



Sacrificial Tooling and Mandrels for Composite Part Fabrication

DESIGN GUIDE



stratasys[®]

THE 3D PRINTING SOLUTIONS COMPANY™

Sacrificial Tooling and Mandrels for Composite Part Fabrication

1 GENERAL	4
1.1 SCOPE	4
INTRODUCTION AND BACKGROUND	4
1.2 APPLICATION OVERVIEW	4
1.3 BACKGROUND AND PURPOSE	4
2 INDUSTRY RELEVANCE AND BENEFITS	5
2.1 DESIGN GUIDE OBJECTIVES	6
2.2 APPLICATION BEST FITS	6
2.3 KEY DESIGN CONSIDERATIONS	7
3 FDM MATERIALS	8
4 TOOL DESIGN AND CONSTRUCTION	9
4.1 DESIGN CONSIDERATION AND IMPACT	9
4.2 SPARSE STYLE	9
4.3 SHELL STYLE	9
4.4 DESIGNING A SPARSE TOOL	9
4.5 DESIGNING A SHELL TOOL	10
4.6 TOOL SEGMENTATION AND JOINING	10
4.6.1 Sectioning	10
4.6.2 Joining	11
5 INSIGHT FILE PROCESSING SOFTWARE	12
5.1 SELECT MODELER PARAMETERS AND IMPORT PART.	12
5.1.1 Importance of Build Orientation	12
5.2 GENERATE SUPPORTS	14
5.2.1 SMART vs. Sparse Support Considerations	14
5.2.2 Grow Supports	14
5.2.3 Stabilizer Walls	15
5.3 NON-STANDARD TOOLPATH GENERATION	16
5.3.1 Permeable Tubular Infill Patterns	16
5.3.2 Air Gap	17
5.3.3 Contours	17
5.3.4 Cap Layers	17
5.4 SEAMS	18
5.5 GLOBAL TOOLPATHS	18
5.5.1 Decreased build time	18
5.5.2 Increased part strength	18

Sacrificial Tooling and Mandrels for Composite Part Fabrication

5.6	SAVE .CMB FILE	18
5.7	PURGE TUBE	19
6	TOOL PREPARATION	20
6.1	SUPPORT REMOVAL	20
6.2	SMOOTHING SURFACES	20
6.3	SEALING AND RELEASE METHODS	20
7	LAY-UP CONSIDERATIONS	20
7.1	RESIN COMPATIBILITY	20
7.2	CONSOLIDATION METHODS	20
7.2.1	Vacuum/autoclave	20
7.2.2	Shrink tubing	21
7.2.3	Shrink tape	21
8	LAY-UP CONSIDERATIONS	21
8.1	COMPOSITE LAY-UP	21
8.2	CONSOLIDATED FDM SACRIFICIAL TOOLS	21
8.3	CURE COMPOSITE	22
9	STORAGE	22
10	TOOL REMOVAL	22
10.1	MANUAL CAP LAYER REMOVAL	22
10.2	DISSOLVE TOOL (WASH OUT)	22
11	CONCLUSION	23
12	USE CASE EXAMPLE	23
13	FREQUENTLY ASKED QUESTIONS	24
14	SACRIFICIAL TOOL READINESS CHECKLIST	25

1 GENERAL

1.1 Scope

This document provides principles for the design of sacrificial tooling for composites manufacturing using FDM® (fused deposition modeling) technology. Following the principles and requirements provided in this design guide for part design, creation, processing, lay-up and post-processing can ensure successful implementation.

INTRODUCTION AND BACKGROUND

1.2 Application Overview

Additive manufacturing has fundamentally changed the procedure for creating complex, hollow composite parts. While basic shapes with constant cross sections are easily manufactured using traditional composite manufacturing techniques, complex composite parts with hollow interiors present a unique manufacturing challenge.

ST-130™ FDM material allows for simple production of sacrificial tooling for hollow and complex composite parts using a thermoplastic material that dissolves in a basic (> 7 pH) solution. This eliminates many of the design and manufacturing limitations caused by using eutectic salts or other sacrificial tooling methods. Figure 1-1 shows an ST-130 sacrificial tool and the final composite part it produced.

FDM sacrificial tooling provides an efficient and cost-effective approach for producing complex composite parts with simplified tool removal, control over internal surface dimensions, and improved internal surface finishes. Composite laminated structures are laid up on or around a sacrificial tool and then cured. The tool is subsequently dissolved in a detergent solution. This method of using additive manufacturing to produce sacrificial tooling is straightforward and enables multiple iterations to be implemented quickly by changing the CAD model and building a new sacrificial tool. This also avoids the need to modify the hard tooling frequently required for traditional sacrificial tool fabrication.



Figure 1-1: FDM sacrificial tooling allows for easy production of tooling for complex hollow composite parts.

Creating a hollow composite part using a male tool, versus compressing it into the cavity of a female tool or bonding sections together, simplifies the lay-up process, reduces labor, and eliminates the need for additional tooling to support the manufacture of the sacrificial tool.

1.3 Background and Purpose

Traditional tooling used to fabricate complex composite parts includes:

- Collapsible tooling
- Inflatable and shape-memory bladders
- Wash-out tooling
- Multi-piece bonded assemblies
- Closed clamshell molds
- Clamshell molds combined with removable tooling

The existing methods present significant challenges due to complex assembly and removal of collapsible tools or the relative frailty of traditional wash-out mandrel materials. Sacrificial tooling made of eutectic salts, ceramics, urethanes or memory bladders requires additional production tooling and/or machining, which increases the overall time and cost of producing the final composite part.

2 INDUSTRY RELEVANCE AND BENEFITS

Sacrificial tooling using FDM technology provides a good business case for replacing existing methods by offering the following benefits:

- Eliminates the need for additional tooling or molds for sacrificial tool production
- Enables easy removal from the composite part
- Provides a robust process that can be transported and handled without damage
- Withstands the mechanical stresses of conventional lay-up procedures
- Offers significant design freedom and the ability to quickly iterate designs

Additionally, FDM tooling is dimensionally stable up to 250 °F (121 °C) and pressures up to 90 psi (at 210 °F; 98 °C).

These six fundamental drivers of additive manufacturing were assessed to determine how FDM sacrificial tooling could improve composite fabrication:

- **Economic low-volume production.** FDM sacrificial tooling is suitable for the low volumes and high product mix typical in the composites industry.
- **Increased product complexity.** FDM sacrificial tooling, similar to composite manufacturing using eutectic salt tooling, ensures that manufacturers can produce trapped-tooling shapes without the need for multi-piece tools.
- **Cost Effective product customization.** The ability to quickly make multiple design iterations allows composite manufacturers to meet customer requirements without the need to produce conventional lay-up tooling, saving the manufacturer and customer time, resources and capital expense.
- **Improved economic sustainability.** The benefit of printing sacrificial tools, jigs and fixtures, and production parts using the same FDM machine easily justifies the equipment and material cost, while developing capabilities and competitive advantage over other composite manufacturers.
- **Optimized supply chains.** Stratasys makes it easy to evaluate the technology to ensure it fits in a unique manufacturing environment. Stratasys Direct Manufacturing can produce sacrificial tooling to ensure the capability to meet requirements. Increasing production throughput justifies printing in-house as a means to reduce tool cost.
- **Increased part functionality.** FDM sacrificial tooling expands the portfolio within a composite-manufacturing facility and allows design by application need, not manufacturing limitations.

Incumbent processes cannot replicate all of these characteristics in a single technology. Traditional methods used to manufacture complex composite parts pose several challenges:

- Conventional tooling is difficult to handle due to frailty
- Traditional wash-out mandrels require secondary tooling for their production
- Clamshell tooling needs expensive capital equipment for manufacture
- Parts produced from clamshell molds still need post-process bonding

Table 2-1 compares FDM sacrificial tooling with other methods for making complex and hollow composite shapes.

	FDM SACRIFICIAL TOOLING	SILICONE BLADDER OVER FOAM	EUTECTIC SALT/SAND PATTERNS	CERAMIC	URETHANE	CLAMSHELL TOOLING
NO PRE-PROCESS TOOLING	✓	---	---	---	---	✓
HANDS-FREE REMOVAL	✓	---	✓	---	○	---
TEMPERATURE RESISTANT UP TO 250 °F (121°C)	✓	---	---	✓	○	✓
HIGH PRESSURE RESISTANT	✓	---	---	✓	○	✓
MINIMAL MANUAL LABOR	✓	○	○	○	○	---
NO BONDING REQUIRED	✓	✓	✓	✓	---	---
ROBUST TO SURVIVE ROUGH HANDLING	✓	---	---	○	✓	✓
SHORT LEAD TIME	✓	○	---	---	○	---
DIMENSIONALLY STABLE	✓	○	---	✓	✓	✓

Key: ✓ = Yes; ○ = Partial; --- = No

Table 2-1: FDM sacrificial tooling provides additional benefits and ease of use over traditional tooling methods.

2.1 Design Guide Objectives

The objective of this design guide is to provide best practices for the manufacture of sacrificial tools to fabricate composite structures. The intent is to bridge the knowledge gap between design-for-additive-manufacturing techniques and composite manufacturing.

The main objectives covered include:

- Engineering design considerations
- CAD design
- Insight™ software processing parameters
- Material properties
- Tool design and construction
- Lay-up considerations
- Post processing

2.2 Application Best Fits

FDM sacrificial tooling is best used for composite fabrication applications where the lay-up process would fully enclose the tool, making removal especially difficult using multi-piece, collapsible tooling or where the tooling is impossible to retrieve. FDM tooling replaces eutectic salt tooling and other difficult composite manufacturing methods and is well-suited for the manufacture of complicated tubes and ducts.

Business rational is achieved by the inherent low-volume production environment this application serves. Each tool is used once, eliminating the need for excess, revision-controlled inventory. Additionally, lead times for FDM tooling are significantly shorter than the existing technologies. With tooling lead time reduced from weeks to days, this process enables on-demand tooling fabrication, streamlining the supply chain.

FDM tooling is a good fit for composite applications cured at or below 250 °F (121 °C) and up to 90 psi if the temperature remains under 210 °F (98 °C). If a higher temperature cure cycle is required, it is possible to cure the composite part in two stages using an FDM tool. The first stage would be performed at lower temperature (and potentially higher pressure) to cure the part sufficiently for handling, followed by a second stage post-cure at a higher temperature to ensure complete cure of the material. Temperatures above the recommended limits during the second stage have the potential to damage the tool, but as long as the part is sufficiently cured, there should be no impact on final part quality.

2.3 Key Design Considerations

CONSIDERATION	REFERENCE
CURE TEMPERATURE AND PRESSURE	<ul style="list-style-type: none"> • Up to 250 °F (121 °C) with little to no pressure • 90 psi (620 kPa) up to 210 °F (98 °C)
COEFFICIENT OF THERMAL EXPANSION	<ul style="list-style-type: none"> • 59 x 10⁻⁶ in/in-°F (140-212 °F) • 98 x 10⁻⁶ in/in-°F (212-266 °F) • 106 μm/m-°C (60-100 °C) • 176 μm/m-°C (100-130 °C)
PROCESS PARAMETERS, BUILD TIME, AND TOOL STRENGTH	<ul style="list-style-type: none"> • See section 6 - Insight FDM File Processing Software
TOOL PREPARATION	<ul style="list-style-type: none"> • See section 7 - Tool Preparation
TOOL LIFE	<ul style="list-style-type: none"> • Being a sacrificial tool, each tool will produce a single composite part

Table 2-2

3 FDM MATERIALS

ST-130 is a proprietary thermoplastic blend used to build FDM sacrificial tooling for composite manufacturing. The material has been tailored specifically to address the tooling challenges of creating complex parts with hollow interiors commonly found within the composites industry. It dissolves in a detergent solution, facilitating hands-free tool removal.

ST-130 material characteristics that should be considered during tool design include the coefficient of thermal expansion (CTE) and the cure temperature and pressure pairings listed in Table 2-2. These pressure and temperature parameters have been determined through experimental testing using a wide range of geometries and may vary for specific geometries. Table 3-1 contains additional ST-130 material properties.

PROPERTY	TEST METHOD	ENGLISH		METRIC	
		XZ ORIENTATION (ON-EDGE)	ZX ORIENTATION (VERTICAL)	XZ ORIENTATION (ON-EDGE)	ZX ORIENTATION (VERTICAL)
TENSILE STRENGTH	ASTM D638	5600 PSI	1500 PSI	38.6 MPa	10.3 MPa
TENSILE ELONGATION	ASTM D638	1.04 %	0.2 %	1.04 %	0.2 %
FLEXURAL STRENGTH	ASTM D790	13800 PSI	2000 PSI	95.1 MPa	13.8 MPa
FLEXURAL MODULUS	ASTM D790	430 KSI	360 KSI	3.0 MPa	2.5 MPa
HDT @ 66 PSI	ASTM D648	250 °F		121 °C	
HDT @ 264 PSI	ASTM D648	226 °F		108 °C	
CTE	ASTM E831	55x10 ⁻⁶ /°F		99x10 ⁻⁶ /°C	
DISSOLUTION RATE* (WATERWORKS)	-	0.42 oz/min		12 g/min	
DISSOLUTION RATE* (ECOWORKS)	-	0.21 oz/min		6 g/min	
*Dissolution rates are dependent on solution quality, surface area in contact with the solution, fluid circulation, and part geometry.					

Table 3-1: ST-130 material properties.

4 TOOL DESIGN AND CONSTRUCTION

4.1 Design Consideration and Impact

Two main design styles should be considered when designing FDM sacrificial tooling. The first is a sparse interior fill pattern, designed as a solid body in CAD, which fills the entire interior area of the composite part and is used with vacuum bagging. The second is a shell style, consisting of a thin shell that is used with an inflatable bladder or through-tool envelope bagging.

4.2 Sparse Style

Sparse style tooling is the most common method used when creating FDM sacrificial tooling. Sparse tools take advantage of the automatic sparse interior fill pattern generated by Insight. The default interior fill pattern is a reduced-density fill specifically created for sacrificial tooling, which provides the optimal balance among build time, dissolution time and tool strength. The inner surface of the final composite part is the primary surface that is modeled using conventional CAD software. A solid body is designed from this surface by closing all open areas, making the tool a single solid part in CAD.

4.3 Shell Style

Consolidation methods that are coupled with envelope bagging or with an inflatable bladder benefit from a shell-style tool. Shell-style tools are thin-walled and used to control the inner surface finish of the composite part, while a clamshell mold is used to control the outer surface finish. An inflatable bladder or through-bagging method is then used to apply pressure to the inside of the tool to compress the composite into the clamshell mold. Shell tools use less build material and can be used for specific geometries (e.g. tubes, ducts and similar shapes) that allow for envelope bagging or inflatable bladders to support the internal cavity of the tool.

4.4 Designing a Sparse Tool

The design of a sparse tool starts with a CAD model of the final composite part as shown in Figure 4-1. The inner mold surface of the part is used as the primary surface for creating the sacrificial tool model.

The sacrificial tool model is created by extracting the primary surface from the composite part and filling the internal cavity of the composite model to create a solid body, shown in Figure 4-2. Hiding the composite model displays the resulting sacrificial tool model.

At this stage of the design process, faces where no composite material will be laid up should be extended beyond the part's perimeter to provide additional surface area for the lay-up process and trimming. Trim lines and other features can be added to the tool at this point as these features aid in post-processing the final composite part. Figure 4-3 shows an FDM sacrificial tool with extended faces, trim lines and locating features added. CTE scaling factors should also be taken into account during this design step.



Figure 4-1: FDM sacrificial tool design starts with a CAD model of the final composite part; in this case, a brake duct inlet.

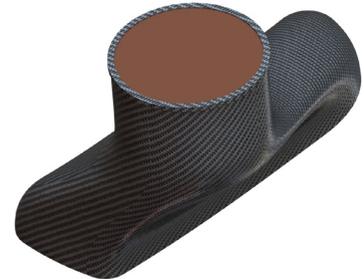


Figure 4-2: A solid body is created by extracting the interior surface of the composite part by closing open areas (top). Hiding the composite part reveals the sacrificial tool (bottom).



Figure 4-3: Extending open faces and adding trim lines and locating features assist in the lay-up and post-processing of the composite part.

4.5 Designing a Shell Tool

Designing a shell tool is similar to designing a sparse tool, with an additional shelling step. The same design procedure as detailed in section 4.4 - Designing a Sparse Tool should be followed for designing a shell tool. The solid CAD tooling geometry is then shelled, as shown in Figure 4-4, to create a body with a wall thickness that is appropriate for the pressures and temperatures used during the composite fabrication process. The wall thickness for the part shown in Figure 4-4 is 0.10 inch, which met the desired needs for the specific use case.

Prior to using Insight, the CAD model of the tool must be watertight, meaning it does not have open faces (holes or gaps) that create meshing and processing errors when converting to an .stl file and while being sliced in Insight. Once the FDM sacrificial tool design is complete, the model must be exported from the CAD software using the .stl file format. For information on exporting a high-quality .stl file format for additive manufacturing, see the, FDM best practice: “[Converting CAD to STL](#).” Figure 4-5 shows the triangulation results that are generated when exporting from CAD to .stl.

4.6 Tool segmentation and joining

4.6.1 Sectioning

Tools that are larger than the 3D printer’s build chamber must be sectioned into multiple components. Sectioning the part in CAD using a lap joint is the recommended method. A lap joint ensures proper surface area for bonding strength. Other joints such as dovetail, tongue and groove, or simple butt joints may be used as required by the geometry. Figure 4-6 shows a single geometry sectioned in two different ways to demonstrate the use of a lap joint for connecting sections together, post-build.

The geometries shown in Figure 4-6 use a 0.100 inch thick lip for the joint to provide adequate space for toolpaths and ensure the lip will not be damaged during assembly or curing operations. A 0.100 inch thickness is typically used for FDM lap joints but may vary depending on geometry or curing requirements. It is recommended to use a total clearance of 0.010 inch (0.005 inch around the perimeter of the joint and each face) between the joined components to allow room for adhesive and for the components to easily slide together as shown in Figure 4-7.



Figure 4-4: An additional shelling step creates a shell tool from a sparse tool.

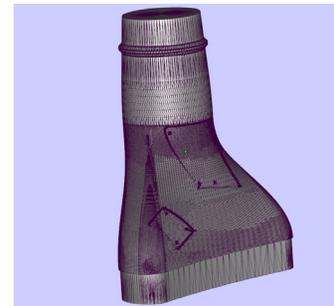


Figure 4-5: The CAD model faces are triangulated to create an .stl file.



Figure 4-6: Tools that are larger than the build chamber of the machine must be sectioned in CAD.



Figure 4-7: A 0.005 inch clearance between each face (a total of 0.010 inch) of components is recommended.

4.6.2 Joining

The parts can be joined together using the methods described in the Technical Application Guide: [“Comparison of Bonding Methods for FDM Materials”](#). The recommended bonding method for sacrificial tooling is hot-air plastic welding. Hot-air welding uses the same material as the FDM sacrificial tool to join the sections. It does not introduce any foreign materials into the tool, allowing complete dissolution during the removal process. The strength of the joint can be increased by coating the interior of the female lap joint with epoxy rather than hot-air welding. Coating the interior keeps the epoxy within the sacrificial tool and away from the composite lay-up area. The epoxy will not dissolve in the detergent solution, requiring manual removal after the tool has been dissolved, but will not stick to the composite part if the exposed areas of epoxy are cured and are properly coated in mold release. Figure 4-8 shows the bonding process of a sectioned tool.



Figure 4-8: Sectioning tools in CAD and bonding them together post-build using hot-air welding and/or epoxy allows for production of tools larger than the build chamber of the FDM machine.

5 INSIGHT FILE-PROCESSING SOFTWARE

5.1 Select Modeler Parameters and Import Part

ST-130 material is available on Fortus 450mc™ and Fortus 900mc™ 3D Printers. Select the correct modeler and ST-130 model material in the *Modeler Setup* menu, located under *Modelers, Setup* before importing the .stl file format into Insight (see Figure 5-1). Import the sacrificial tool .stl file using *File, Open*.

5.1.1 Importance of Build Orientation

The build orientation of an .stl file is extremely important. Build orientation will affect build speed, the amount of support material required, surface quality, and overall part performance. The end tool design intent and requirement must be considered when orienting the part to incorporate surface finish, build time, and other aspects affected by orientation. Slice the .stl file after the desired build orientation has been selected. Examples of three varying orientations are described in the following sections.

5.1.1.1 Vertical

A vertical orientation, shown in Figure 5-2, is recommended for this tool, because it will yield the best surface finish over the majority of the part and use the least amount of ST-130_S support material. The internal fill pattern is porous to promote fluid flow through the part, reducing dissolution time. Figure 5-3 shows how the sparse interior structure stacks up in the vertical orientation and allows for fluid to flow through a part. Additionally, cap layers are easily removed in Insight in the vertical orientation to further promote fluid flow. The process of removing cap layers is detailed in section 6.3.4 - Cap Layers.

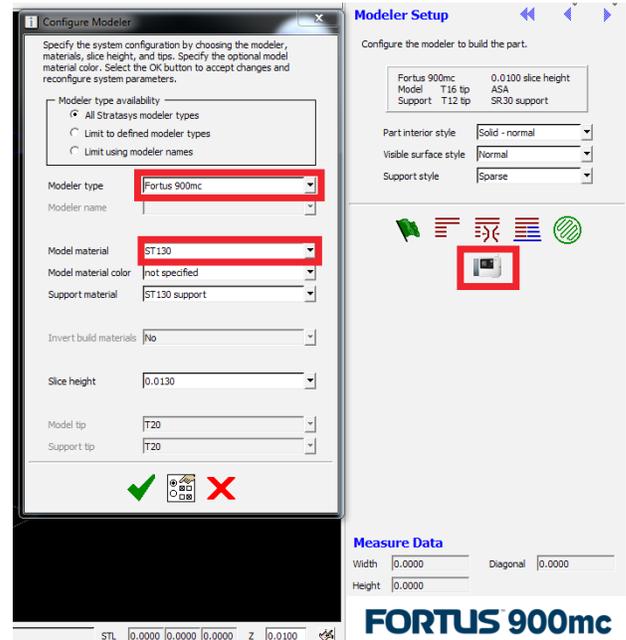


Figure 5-1: The printer setup menu within Insight allows for control over different printer and material types.

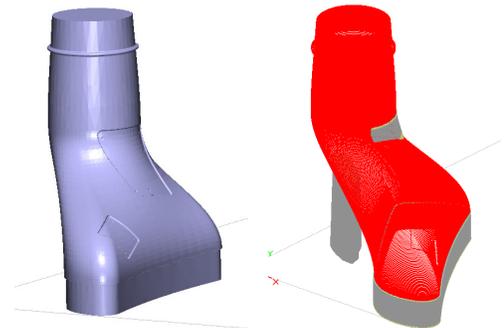


Figure 5-2: A vertical build orientation improves surface finish, promotes fluid flow, and provides adequate tool strength for this geometry.



Figure 5-3: The interior fill pattern custom created for FDM sacrificial tooling provides a path for fluid flow.

5.1.1.2 Flat (horizontal)

The flat (horizontal) orientation shown in Figure 5-4 will result in a sub-optimal surface finish for this specific geometry due to stair stepping, a step-like pattern produced on curved surfaces due to the geometrical constraints of the bead profile. Stair stepping requires more in-depth post-processing to achieve adequate surface finish compared with the vertically-oriented tool. The .stl file seen in Figure 5-4 also requires more support material beneath the tool relative to the other orientations.

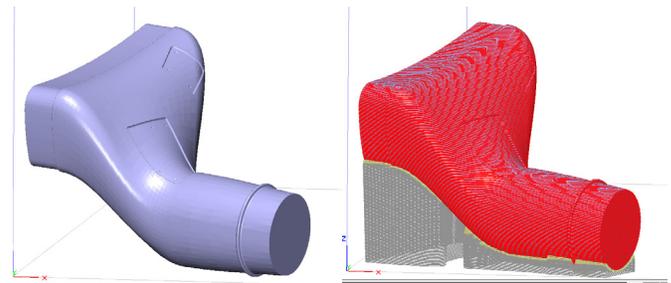


Figure 5-4: A horizontal orientation results in stair stepping on the surface (depicted in red) and requires a high volume of support material (depicted in gray).

5.1.1.3 Other Orientations

Parts may be repositioned in the build space to achieve the desired characteristics if they cannot be easily printed in vertical or horizontal orientations, or if these orientations result in undesirable part properties. To achieve the on-edge orientation seen in Figure 5-5, use the rotation menu shown in Figure 5-6. The X, Y and Z angles displayed are absolute references to the original orientation. To incrementally change the orientation, input the degree change and click the individual axes rotate buttons.

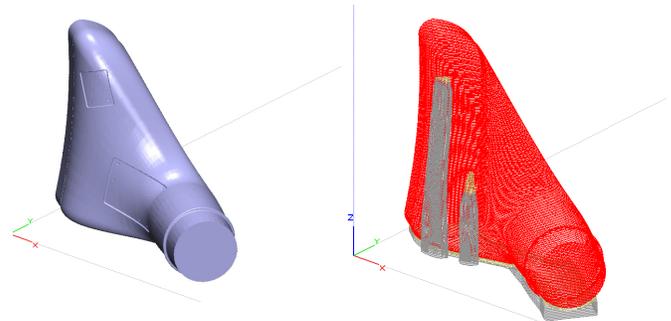


Figure 5-5: An alternate orientation reduces the amount of support material required, but creates stair stepping in multiple regions of the part.

In summary, surface finish, fluid flow, support material reduction, part strength and decreased build time are the main considerations to review when orienting a part for FDM sacrificial tooling.

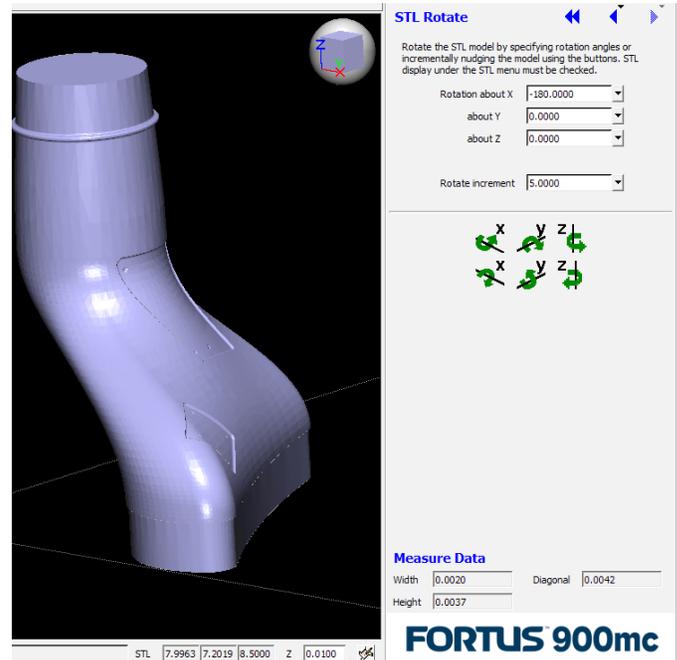


Figure 5-6: The rotation menu in Insight allows for unique orientations to be achieved by incrementally rotating the STL by a specified angle about the X, Y and Z axis.

5.2 Generate supports

Two different types of support structures are typically used when building a sacrificial tool. The default is the SMART option. SMART supports start with a small base and gradually grow outward as they progress upward as shown in Figure 5-7. The purpose of this support style is to reduce the amount of support material used and the build time using smaller support structures.

5.2.1 SMART vs. Sparse Support Considerations

SMART ST-130_S structures should be used by default unless tall, narrow support structures are needed to reinforce certain features on the sacrificial tool. A comparison of SMART ST-130_S and sparse ST130_S supports is shown in Figure 5-8. The sparse support gradient increases inversely to the Z-direction, providing a larger support base compared with SMART supports. As shown in Figure 5-8, the growth feature of the sparse supports grew the two support towers together automatically, providing a stable structure for overhanging geometries.

5.2.2 Grow Supports

The size of the base for sparse ST130_S supports can be controlled by modifying the *Grow supports* options in the support options menu. The Grow supports setting can be accessed on the *Support* tab, under *Setup*, by selecting the *more options* button. To obtain an ideal support structure, it is recommended that the *Grow supports* option be changed to *All* and the *Support growth angle* value be changed to 2.5, shown in Figure 5-9. The *Support growth angle* can be further modified to accommodate specific geometries.

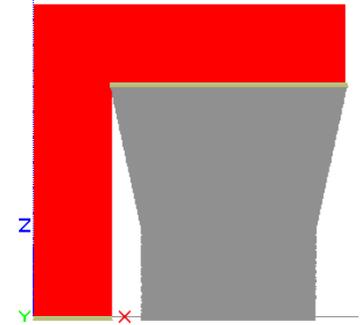


Figure 5-7: SMART supports grow outward as they increase in the Z-direction to reduce the amount of support material used.

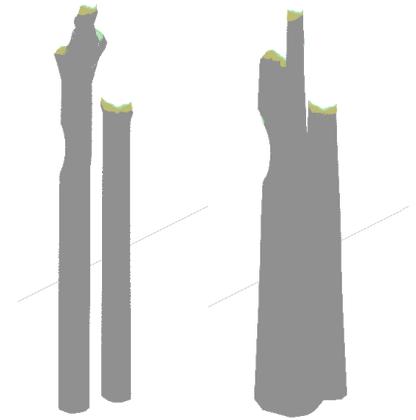


Figure 5-8: SMART supports (left) can lead to tall, thin support structures that are prone to tipping over. Sparse supports (right) grow inversely to the Z-direction, leading to a larger base

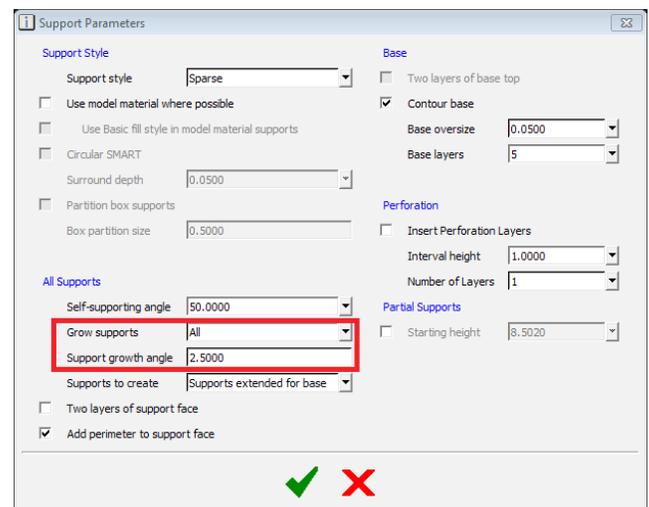


Figure 5-9: Changing Grow supports and the angle at which they are built can provide an ideal support structure for problem geometries.

5.2.3 Stabilizer Walls

Sacrificial tools that have tall designs with a small footprint on the build sheet may require additional support to enhance stability. Stabilizer walls are single-bead structures that slightly penetrate into the tool to help stabilize it during building, shown in Figure 5-10. The stabilizing structures are easily removed after the build is complete due to their perforations, similar to a perforated piece of paper.

A top-down view of a stabilizer wall and specific design details are shown in Figure 5-11. Parameters not detailed are recommended to be left as default.

- The **Separation** value controls how far the back of the wall is separated from the tool. It is recommended that the default value of 2.0 inches be increased to 3.0 inches to improve the stabilizer wall bond to the build sheet.
- The **Contact interval** value controls how much distance is between each contact leg of the stabilizer wall.
- The **Penetration** value controls how deep the wall penetrates into the tool.
- The units of these values are the same as the units selected within Insight (mm or inches).

Note: While processing the .stl file in Insight, the stabilizer walls will appear to have no support material under them. However, the FDM machine will automatically add the necessary base layers of support when the file is sent to the printer. The ST-130_S support material ensures the stabilizer walls attach to the build sheet, shown in Figure 5-12.

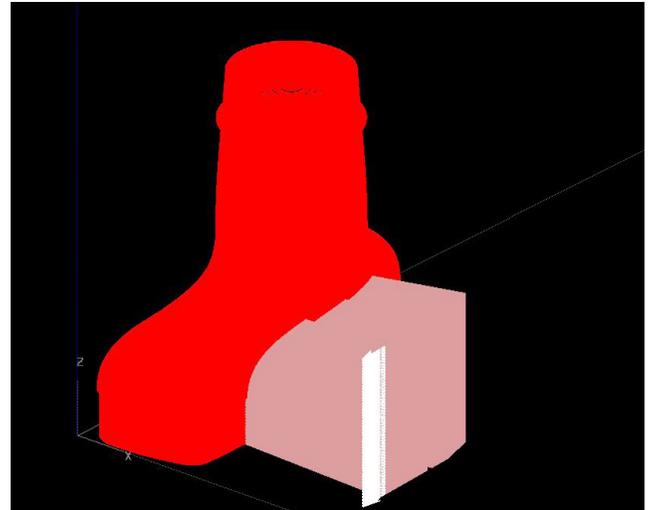


Figure 5-10: A stabilizer wall (salmon) provides additional support to the FDM sacrificial tool (red).

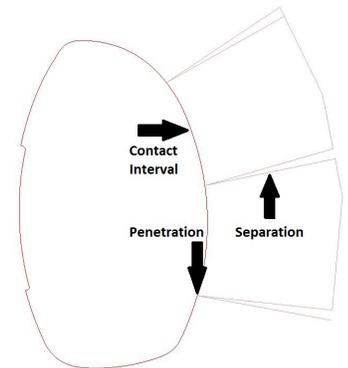
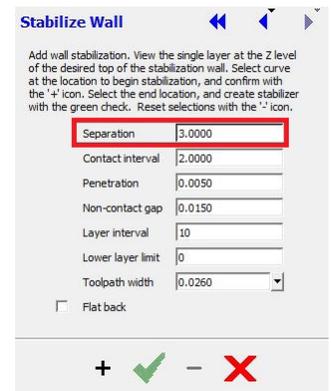


Figure 5-11: A stabilizer wall can be tailored to the specific tool geometry.



Figure 5-12: Two layers of ST-130_S support material are automatically generated underneath the stabilizer wall, ensuring that it adheres to the build sheet.

5.3 Non-Standard Toolpath Generation

Custom groups can be created to modify toolpaths from the default values in certain areas or for specific features of a tool. Varying parameters can be changed within the custom group menu, including:

- Part infill patterns
- Air gap spacing
- Number of contours
- Cap layers

Create a new custom group under *Toolpaths, Custom groups, New*, which will bring up the menu shown in Figure 5-13.

5.3.1 Permeable Tubular Infill Patterns

Two infill patterns are recommended for sacrificial tooling:

- Permeable triangular infill is the default fill pattern for ST-130 and is ideal for building quickly while minimizing material use.
- Permeable tubular builds a stronger tool but will require significantly more time to build.

Consolidation methods used and specific geometries may require the use of the permeable tubular infill pattern to prevent crushing the sacrificial tool. To change the infill pattern to permeable tubular, select the *Include in part sparse fill* box under the *Sparse fill control* tab, select the *Use alternate sparse fill pattern* box, and change the *Alternate sparse fill* style to *Permeable tubular* shown in Figure 5-14. Select all the layers of the part and add them to this group. Alternatively, select specific layers of the part to apply these parameters to those specific layers.

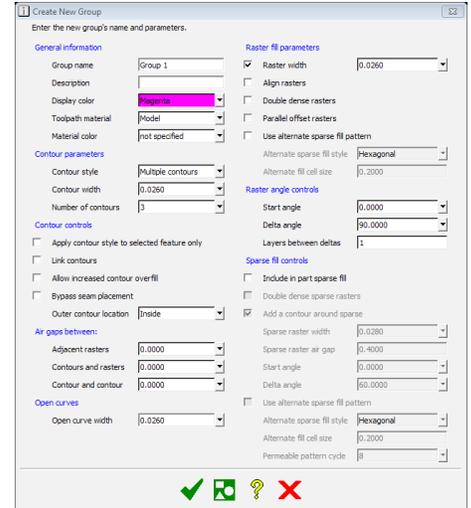


Figure 5-13: The custom groups menu allows certain toolpath parameters to be changed to decrease build time, increase tool strength or control other aspects of the tool.

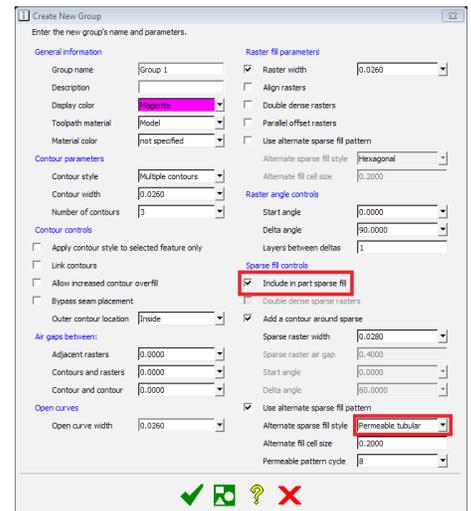


Figure 5-14: Changing the infill pattern to Permeable tubular increases tool strength at the expense of build time.

5.3.2 Air Gap

The toolpath air gap spacing can be changed to either increase the speed of the build or the strength of the sacrificial tool. Increasing the toolpath air gap spacing results in faster build and dissolution because less material is used to build the tool. However, increasing the air gap also decreases tool strength and may not be suitable for higher processing pressures. The toolpath air gap spacing can be changed by adjusting the value in the *Alternate fill cell size*, shown in Figure 5-15.

The air gap can also be reduced to provide additional strength to the tool. This will result in more material use and increased build and dissolution times. Select all the layers of the part and add them to this group. Alternatively, select specific layers of the part to apply these parameters to those specific layers.

5.3.3 Contours

Changing the number of contours has a similar effect as changing the toolpath air gap ratio. Increasing the number of contours increases the compression strength of the FDM sacrificial tool slightly by increasing the amount of material at the surface of the sacrificial tool. Likewise, decreasing the number of contours can result in decreased compressive strength and build times. Because the tool is under compression, the overall strength of the part is limited by the internal fill. For greater increases in strength, sparse air gaps should be modified rather than the number of contours. If desired, the number of contours can be changed by changing the value in the *Number of Contours* box shown in Figure 5-16.

5.3.4 Cap Layers

The removal of cap layers on areas where composite material will not be laid up can be useful for improving fluid flow, speeding tool dissolution. To remove cap layers, the tool must be oriented so that the cap layers being removed are relatively normal to the build plane. Position the tool within the build chamber as desired and slice the tool. Open up the *Custom groups* menu, create a new group, and generate toolpaths to see what layers are capped (default is 5 layers). Then select these layers normal to the surface, as shown in Figure 5-17.

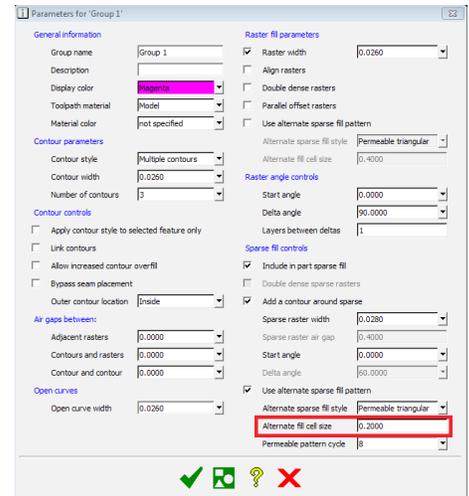


Figure 5-15: Modifications to the air gap spacing can be made to improve certain properties of the tool at the expense of other variables.

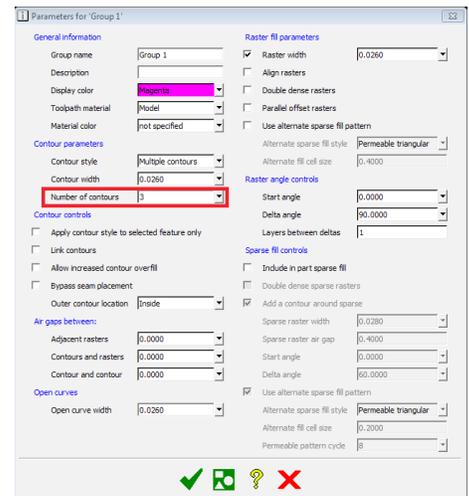


Figure 5-16: Changing the number of contours will change the tool strength and build speed.

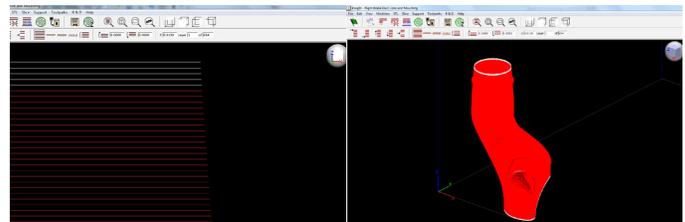


Figure 5-17: Removing capping layers using a custom group allows and promotes fluid flow through the interior of the tool during the removal process, reducing dissolution time.

Once the 5 layers have been selected, a new custom group must be created for them. To remove the cap layers in custom groups, select **Use Alternate Sparse Fill Pattern** under the **Raster Fill Parameters**, then select the intended fill pattern (default fill pattern is *Permeable triangular*), as shown in Figure 5-18.

After the custom group is created and the toolpaths have been generated, the cap layers will be removed and the infill pattern will continue through the respective layers, as shown in Figure 5-19.

5.4 Seams

A seam is defined as the start and stop location of a toolpath, as shown in Figure 5-20. A seam can potentially cause a blemish in the final composite part so its placement is important when processing the tool in Insight. A seam should be placed on the least important side of the tool or a face that can easily be smoothed by light sanding after the tool has been built. The seam placement on the part can be changed under **Toolpaths, Seam control**. A purge tube can be added to improve the seam smoothness, detailed in section 6.7 - Purge Tube.

5.5 Global Toolpaths

ST-130 material has multiple options that can be controlled using the global toolpath settings. Many of these settings are similar to those seen in custom groups. Any changes in this menu will affect all toolpaths not currently in a custom group. Default parameters are recommended for processing sacrificial tools, but may be modified to account for specific geometries or use cases. Test parts built with default parameters have passed rigorous testing at 90 psi, up to 208 °F (98 °C).

5.5.1 Decreased build time

A common desire when building sacrificial tools is to reduce the build time. Reduce build time by increasing the **Part sparse fill air gap**, which increases the cell size of the interior fill, and/or by reducing the number of contours on the part. This will require less material for the overall build, reducing build time.

Note: Making one or both of these changes will reduce the overall part strength.

5.5.2 Increased part strength

It is also a common desire to increase the overall strength of the tool. This is achieved by increasing the number of contours (which provides a slight increase in strength) or reducing the **Part sparse fill air gap** (providing a greater increase in strength). Performing these changes will lengthen the build time due to increased material use.

5.6 Save .cmb File

Once all supports, toolpaths and custom groups are satisfactory, generate toolpaths and save the job, creating a .cmb file. The .cmb file is then packed in Control Center™ and sent to the 3D printer for production.

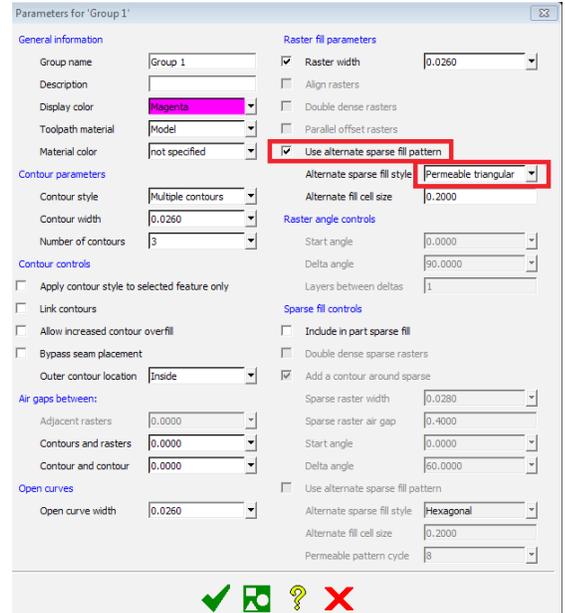


Figure 5-18: Select the appropriate options within the custom groups menu to remove cap layers in undesirable areas.

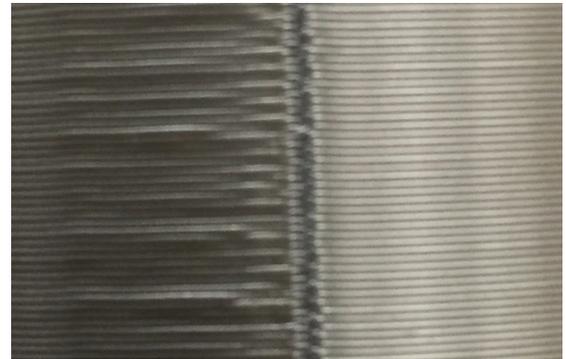


Figure 5-20: The seams should be considered during part processing to prevent blemishes on critical part surfaces.

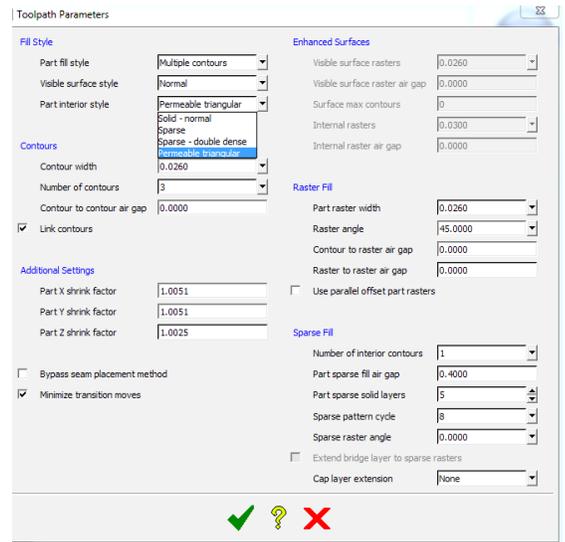


Figure 5-21: Changing global toolpath values will affect all layers not currently in a custom group.

5.7 Purge Tube

A material purge occurs when the printer is switching between model and support materials. The machine purges the material sitting idle in the tip while the alternate tip is building, which prevents building with degraded material.

To improve seam quality on the tool itself, it is strongly recommended to include a purge tube in each build. This circular structure is built alongside, but independently from, the tool or tools in the build chamber. It is the first geometry built after the material purge, to ensure material is flowing well out of the tip once it reaches a tool. Excess purge material that may sometimes remain on the tip after tip wiping is deposited on the purge tube rather than the tool, resulting in a cleaner seam. After the build, the purge tube is discarded.

The purge tube can be created in a new Insight window by drawing a circle under the *Edit > Draw > Circle* menu. Draw a two inch diameter circle and copy through Z to the height of the tallest sacrificial tool that will be in the pack using the *Edit > Copy curves through Z* menu. Create a custom group for the purge tube by selecting *multiple contours*, change the *Number of contours* to 3, and uncheck the *Raster width* box as shown in Figure 5-22.

Note: It is not recommended to use the automatic *purge part generation* available in Control Center for sacrificial tooling.

Using the specified custom group settings will create a purge tube that consists of three contours and no infill to prevent material waste as shown in Figure 5-23. Generate supports for the purge tube and save the file.

Pack the purge tube and all sacrificial tools using Control Center. Ensure that the purge tube is listed first under the pack details menu shown in Figure 5-24. Click and drag it to the top of the list to ensure that it is the first part in the pack to be built.

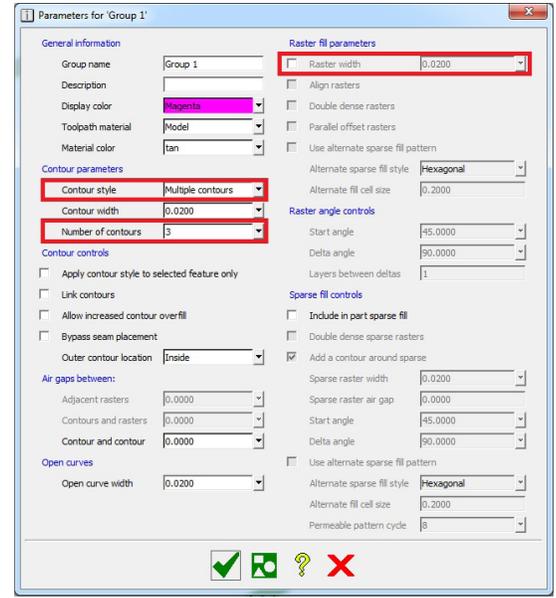


Figure 5-22: Building the purge tube using three contours and no rasters prevents unnecessary material waste.

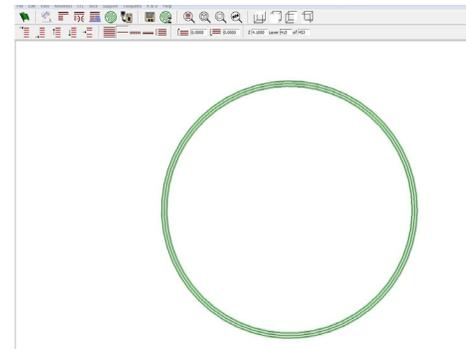


Figure 5-23: Using the detailed custom group settings, a circle with three contours is created.

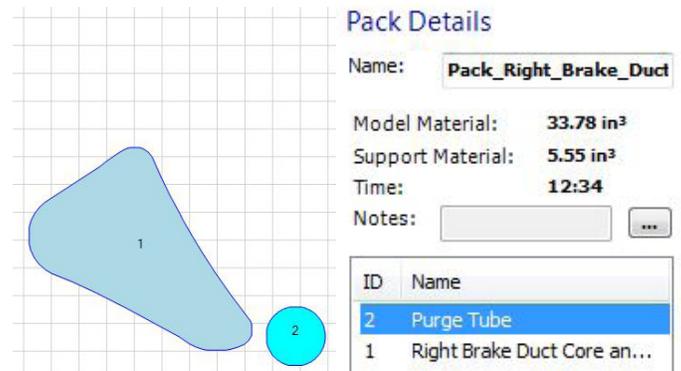


Figure 5-24: Packing a purge tube with sacrificial tools will improve seam quality (left). Placing the purge tube at the top of the pack list (right) ensures that it will be the first part built, acquiring the poor seam and preventing it from being transmitted to the tool.

6 TOOL PREPARATION

6.1 Support removal

ST-130_S support material is less brittle than ST-130 model material. Special care should be taken when removing support material in areas where there is a large interface between support and model material so as not to damage the tool. If the tool has cooled down after removal from the build chamber, it is recommended to reheat the tool up to 212 °F (100 °C) to ease support removal. Proper protection such as leather gloves should be used when handling the part at elevated temperatures to reduce the risk of injury.

6.2 Smoothing Surfaces

The tool may be hand-sanded with 120- to 600-grit sandpaper, shown in Figure 6-1, to improve surface finish. Optionally, a grit/soda blaster may be used, loaded with walnut shells or baking soda. See the FDM best practice: "[Media Blasting](#)."

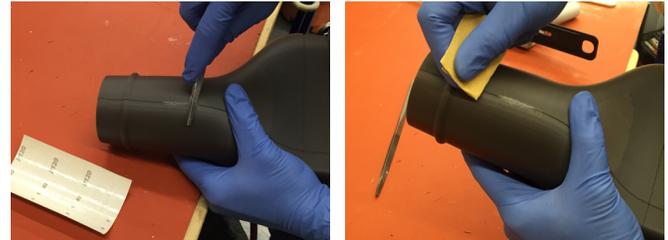


Figure 6-1: Tool surfaces and seams can be sanded or media blasted to improve surface finish.

6.3 Sealing and Release Methods

The resin from composite materials can impinge on the surface or seep into the layers of the sacrificial tool if it is not sealed properly, preventing complete dissolution of the tool. The composite part will have an irregular "lining" of undesired material and the resulting part could suffer from resin starvation if resin seeps into the tool. Sealers should be brushed or rolled onto the tool to prevent this. Zyxax Quickskin has been used successfully but equivalent sealers can also be used. Follow all manufacturer recommendations with one exception: Do not treat the surface of ST-130 with acetone as it will degrade the material. Isopropyl alcohol is recommended instead.

NOTE: It is recommended to prep tools no more than 3 days prior to lay-up. The chemicals in sealers and releases can cause mild degradation in the plastic and weaken the pressure resistance of the tool.

7 COMPOSITE CONSOLIDATION

7.1 Resin Compatibility

FDM sacrificial tools are compatible with most epoxy resin systems. However, the solution that dissolves the tool can attack and weaken certain polyester resins. Consult the resin manufacturer's specifications for compatibility with basic solutions.

7.2 Consolidation Methods

FDM sacrificial tools are compatible with the following consolidation methods.

7.2.1 Vacuum/autoclave

Using default Insight settings, vacuum bagging with the use of autoclave pressures has been successfully tested up to 90 psi (620 kPa) at 210 °F (98 °C). Above this temperature, the pressure must be reduced to prevent tool damage. Testing has been successful up to 250 °F (121 °C) with vacuum only and these conditions should not be exceeded. Survival of the tool under the above temperature and pressure combinations is dependent on the geometry of the tool. Part-specific testing may be necessary to establish acceptable parameters for certain tools.

Envelope bagging is a common method for consolidation of composites during the curing process. The entire assembly is enclosed in an airtight, pressure- and temperature-resistant bag after the composite is laid up. Envelope bagging is one of the most economical ways of applying pressure by vacuum. The entire assembly can also be placed in an autoclave for additional consolidation pressure if necessary.

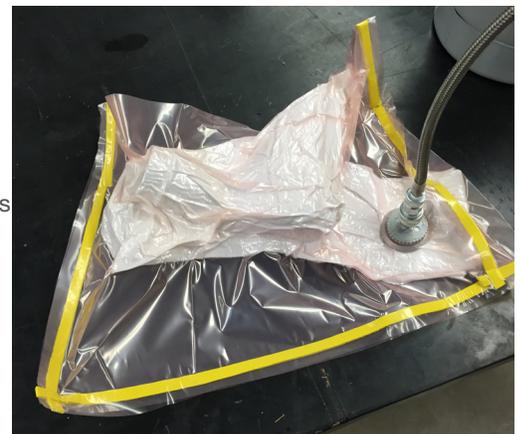


Figure 7-1: Envelope bagging is a common consolidation method, but does not allow for good control of the outer surface finish.

One drawback of the envelope bagging technique is a lack of outer surface finish control. The bag is not perfectly formed to the outer surface of the part and contains wrinkles, causing imperfections. During consolidation they transfer to the surface of the part and can be seen after curing.

7.2.2 Shrink tubing

Shrink tubing, shown in Figure 7-2, promotes a good finish on external surfaces. It is a low-pressure consolidation method that does not require a vacuum or autoclave. Composite parts with concave surfaces and/or tight bends are better suited to other methods since shrink tubing does not conform well to these features. Additionally, shrink tubing requires that all areas of the part fall within the minimum and maximum perimeter sizes of the tubing, making it being best suited for parts with relatively constant cross-sections.

7.2.3 Shrink tape

Similar to shrink tubing, shrink tape (Figure 7-3) is a low-pressure consolidation method that does not require a vacuum or autoclave. Although it offers a better exterior surface finish than vacuum bagging, the composite part will have an imprinted spiral pattern. Shrink tape conforms in areas that have concave surfaces and accommodates tight bends better than a sleeve of shrink tubing. Shrink tape also conforms well to geometries with large changes in circumference.



Figure 7-2: Shrink tubing is a simple consolidation method that can be used for specific geometries.



Figure 7-3: Shrink tape can conform to complex geometries, but will leave a spiral impression on the final composite part.

8 LAY-UP CONSIDERATIONS

8.1 Composite Lay-up

Perform the composite lay-up or winding (Figure 8-1). Once the tool is prepared for lay-up, no special considerations are needed for FDM sacrificial tooling.

8.2 Consolidated FDM Sacrificial Tools

Consolidate the composite part (Figure 8-2) using one of the methods described in section 7.2 - Consolidation Methods



Figure 8-1: Follow standard lay-up procedures when using FDM sacrificial tooling.



Figure 8-2: Envelope bagging is commonly used with FDM sacrificial tooling.

8.3 Cure Composite

FDM sacrificial tools are compatible with both room-temperature and elevated-temperature curing methods. A room-temperature cure option preserves tool strength and minimizes the effects of thermal expansion on the composite part. To prevent damage to the sacrificial tool during an elevated-temperature cure, the following temperature and pressure combinations should not be exceeded: 210 °F (98 °C) at 90 psi, and 250 °F (121 °C) with little to no compression (i.e., vacuum only).

If the composite part allows, a resin system with a higher cure temperature can be used in a two-stage cure approach:

- Perform an initial cure at a lower temperature within the recommended temperature and pressure combinations above.
- Dissolve the tool (section 11 - Tool Removal).
Note: Optionally, the tool can remain in the lay-up during post-cure, provided the temperatures do not exceed 392 °F (200 °C) or the tool will begin to burn/char.
- Complete the post-cure at the higher temperature.

9 STORAGE

After the FDM sacrificial tool has been created, it must be stored at ambient temperatures and kept away from chemicals.

10 TOOL REMOVAL

10.1 Manual Cap Layer Removal

Exposed faces of the sacrificial tool should be broken open to expose the sparse interior to the detergent bath (Figure 10-1) if they were not previously removed in Insight during file preparation (section 6.3.4 - Cap Layers). This allows for quicker access to the interior of the tool and accelerates the dissolving process. One option to expose the internal fill is to drill several holes through the cap layers with a large diameter drill bit. Sections of the tool can be manually extracted if they are easily accessible. This reduces the amount of material that needs to dissolve in the detergent solution.

10.2 Dissolve tool (wash out)

To dissolve the tool, follow all guidelines laid out in the best practice: “[FDM Support Removal](#)”, Section 2 for WaterWorks solution or Section 3 for EcoWorks solution.

Note: Circulation of the detergent is important for reducing dissolution time. In some circumstances, an auxiliary pump can be placed near exposed areas to increase the rate of dissolution. Common configurations that benefit from increased circulation are long, narrow passages and passages with multiple or tight bends. If possible, pump detergent directly into the tool using flexible tubing to reduce the dissolving time.

Monitor the process and remove the part when the tool has completely dissolved (Figure 10-2). Complete the process with a thorough rinse, using tap water to remove all of the detergent solution.



Figure 10-1: Tooling faces without composite material should be broken open or built without cap layers to promote fluid flow through the tool.



Figure 10-2: FDM sacrificial tooling easily washes out in a detergent solution.

11 CONCLUSION

FDM sacrificial tooling provides a simple, cost-effective alternative method for producing complex composite parts with hollow interiors. Traditional tooling methods require complex assembly and removal or involve relatively frail wash-out tools that can be damaged during handling or lay-up processes. They may also require additional tooling for their creation. In contrast, FDM sacrificial tooling provides hands-free tool production and a robust tool that easily dissolves in a detergent solution.

12 USE CASE EXAMPLE

The part shown throughout this design guide is an implementation example of FDM sacrificial tooling. The part is a brake duct inlet used in automotive racing that provides air from the front bumper of the car to the brakes to keep the calipers and rotors cool from the vast amount of braking heat generated during the race. Figure 12-1 shows where the duct is located on the vehicle.

The duct needed a smooth interior surface finish for unimpeded airflow and was too small to be fabricated using a female mold, which led to the use of sacrificial tooling. Without sacrificial tooling, the inlet would have been produced in two pieces using clamshell tooling and then bonded together, adding extra time and fabrication steps. Instead, the FDM sacrificial tool was produced in 9.75 hours using a Fortus 900mc 3D Printer under lights-out manufacturing. This provided the capability to subsequently produce a lightweight, single-piece, composite brake duct.

Lightweight parts push the motorsports industry to use composites and the limited number of composite parts produced per year makes this application a best fit for the use of FDM sacrificial tooling.

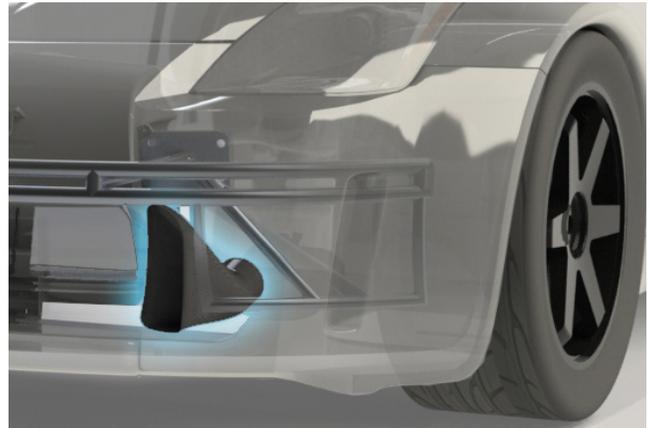


Figure 12-1 - The composite brake duct produced using FDM sacrificial tooling draws air from the front of the car and routes it to the brakes.

13 FREQUENTLY ASKED QUESTIONS

I have used FDM in the past to build “soluble cores” (sacrificial tooling). What has changed?

The new ST-130 solution was created to run as reliably as any standard FDM model material, therefore any previous actions regarding particular part placement in the build chamber, model/support inversion in the Insight software, or custom grouping are no longer necessary. Additionally, new, optimized fill patterns were created and tested for improved material removal (dissolution rate) while delivering improved processing performance regarding temperature and pressure settings during cure (e.g., autoclave consolidation pressures up to 90 psi at 210 °F [620 kPa, 99 °C]).

What approach should be used to produce the highest quality sacrificial tools?

Standard default (or green flag) settings have been tuned to produce the best all-around output from the new ST-130 material. After importing and orienting a part to best suit the desired characteristics, using the green flag settings (options) removes the need for any additional tips or tricks in the setup to produce a high-quality part.

Do tools built with ST-130 need stabilization during the FDM build?

As with any high-aspect-ratio parts, a stabilizer wall is recommended to prevent unnecessary failures due to machine vibrations. Within the Insight menu (version 10.8 or higher) *Support*→*Stabilizer Wall*→*Separation*, it is recommended that you increase the value from the default of 2.0 inches to 3.0 inches, which increases the standoff distance from the part to improve the stabilizer wall bond to the build sheet.

When building composite parts, what temperature and pressure can I use for ST-130 molds and mandrels?

The improved and optimized fill pattern, *Permeable Triangular*, for ST-130 allows parts to be processed at 250 °F (120° C) with minimal consolidation pressure. Although all results are geometry-dependent, testing confirmed that ST-130 will maintain mechanical integrity through final composite part production at full vacuum (equivalent to 14.7 psi [101.35 kPa]) and 250 °F (121 °C) processing conditions. For processing conditions up to 210 °F (99 °C), ST-130 parts have withstood pressures in excess of 90 psi (620 kPa), but could require additional toolpath customization to increase mechanical strength for some geometries.

Is it recommended to use purge towers while building tooling in ST-130?

The default purge tower in Insight has been observed to debond from the build sheet at heights greater than 3 inches for this material pairing. As a result, a manual purge tower procedure was added to the this design guide (section 5-7) to overcome this issue. Purge towers are recommended when stringing or excess seam ooze is observed.

14 SACRIFICIAL TOOL READINESS CHECKLIST

Design

- Curing method will not exceed recommended temperature and pressure pairings
- Shell or sparse style tool chosen
- Initial tool design complete
- Trim lines and locating features added
- CAD model is a single solid body
- CTE factored into design
- Tool sectioned with incorporated joints and proper clearances if larger than build chamber

Insight Software

- Proper machine and ST-130 material selected
- Tool properly oriented based on surface finish, fluid flow and strength
- Tool sliced
- Supports generated
- Stabilizer walls added (optional)
- Custom groups created (optional)
- Cap layers removed (optional)
- Seams located on least critical surface (optional)
- Toolpaths generated
- File saved
- Purge tube created (optional)

Post-Processing

- Supports removed
- Surfaces sanded to improve finish (optional)
- Tool bonded together (if produced in multiple pieces)
- Tool sealed



E INFO@STRATASYS.COM / STRATASYS.COM

HEADQUARTERS

7665 Commerce Way, Eden Prairie, MN 55344
+1 800 801 6491 (US Toll Free)
+1 952 937 3000 (Intl)
+1 952 937 0070 (Fax)

2 Holtzman St., Science Park, PO Box 2496
Rehovot 76124, Israel
+972 74 745-4000
+972 74 745-5000 (Fax)

ISO 9001:2008 Certified

© 2016 Stratasys. All rights reserved. Stratasys, Stratasys signet, FDM and Fortus are registered trademarks of Stratasys Inc. Fortus 450mc, Fortus 900mc, Insight, ST-130 and Control Center are trademarks of Stratasys, Inc. All other trademarks are the property of their respective owners, and Stratasys assumes no responsibility with regard to the selection, performance, or use of these non-Stratasys products. Product specifications subject to change without notice. Printed in the USA. DG_FDM_SacrificialTooling_0417a