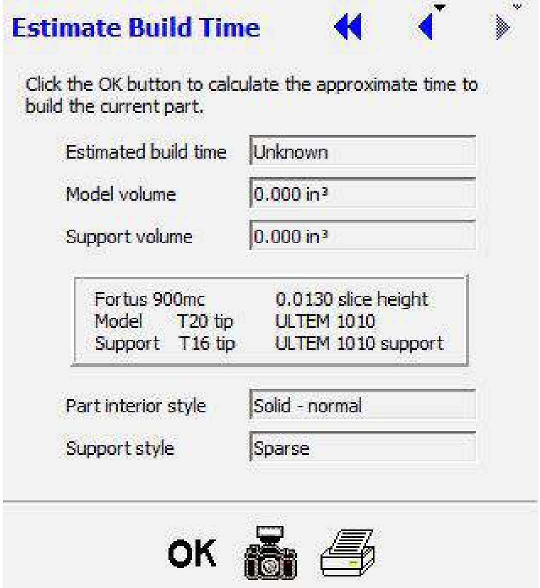
Figure 8-6: Before and after toolpaths have been shaded.

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.



Click the OK button to calculate the approximate time to build the current part.

Estimated build time: Unknown

Model volume: 0.000 in³

Support volume: 0.000 in³

Fortus 900mc 0.0130 slice height
Model T20 tip ULTEM 1010
Support T16 tip ULTEM 1010 support

Part interior style: Solid - normal

Support style: Sparse

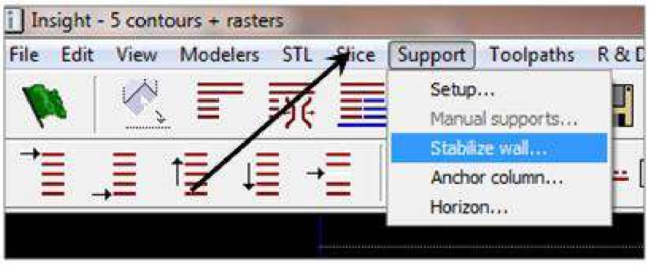
Select ***OK*** to run the estimation. The build time, model volume and support volume is displayed. A standard canister of FDM thermoplastic contains 92 cubic inches of material, which is helpful for estimating the amount of canisters required.

Stabilizing walls

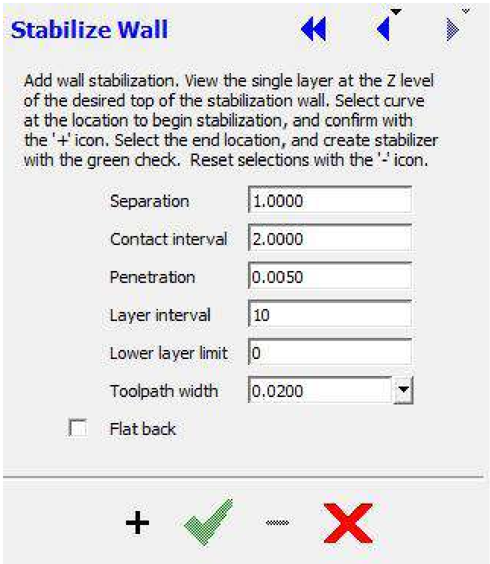
Vibrational forces within the machine can affect large, thin parts and may lead to dimensional inaccuracies. Adding stabilizing walls can prevent this. Stabilizing walls are sacrificial support columns, made out of model material, that help brace and anchor the part to the build sheet. Insight enables 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***.



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***.



Stabilize Wall

Add wall stabilization. View the single layer at the Z level of the desired top of the stabilization wall. Select curve at the location to begin stabilization, and confirm with the '+' icon. Select the end location, and create stabilizer with the green check. Reset selections with the '-' icon.

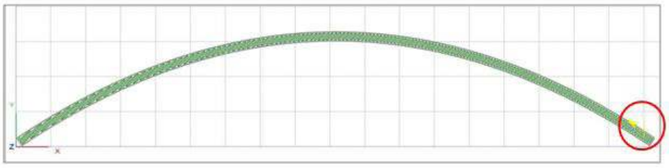
Separation	1.0000
Contact interval	2.0000
Penetration	0.0050
Layer interval	10
Lower layer limit	0
Toolpath width	0.0200

☐ Flat back

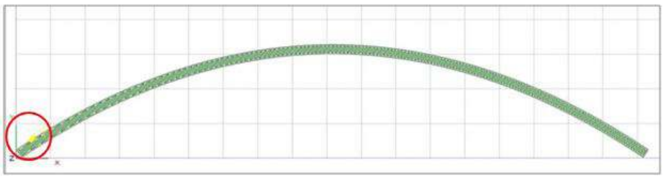
Once the features of the stabilizer wall have been established, select the layer that the stabilizer walls will go up to by entering a layer number. In the example below, the walls will reach layer 500 (see the red circle).

0.0000 0.0000 Z 4.9500 Layer 500 of 605

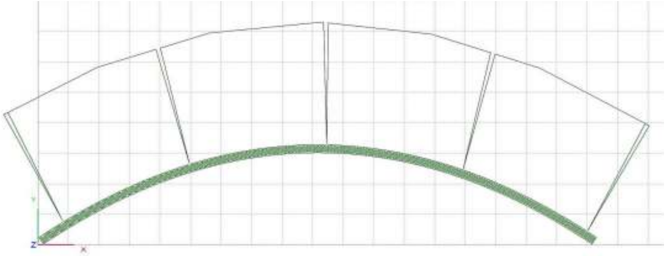
Select a point near the edge of the part as indicated below by the circled arrow. This will be the starting point of the stabilizer wall.



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.

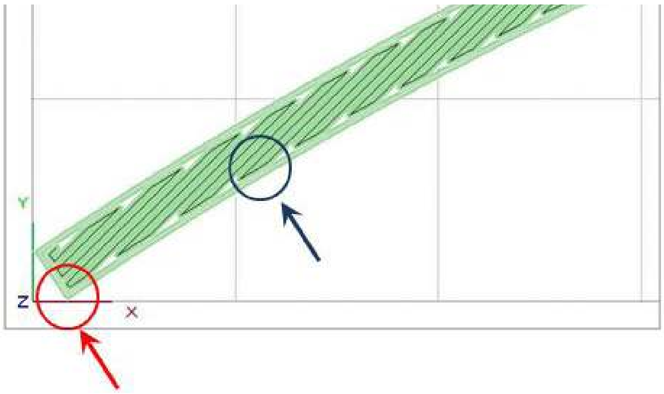


Select the green check mark. The stabilizer wall will appear and should look similar to the image below. The spacing and number of contact points may be different based on the chosen features.

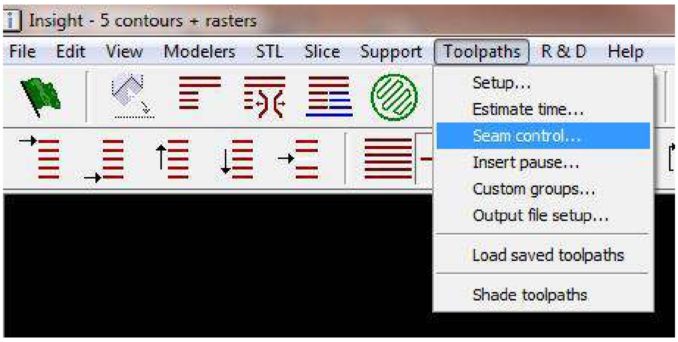


Seam control

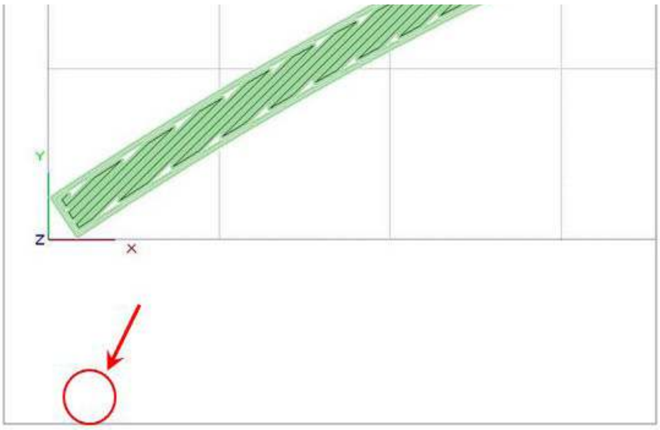
When the printer applies a bead of material, the place where it starts and stops is called the seam. In certain cases the seam can cause a slight blemish on the surface of the part, which can lead to an unacceptable composite tool surface. Solve this by moving the seam to a non-critical surface (typically 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.



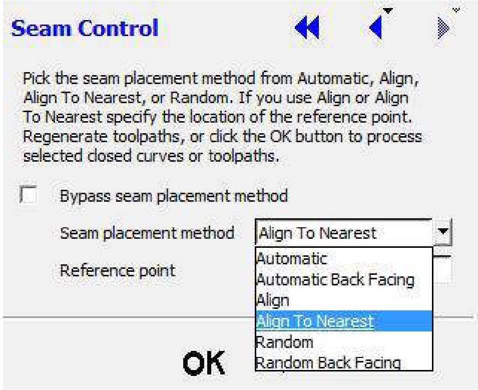
1. Move the seam by selecting Toolpaths, Seam Control.



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.



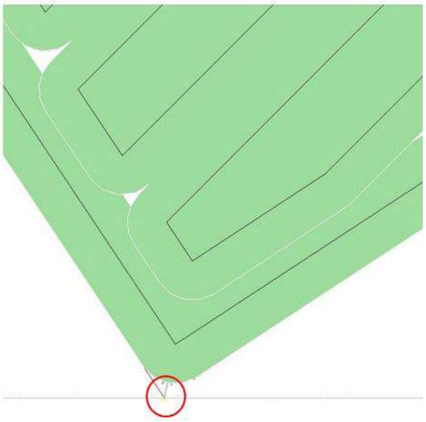
Seam Control

Pick the seam placement method from Automatic, Align, Align To Nearest, or Random. If you use Align or Align To Nearest specify the location of the reference point. Regenerate toolpaths, or click the OK button to process selected closed curves or toolpaths.

☐ Bypass seam placement method

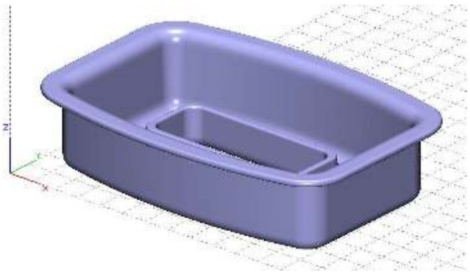
Seam placement method: Align To Nearest

Reference point



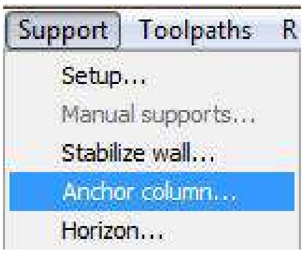
Using anchor columns to secure a part to the build sheet

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.



The best method to prevent parts from delaminating is to add anchor columns. An anchor column is a column of model material that directly connects the build sheet to the part.

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***



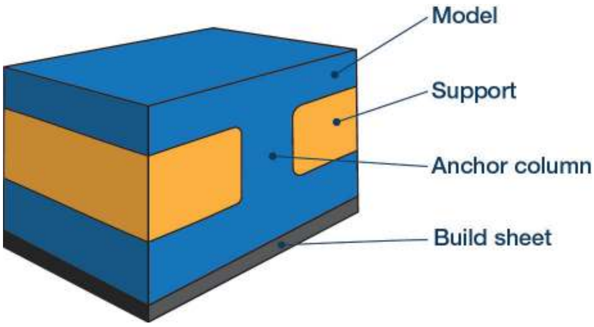
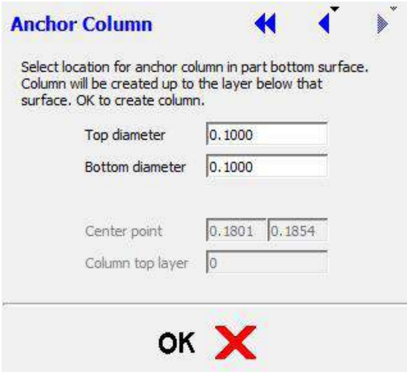


Figure 8-8: Cross-sectional view of an anchor column.

4. Select the diameter of the anchor column. 0.1 inch 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.



Anchor Column

Select location for anchor column in part bottom surface. Column will be created up to the layer below that surface. OK to create column.

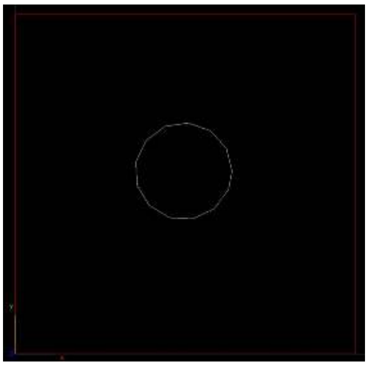
Top diameter 0.1000

Bottom diameter 0.1000

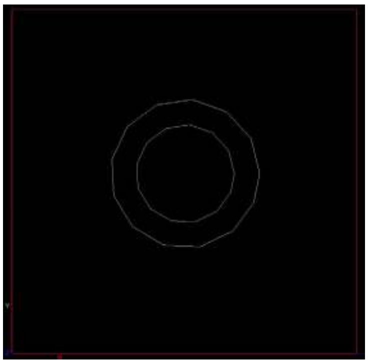
Center point 0.1801 0.1854

Column top layer 0

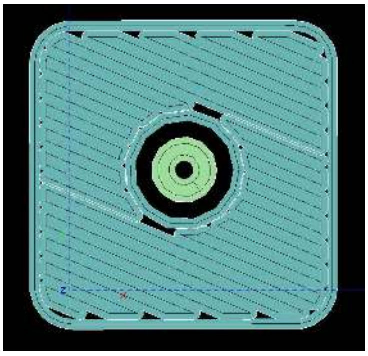
5. Select the location of the anchor column. A white circle will appear. Click OK. Note: The circle will disappear when the user clicks OK, but it has not been deleted.



6. Generate support and toolpaths.



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



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.

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

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

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

Figure A-2: Soldering nozzle attachment.

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

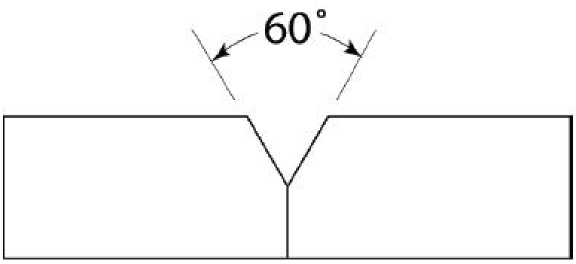


Figure A-4: V-groove channel for bonding two sections together.

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.

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-5: Filament welding two sections together.

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

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:

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 sanding the tool using 120-grit sandpaper. This abrasion allows for a good bond between the epoxy and the tool.

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

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 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.

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.

Figure B-4: Pouring epoxy on the tool.

Figure B-5: Wiping the tool.

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

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

11. Sand the tool using 120-grit sandpaper.

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

13. Sand the tool using 120-grit sandpaper.

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

15. Sand the tool using the following progressively finer sandpaper grits: 220, 320, 400 and 600. Wipe the dust off with a dry cloth between each sanding step.

Figure B-6: Sanding the tool.

16. Polish the tool by wet sanding with 800-grit sandpaper.

Figure B-7: Polishing the tool.

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

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

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 lay-up, form and cure composite materials in the production of composite parts. In this guide it is used synonymously with the terms lay-up 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.

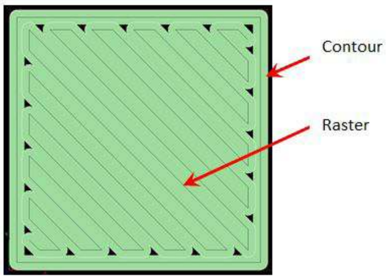


Figure B-7: Contours and rasters.

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.

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 x 2 x 3 feet. The second-largest offering in this family is the Fortus 450mc, with a build volume of 16 x 14 x 16 inches.

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

Figure C-3:
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.”

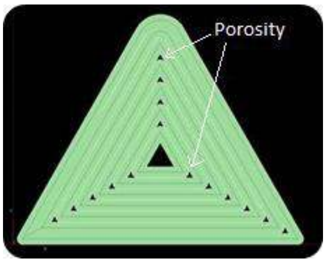


Figure C-4: Natural gaps between toolpaths resulting in porosity.

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.

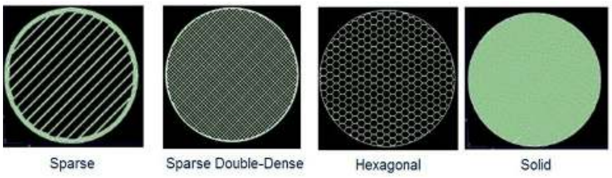


Figure C-6: Various fill patterns.

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.

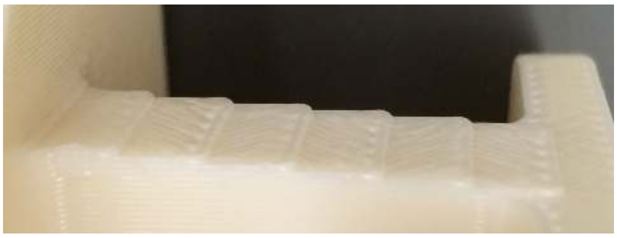


Figure C-8: Stair-stepping with a small slice height.

support material: A sacrificial material used to buttress overhanging features of parts during the printing process.

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.

APPENDIX D – 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
- ☐ 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

www.sys-uk.com info@sys-uk.com 01283 585955

ISO 9001:2008 Certified