End-use parts include all components, sub-assemblies and products from any manufacturing process. These parts are the output of both series production and custom manufacturing.

Additive manufacturing is emerging as a viable option to molding, machining, forming and casting of end-use parts. These traditional processes generally rely on tools and dies when production volumes are moderate or high. For low quantities, conventional approaches often rely on manual or semi-automated machining, forming and fabrication.

FDM, an additive manufacturing technology, eliminates tools and dies while automating the production process. Substituting FDM for the traditional manufacturing processes will substantially reduce lead times and costs. It also makes production a simple, unattended and virtually labor-free operation.

For those willing to break free of the status quo, FDM creates new design opportunities. Parts can be designed for optimal performance and ergonomics since additive manufacturing places few constraints on design configurations. And designs are fluid. There is no investment in tooling, so part configurations can change with each production run.

CONTENTS

1. OVERVIEW........................................................................................................................................2
   1.1. Application ........................................................................................................................................2
   1.2. FDM is a best fit when .........................................................................................................................2

2. ROLES & STRATEGIES.........................................................................................................................2
   2.1. Pilot production (Figure 6) ..................................................................................................................3
   2.2. Bridge-to-production ...........................................................................................................................3
   2.3. Full production (Figure 7) ..................................................................................................................3

3. DESIGN FOR FUNCTION (CAD) .......................................................................................................3
   3.1. Integrated design (consolidation) (*) (De) .........................................................................................4
   3.2. Integrated features (*) (De) ...............................................................................................................4
   3.3. Maximize fill density (**) (St) ...........................................................................................................5
   3.4. Self-supporting angles (***)(Ti) ........................................................................................................6
   3.5. Variable density (***)(St, Ti) .............................................................................................................6

4. SELECTIONS: MATERIAL AND SLICE ............................................................................................8
   4.1. Material selection (*) (Ae, St, Ti) ......................................................................................................8
   4.2. Slice selection (*) (Ae, St, Ti) ..........................................................................................................9

5. FILE PROCESSING (INSIGHT) ...........................................................................................................10
   5.1. Part orientation (*) (Ae, St, Ti) ..........................................................................................................10
   5.2. Build styles (*) (St, Ti) .....................................................................................................................11
   5.3. Visible surface style (*) (Ae, Ti) .......................................................................................................13
   5.4. Custom group: fill density/porosity (*) (St) .....................................................................................13
   5.5. Custom group: gradual slopes (*) (Ae) .............................................................................................15
   5.6. Custom group: supports made with model material (**) (Ti) .......................................................16
   5.7. Custom group: wide supports (**) (Ti) ............................................................................................17

6. THROUGHPUT & PRODUCTION PLANNING ................................................................................19
   6.1. Packing the build sheet (*) (Ti) .......................................................................................................20
   6.2. Nesting parts (***)(Ti) ....................................................................................................................20

7. RECAP - CRITICAL SUCCESS FACTORS.................................................................................21
   7.1. Leverage advantages of additive manufacturing ................................................................................21
   7.2. Invest time accordingly ....................................................................................................................21
   7.3. Optimize jobs for .............................................................................................................................22

Legend:
*  All users
** Intermediate users
*** Advanced users

De  Design optimization
Ae  Aesthetic optimization
St  Strength optimization
Ti  Time optimization
1. OVERVIEW

1.1. Application:

FDM is an alternative manufacturing method for end-use parts including finished goods and sub-assemblies. It may be substituted for processes that use molding, machining, casting and forming.

1.2. FDM is a best fit when:

- Complex designs.
  - Feature-laden.
  - Organic, flowing shapes.
  - Consolidated assemblies.
- Small to moderate size.
  - Ideal (but smaller or larger are possible):
    - > 1 in. x 1 in. x 1 in. (25 mm x 25 mm x 25 mm)
    - < 16 in. x 14 in. x 16 in. (405 mm x 355 mm x 405 mm)
  - Low-volume or custom manufacturing.
  - Pilot, bridge and full production.
  - Quantities:
    - Dependent on part size and volume (inverse relationship).
    - Typically 1 to 1000 pieces.
- Many product revisions or variations.
  - Use when design modifications and iterations are likely.
  - Use when there are a large number of configuration options for a small volume of products.
- Materials are compatible.
  - Thermoplastic materials—confirm suitability of mechanical properties, electrical properties, chemical resistance and thermal resistance.
- Accuracy is suitable.
  - Without secondary machining, use with applications that require tolerances of 0.005 in. (0.13 mm) or higher.
  - In many cases, secondary machining can be used to improve the accuracy for higher tolerance requirements.

2. ROLES & STRATEGIES

This guide presents tips and techniques that leverage the advantages of FDM while optimizing part quality and operational efficiency. The degree to which these are applied is dependent on the role that the end-use parts will play. If used as an interim, stopgap solution, minor adjustments may be all that is warranted. On the other hand, if FDM will be the long-term production process, any time invested in design and preparation will be justified by the gains in quality and productivity.
2.1. Pilot production (Figure 6).

FDM is the short-term solution for end-use part production. After the pilot is complete, parts are produced through a traditional manufacturing process. In this capacity, design modifications are undesirable and gains from process refinement are short-lived.

For the pilot production, use the part design that suits the conventional production method. The “freedom of design” that additive manufacturing offers means that changes may be unnecessary.

- Leave part design unchanged.
- Make minor process adjustments.

2.2. Bridge-to-production.

In this role, FDM is an interim solution used to bridge the time span between product launch and production line start-up. For example, FDM could be used to produce parts while awaiting delivery of an injection mold.

Since production will be handed off to a traditional manufacturing process, design alterations are not desirable. However, if the time span and production quantity are significant, minor design alterations may be incorporated. This production quantity is also likely to warrant a larger upfront investment of time in optimizing FDM build parameters.

- Consider minor design changes.
- Tune process to optimize performance.

2.3. Full production (Figure 7).

When FDM will be the only method of end-use part production, optimize every aspect of the part and its production. This investment will maximize product performance and throughput while minimizing cost.

Optimization includes designing the part purely for its function; ignore established design for manufacturability rules.

- Complete redesign is justifiable.
- Fine-tune process for maximum optimization.
- Continue to redesign throughout the product life cycle.

3. DESIGN FOR FUNCTION (CAD)

A simple approach to apply FDM to end-use part manufacturing is to use existing designs and current design rules. However, this may be an inefficient approach that does not capitalize on the advantages of additive manufacturing. If left unchanged, the rules that govern designs for conventional processes are artificially imposed on FDM parts.

Whenever time allows, it is best to start with a fresh design that builds from the design freedoms offered by additive processes and optimizes part production.
3.1. Integrated design (consolidation). (*) (De)

Convert assemblies into single parts. Components are often dissected into many pieces in order to make conventional manufacturing processes feasible and affordable (Figures 8 and 9). This is unnecessary with FDM.

If reproducing an existing part, start with a redesign that consolidates as many components as possible into one piece. If designing a new item, create it as one piece (Figure 10); only split off parts when it is advantageous to the product’s operation or maintenance.

Integrated design has many advantages, including:

- **Design for function.**
  - Focus on the task that the tool will perform. Optimize the design for its function rather than the processes used to make it.
  - Eliminate tolerance challenges.
    - Holding tight tolerances is costly but avoidable. If two mating parts are combined into one, all concerns over—and all costs for—controlling the tolerances where they mate are eliminated.

- **Eliminate assembly time.**
  - Assemblies, obviously, must be assembled, and this takes time. Consolidate all parts into a single piece to eliminate the time needed for assembly.

- **Absorb costly neighbors.**
  - Look beyond the sub-assembly to find expensive or long lead time parts in the vicinity. Including them in the consolidation further reduces costs and production time.

- **Decrease BOM count.**
  - Smaller part counts decrease the time and expense for managing and warehousing inventory.

3.2. Integrated features. (*) (De)

There are two aspects to integrated features: adding features and inserting hardware.

3.2.1. Adding features:

With traditional manufacturing processes there is usually a cost associated with every feature added to a part. This is not true with additive manufacturing, especially when material is removed.

In the design of an FDM part, consider adding features that improve performance, reduce build times and reduce material costs. For example:

- Add pockets, channels and holes (Figure 13).
  - Removing mass from the part will decrease weight, material consumption and build time, in most cases.
• Add ribs, bosses and gussets.
  − There will be a slight (in most cases) increase in build time and material cost. However, the impact is far less than that of a traditional process when the feature adds a tool change, setup or side pull.

• Add embellishments (Figure 14).
  − Incorporate part numbers, storage locations, alignment aids or operational instructions directly on the part.

3.2.2. Inserting hardware (Figures 15 - 17):

As with traditional manufacturing, hardware may be added in a secondary operation. For example, threaded inserts can be press-fit into location holes added to the CAD model or machined after the FDM part has been built.

Another alternative is to mimic insert molding. In the CAD model, add a pocket or hole that will contain a sensor, RFID tag, bushing or threaded insert. During the FDM build process, pause the job, place the item in the part, and resume the build. The hardware is now integrated with the FDM part.

3.3. Maximize fill density. (**) (St)

FDM parts use an internal raster fill bounded by external contours. For thin features—those less than 0.060 in. (1.52 mm)—there may not be enough space for the internal rasters (Figure 18). This may result in an internal void. These voids may be eliminated in Insight (see Section 5.3) or prevented in CAD.

In the CAD model, avoid tapers or draft angles for the thin-walled features. Since FDM uses a continuous, fixed-width extrusion for each curve, the gradual change in width will yield a void.

3.3.1. CAD revision (Figure 19).

To prevent these gaps, make the feature thickness constant in the XY plane. The thickness may vary at different Z elevations, but do so with discrete, stepped changes rather than having gradual changes (e.g., from a draft angle).

3.3.2. Optional revision.

Create each feature such that its wall thickness is an even multiple of the contour width (see Section 5.4.2 for details) that will be used to make the part. If modeled in this fashion, the feature will not require a custom group to have full density.
3.4. Self-supporting angles. (***) (Ti)

Supports are added to the bottom of a part and to all features with an angle, as measured from the XY plane, that is less than the Self-supporting angle in Insight (Figure 20). Since supports increase build times and material expense, it is advantageous to minimize them. One option to do so is to modify the CAD model.

3.4.1. Identify features (Figures 21 and 22).

The default Self-supporting angle ranges from 40° to 45°. Review the model to identify any features that are near this angle and can be modified.

3.4.2. Adjust features.

Alter the design of the identified features by increasing the angle such that it is > 45° from the XY plane. For features less than 0.25 in. (6.3 mm) wide in the XY plane, a cylindrical profile may be used (Figure 23).

Alternatively, the Self-supporting angle may be adjusted in Insight (Support > Setup > Advance settings). In general, a 5° to 10° adjustment is safe.

3.5. Variable density. (***) (St, Ti)

A unique characteristic of FDM is that a single part can have regions with different build styles. The advantages include:

- Varying density for optimal strength and weight.
- Varying density for optimal time and cost.

To some degree, this can be achieved in Insight by using custom groups (Figures 24 and 25). But for advanced control, make changes to the CAD model such that regions of the part can be processed with different toolpath controls (Figures 26 and 27).

3.5.1. Modify CAD model.

3.5.1.1. Create CAD model of part.

Begin with a complete CAD model of the part. Next, place the model in the same orientation in which it will be built.

To assist with alignment in Insight, add a reference feature. Locate this feature at 0, 0, 0 and make it slightly taller than the part (Figure 28).

Save the model.
3.5.1.2. Extract first region.

Extract the first variable density region from the CAD model and delete the balance (Figure 29). Note that the reference feature must be retained.

Export this file as an STL.

3.5.1.3. Extract second region (Figure 30).

Open the original CAD file and extract the second region. As with the first, retain the reference feature while deleting the balance of the model.

Since Insight will combine curves that appear to overlap, adjust this region by offsetting all surfaces that are coincident with the first region. A surface offset of 0.001 in. (0.03 mm) will suffice.

Export the file as an STL. Note that a high-resolution file is recommended for this and the previous STL. Large facets can create regions of overlapping curves between parts; these overlapping curves can create problems during toolpath generation. To avoid this problem use small facets and visually inspect, in Insight, all layers after toolpath generation.

Repeat this step for all other regions of the tool.

3.5.2. Process regions in Insight.

3.5.2.1. Process first region.

Open the STL file for first region (Figure 31). Select the model material and slice height and then orient and slice the file. Be sure to confirm that the reference feature is located at the origin to ensure that all regions will be aligned.

Next, apply toolpath settings to achieve the desired characteristics for the region. For example, change the Part interior style to Sparse-double dense, but note that this must be done through custom groups.

Now, save the job (File > Save as).

3.5.2.2. Repeat.

For all but the last region, repeat step 3.5.2.1. Note that all must use the same Model material, Slice height, and orientation.

3.5.2.3. Process last region.

Open, orient and slice the last STL and apply the desired toolpaths by using custom groups.

Save the job (File > Save as) and keep the file open.

Next, add the previous regions through the Slice > Combine slice curve files function. The reference feature will force all regions to align to each other (Figure 32).
After all regions have been added, select and delete (Edit > Delete > Curves) the curves for all of the reference features.

Save the job and continue with the usual file preparation procedures.

4. SELECTIONS: MATERIAL AND SLICE

When manufacturing end-use parts with FDM, spend a few minutes to consider the options for materials and slice heights. These are the first selections made when processing parts in Insight, and they are fundamental to product’s performance as well as the efficiency of the operation.

Start with the material decision (Figure 34); it is the more important of the two, and slice height options are material dependent. Then move on to slice height selection.

4.1. Material selection. (*)(Ae, St, Ti)

Criteria:

- Material properties (Figure 35).
  Consider the operational demands for the part in terms of mechanical, thermal, chemical and electrical properties.

- Support material.
  FDM offers two support styles: breakaway and soluble. Material selection dictates which styles are available. So, consider if the advantage of soluble supports—automated, labor-free and few concerns over access—are needed. If so, pick a material that uses them (Table 1).

- Slice height (Figure 36).
  Materials also determine which slice heights are available (Table 1). Slice height affects surface smoothness and specifies available raster and contour widths (see Section 4.2).
4.2. Slice selection. (*) (Ae, St, Ti)

Table 1 lists the available slice heights for each of the FDM materials. Slice height specifies the thickness of each layer, which affects surface resolution and stairstepping. It also determines which extrusion tip will be used, which in turn dictates the range of raster and contour widths (Table 2).

So, this one parameter influences build time, surface quality, feature resolution and even part strength. Therefore, this decision must take into account each of these factors.

<table>
<thead>
<tr>
<th>FDM Material</th>
<th>Support Style</th>
<th>Slice Heights inch (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABS+</td>
<td>Breakaway</td>
<td>0.005 (0.13)</td>
</tr>
<tr>
<td></td>
<td>Soluble</td>
<td>0.007 (0.18)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.010 (0.25)</td>
</tr>
<tr>
<td>ABSi</td>
<td>Soluble</td>
<td>0.005 (0.13)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.007 (0.18)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.010 (0.25)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.013 (0.33)</td>
</tr>
<tr>
<td>ABS-M30</td>
<td>Soluble</td>
<td>0.005 (0.13)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.007 (0.18)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.010 (0.25)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.013 (0.33)</td>
</tr>
<tr>
<td>ABS-M30i</td>
<td>Soluble</td>
<td>0.005 (0.13)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.007 (0.18)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.010 (0.25)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.013 (0.33)</td>
</tr>
<tr>
<td>PC-ABS</td>
<td>Soluble</td>
<td>0.005 (0.13)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.007 (0.18)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.010 (0.25)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.013 (0.33)</td>
</tr>
<tr>
<td>PC-ISO</td>
<td>Breakaway</td>
<td>0.007 (0.18)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.010 (0.25)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.013 (0.33)</td>
</tr>
<tr>
<td>PPSF</td>
<td>Breakaway</td>
<td>0.010 (0.25)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.013 (0.33)</td>
</tr>
<tr>
<td>ULTEM 9085</td>
<td>Breakaway</td>
<td>0.010 (0.25)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.013 (0.33)</td>
</tr>
</tbody>
</table>

Table 1: FDM material options.

* 400mc only
** 900mc only

<table>
<thead>
<tr>
<th>Slice Height</th>
<th>Tip</th>
<th>Minimum Toothpath</th>
<th>Maximum Toolpath</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.005 (0.13)</td>
<td>T10</td>
<td>0.008 (0.20)</td>
<td>0.022 (0.58)</td>
</tr>
<tr>
<td>0.007 (0.18)</td>
<td>T12</td>
<td>0.012 (0.30)</td>
<td>0.028 (0.73)</td>
</tr>
<tr>
<td>PC &amp; PC-ISO</td>
<td></td>
<td>0.010 (0.25)</td>
<td>0.026 (0.68)</td>
</tr>
<tr>
<td>0.010 (0.25)</td>
<td>T16</td>
<td>0.016 (0.41)</td>
<td>0.032 (0.83)</td>
</tr>
<tr>
<td>PC-ABS &amp; PC-ISO</td>
<td></td>
<td>0.034 (0.88)</td>
<td></td>
</tr>
<tr>
<td>PPSF &amp; ULTEM</td>
<td></td>
<td>0.030 (0.78)</td>
<td></td>
</tr>
<tr>
<td>0.013 (0.33)</td>
<td>T20</td>
<td>0.018 (0.46)</td>
<td>0.038 (0.98)</td>
</tr>
</tbody>
</table>

Table 2: Slice height variables. All units in inch (millimeter)
Thinner slices:
- Thinner layers for smoother surfaces.
- More layers, which increases build time.
- Thinner toolpaths (rasters and contours) for:
  - Finer feature resolution.
  - Better (denser) fills for thin-walls.

Thicker slices:
- Thicker layers produce faster builds.
- Fewer layers, which increases stairstepping.
- Thicker toolpaths (rasters and contours) for:
  - Faster deposition and shorter builds.
  - Stronger parts.

5. FILE PROCESSING (INSIGHT)

Although Insight’s “green flag” defaults will produce acceptable, and usually very good, results, there are many user-level controls to consider. These options apply to both model and support. They are used to improve part quality and process throughput.

The following file processing actions are commonly used to improve end-use parts while maximizing process speed.

5.1. Part orientation. (*Ae, St, Ti)

Part orientation is one of the few steps that must be competed for every part. Whether using the green flag defaults or customizing toolpaths, the first required action is part orientation. Yet, many give the impact and effects of orientation little thought. Although orienting to minimize the Z-height, and possibly build time, may be a viable strategy, there are other, equally important considerations (Figures 37 - 39).

5.1.1. Part orientation considerations.
- Strength.
  - Orient the part such that forces and loads are perpendicular to the slices.
- Surface finish.
  - Orient the part such that contours are in the XY plane.
- Build time.
  - Orient the part to:
    - Minimize build height.
    - Reduce the volume of support material.
    - Minimize the number of layers that contain both model and support.
5.1.2. Orienting parts.

There are two recommended options in Insight to orient a part (Figure 40).

- **STL > Orient by selected facet.**
- **STL > Rotate.**

5.2. Build styles. (*) (St, Ti)

Insight offers three **Part interior styles: Solid-normal, Sparse** and **Sparse-double dense**. Select the appropriate style for the following characteristics:

- Strength.
- Weight.
- Material consumption.
- Build time.

Part interior style may be selected from **Modeler > Setup** or **Toolpaths > Setup**.

Following are descriptions and characteristics of the fill styles.

5.2.1. Solid-normal (Figure 41).

- Dense fill with no gap between adjacent rasters.
  - Rasters run perpendicular to those on the proceeding layer.

- Characteristics:
  - Strongest.
  - Heaviest.
  - Highest model material consumption.
  - Longest build times.

5.2.2. Sparse (Figure 42).

- Hollow interior with internal lattice for structural rigidity.
  - Large air gap between rasters.
  - Uni-directional rasters on each layer (Figure 43).
  - Raster angle alternates between layers.

- Characteristics:
  - Weakest.
  - Lightest.
  - Lowest model material consumption.
  - Shortest build times.
5.2.3. Sparse – double dense (Figure 44).

- Hollow interior with internal lattice for structural rigidity.
  - Same as sparse, except bi-directional rasters on each layer (Figure 45).

- Characteristics (versus sparse):
  - Stronger.
  - Slightly heavier.
  - Slightly more model material.
  - Slightly longer build time.

5.2.4. Optimizing sparse fills.

For both Sparse and Sparse-double dense, several toolpath parameters can be changed to optimize weight, strength and build time. Make adjustments through Toolpaths > Setup > Advanced parameters or Toolpaths > Custom Groups > New/Modify.

The most frequently changed settings are (Figure 46):

- **Part interior depth**: Thickness of the perimeter contours.
  - Default is the thickness of one contour. Increase to add strength to surface of part.

  Note: the total perimeter thickness will be the **Part interior depth** value plus two default contours—in addition to the depth value there will be two contours on the outside of the part. In the case of the default value (**Part interior depth** = 1) the total perimeter thickness would be three contours (1+2).

- **Part sparse solid layers**: Number of slices above and below the sparse fill that will have a Solid-normal fill style.
  - Default is four layers. Increase to add strength to the top and bottom surfaces.

- **Part sparse fill air gap**: Distance between rasters.
  - Default, which ranges from 0.060 in. to 0.100 in. (1.52 mm to 2.54 mm), is dependent on material and tip size.
  - Increase to reduce build time and material consumption.
    - Can safely be increased to 250% of default value. Larger values are possible but adjustments may be needed for other parameters.
5.3. Visible surface style (*) (Ae, Ti)

Set **Visible surface style** to **Enhanced mode (Modeler > Setup)**.

**Enhanced mode** use small rasters for the toolpaths of external, visible surfaces. The visible surfaces also have a small negative air gap to press the rasters together (Figure 47). Internally, it uses thicker rasters with no air gap (Figure 48). The default values are listed in Table 3.

This style improves surface finish while decreasing build time.

**Enhanced mode** allows separate adjustment of raster widths and air gaps for the internal and external toolpaths. Accessed through **Toolpaths > Setup > Advanced settings** (Figure 49), they include:

- **Visible surface rasters**.
- **Visible surface rasters air gap**.
- **Internal rasters**.
- **Internal rasters air gap**.

<table>
<thead>
<tr>
<th>Material</th>
<th>Tip Size</th>
<th>Visible Raster</th>
<th>Visible Raster</th>
<th>Internal Raster</th>
<th>Internal Raster</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>inch (mm)</td>
<td>Air Gap</td>
<td>Air Gap</td>
<td>inch (mm)</td>
<td>Air Gap (mm)</td>
</tr>
<tr>
<td>ABS-M30, ABS-M30i, ABSi, PC-ABS</td>
<td>T-10 0.009 (0.23)</td>
<td>-0.001 (-0.03)</td>
<td>0.016 (0.41)</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>T-12 0.012 (0.30)</td>
<td>-0.001 (-0.03)</td>
<td>0.020 (0.51)</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>T-16 0.017 (0.43)</td>
<td>-0.001 (-0.03)</td>
<td>0.024 (0.61)</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>T-20 0.022 (0.56)</td>
<td>-0.001 (-0.03)</td>
<td>0.032 (0.81)</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>PC, PC-ISO</td>
<td>T-12 0.012 (0.30)</td>
<td>0.000</td>
<td>0.020 (0.51)</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>T-16 0.017 (0.43)</td>
<td>0.000</td>
<td>0.024 (0.61)</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>T-20 0.022 (0.56)</td>
<td>0.000</td>
<td>0.032 (0.81)</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>PPSF</td>
<td>T-16 0.017 (0.43)</td>
<td>-0.001 (-0.03)</td>
<td>0.024 (0.61)</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>T-20 0.022 (0.56)</td>
<td>-0.001 (-0.03)</td>
<td>0.032 (0.81)</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td>ULTEM 9085</td>
<td>T-16 0.017 (0.43)</td>
<td>-0.001 (-0.03)</td>
<td>0.024 (0.61)</td>
<td>0.000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>T-20 0.022 (0.56)</td>
<td>-0.001 (-0.03)</td>
<td>0.032 (0.81)</td>
<td>0.000</td>
<td></td>
</tr>
</tbody>
</table>

Table 3: Enhanced mode default settings.

5.4. Custom group: fill density/porosity. (*) (St)

Using the defaults for **Contour style** and **Contour width** across all features on an FDM part may result in porosity and air gaps. These voids will decrease the strength of features such as thin walls and bosses. For example:

- **Thin wall**: A single contour pass may leave an air gap if the distance between the contour’s sides is not large enough for a raster fill (Figure 50).
- **Boss**: At the contact points of the rasters with the contour, the raster turnaround will leave a small air gap. If tapping the boss, for example, the porosity would be exposed (Figure 51).

Eliminating the voids increases strength. To do this, custom groups apply user-defined values to the style and width of the outer shell, the contours, of individual features. The first approach uses the Contours to depth style.
Procedure:

5.4.1. Create custom group.

*Toolpaths > Custom groups > New.* Give the custom group a *Group name* and click the checkmark to save it.

5.4.2. Calculate contour width.

To eliminate voids in a wall or boss, Insight needs a contour width that is an even multiple of the feature’s thickness since all curves must be a closed loop. For example, an 0.080 in. (2.0 mm) wall could potentially have a contour of 0.040 in. (1.0 mm), 0.020 in. (0.5 mm), 0.013 in. (0.3 mm) or 0.010 (0.25 mm). So, begin by measuring* its feature and dividing its value by even numbers.

The next step is to confirm that the desired width is available for the job’s tip size. For a T16 tip, the 0.020 in. (0.5 mm) value is the only option. The available widths are visible in the dropdown menus of the toolpaths and custom group settings functions.

The selected width value becomes the *Contour width* setting in the next step.

*Measuring features:

- Display measurements *(View > Measure data).*
- Right click and select either Snap-measure or measure.
- Left click and drag over feature.

5.4.3. Define toolpath settings (Figure 52).

With the custom group displayed in the “Group Name” dialogue box, select *Modify from the custom groups pane.* Change *Contour style* to *Contours to depth,* which provides a user-definable perimeter thickness made from one or more contours.

Next, set the value for *Contour width,* which was 0.020 in. (0.5 mm) in the above example. Then set the *Depth of contour* equal to one-half of the feature’s thickness. In the example, this is 0.040 in. (1.0 mm).

The result is a feature with both an inside- and outside-surface thickness of 0.040 in. (1.0 mm).

5.4.4. Add curves to custom group.

Select all curves that will use the new custom group and click *Add.*

5.4.5. Generate toolpaths (Figure 53).

The last step is to generate toolpaths, shade them (Toolpaths > Shade toolpaths or right click > *Shade toolpaths*) and view them. The result should be solid fills with no gaps between rasters. If gaps exist, adjust the custom group values accordingly.
This type of custom group can also create features with an internal raster bounded by the user-defined **Contours to depth**. The next approach to eliminate air gaps is very similar, but it uses the **Perimeter only** style, which can have only one contour.

Procedure:

5.4.6. Create custom group.

Follow the instructions for 5.4.1 and 5.4.2. For the latter step, divide the measurement by 2 only.

5.4.7. Define toolpath settings (Figure 54).

From the custom groups window, select **Modify**. Change **Contour style** to **Perimeter only**, which provides a user-definable perimeter thickness made from one contour. Next, set the **Contour width** value to that calculated in 5.4.6.

Note that this method would not work with a 0.080 in. (2.0 mm) feature using a T16 tip. The necessary value (0.040 in./1.0 mm) is not an option for this tip size.

5.4.8. Add curves to custom group.

Select all curves that will use the new custom group and click **Add**.

5.4.9. Generate toolpaths (Figure 55).

The last step is to generate toolpaths, shade them (**Toolpaths > Shade toolpaths** or right click > **Shade toolpaths**) and view them. The result should be solid fills with no gaps between rasters. If gaps exist, adjust the custom group values.

5.5. Custom group: gradual slopes. (**) (Ae)

On surfaces with a gradual slope, default toolpaths may produce visible pin holes at the point of raster turnarounds. This happens when the offset of a layer is larger than the contours of the layer below (Figure 56). To conceal the raster turnarounds and have a solid external surface, create and apply a custom group to the affected slices (Figure 57).

Procedure:

5.5.1. Create custom group.

**Toolpaths > Custom groups > New.** Give the custom group a **Group name** and click the checkmark to save it.

Note: Do not give the group a name that already exists; all other curves with that name will be overwritten.

5.5.2. Modify toolpath settings.

To conceal the raster turnarounds, increase the contour depth such that it overlaps with the contour for the layer above it.
Select **Modify** from the custom group window. Change **Contour style** to **Contours to depth**. Next, increase the **Depth of contours** value; make it large enough inset the rasters below the contours for the next layer.

5.5.3. Add curves to custom group (Figure 58).

Select all curves on the gradual slope surfaces and click **Add**.

5.5.4. Generate toolpaths.

After generating toolpaths, shade them (**Toolpaths > Shade Toolpaths**) and view all affected layers. Use the **View layer plus layer above/below** setting to see if all raster turnarounds are concealed by the layer contours (Figures 59 and 60). If adjustments are necessary, return to the **Modify** function and adjust the **Depth of contours** value.

5.6. Custom group: supports made with model material. (**) (Ti)

Although it takes only a few seconds per layer, the time to switch between model and support material can have a noticeable impact on throughput. The increase in build time becomes more significant as the number of layers increase.

To eliminate this time from the build, create a custom group that constructs supports from model material.

Apply this technique when there are a large number of layers with both model and support curves (Sparse style) and when the supports are accessible and easily removed. In addition, this technique is only applicable for layers that contain just part curves (red) and **SupportSparse** curves (gray). To determine if these conditions exist:

- **Generate supports.**
  - After slicing the file, set **Support style** to **SMART** or **Sparse** (**Support > Setup**). Then click the **Create supports** icon.
- **View slices** (Figure 62).
  - Scroll through all the slices. Look for:
    - Layers with only model (red) and sparse/SMART supports (gray).
    - Supports curves that that can easily be removed.
      Avoid supports that surround model curves and supports that are surrounded by model curves. With either condition, the supports will be difficult to remove without damaging the part.

To swap model for support material, use the following procedure:

5.6.1. Create custom group.

**Toolpaths > Custom groups > New.** Click the **Templates** icon and select **SupportSparse** from the Template group menu (Figure 63). Click the checkmark.
By using the template, the new custom group inherits all of the toolpath settings from *SupportSparse*.

5.6.2. Modify toolpath settings.

The only setting that will change is *Toolpath material*. To make supports from model material, simply select *Model* from the dropdown menu (Figure 64).

Give the custom group a *Group* name and click the checkmark to save it.

5.6.3. Add default support curves to custom group.

Select only the support curves (gray) that are accessible and removable. DO NOT add basic supports (gold) or support interface (green) curves.

After selecting the curves, click *Add* from the custom groups window (Figures 65 and 66).

5.7. Custom group: wide supports (**) (Ti)

The default settings for supports create a strong, rigid structure made from soluble or breakaway support material. To reduce the volume of supports, and therefore the time to construct them, increase the air gap between each support raster (Figures 67 and 68).

The wide support procedure is defined in a custom group that increases the air gap and replaces support material with model material (Figures 69 and 70).

Apply this technique when there are large, tall groupings of *SupportSparse* towers—or more generally, a lot of *SupportSparse* surface area—that are accessible and do not surround any model features (Figure 71).

Procedure:

5.7.1. Create custom group 1.

*Toolpaths > Custom groups > New*. Click the *Templates* icon and select *SupportSparse* from the *Template* group menu and click the checkmark.

By using the template, the new custom group inherits all of the toolpath settings from *SupportSparse*.

5.7.2. Modify toolpath settings (Figure 72).

Change *Toolpath material* to *Model* so that the supports will have the strength to support the part. Next, set the *Start angle* to -30° (90° counterclockwise from the default angle).

The last setting adjusts the *Adjacent rasters* air gap by increasing the spacing between the support rasters. This value can be up to 0.75 in. (19.1 mm) with no other alterations.

Give the custom group a *Group name* and click the checkmark to save it.
5.7.3. Create custom group 2.

**Toolpaths > Custom groups > New.** Click the **Templates** icon and select **SupportSparse** from the **Template** group menu and click the checkmark.

5.7.4. Modify toolpath settings (Figure 73).

Use the same settings as in Step 5.7.2 for **Toolpath material (Model)** and **Adjacent rasters** air gap.

Set the **Delta angle** to 90° and the **Layers between delta** to 99. These settings place a layer of support that runs perpendicular to the 99 layers below it. This acts as a cross-member that keeps the support column rigid and stable.

Give the custom group a **Group name** and click the checkmark to save it.

5.7.5. Add curves to custom groups.

After selecting and applying the custom groups, the support columns should build with the following order of toolpaths (Figure 74):

- **Bottom 10 layers:** Custom group 1 (-30° raster angle).
  - The wide supports cut across the defaults supports (if parallel to them, they would “fall through the cracks”). (Figure 75)
- **Middle layers:** Custom group 2.
  - First 99 layers (counted from the first part layer): -30° raster angle.
  - Next layer: 60° raster angle.
  
  This set of alternating raster angles repeats every one hundred layers and stops when only 10 layers remain.
- **Next 10 layers:** Custom group 1 (-30° raster angle)
  - To form a bridge for the final layers.
- **Top 10 layers:** Default supports.
  
  This is the transition zone where the support style returns to that used for all other supports in the file and where the change to support interface occurs.
To establish this pattern, locate the layers that contain the support curves that will use wide air gaps. Next, set the **Range top** to 10 layers below the top of the supports and **Range bottom** to the bottommost layer of supports. Now, display this range by clicking on **View range of layers**.

Select the bottom 10 and top 10 layers for the range and add them to custom group 1. Then select all remaining layers in the range and add them to custom group 2. For parts with many features, selection may be easier if some model curves are hidden (**View > Hide curves**). To further simplify selection, use the **Selection tools** (Figure 76) that appear in the custom group pane (e.g. by group name, color or curve type).

### 6. THROUGHPUT & PRODUCTION PLANNING

When compare to conventional manufacturing processes, FDM is unique in several ways:

- It can operate as a self-service process.
- It can build a variety of parts in a single operation.
- It is an unattended operation.

This uniqueness offers advantages but only when they are leveraged. To do so, a change in thinking and operational style may be needed. When scheduling an FDM system, consider the following:

- No setup (changeover) time required (Figure 77).
  - Small batch runs and one-off builds are economical and efficient. So, parts may be built one at a time and as needed.
- Production mix is unlimited (Figure 78).
  - Build any combination of parts, prototypes and manufacturing tools.
  - Combine work for multiple staff members, departments or divisions.
- Multi-piece runs increase throughput.
  - By sharing the operational overhead, such as the time to purge a tip, the time per part decreases.
- Tallest part carries most of the process overhead.
  - Much of the operational overhead is dictated by the height of the tallest part. So, a second copy of it, or adding shorter parts, will add far less time than that to build them separately.
- Builds can be matched to shift schedules (Figure 79).
To maximize throughput, consider the shift schedule when laying out builds, launching runs and planning the production schedule. A five-hour build started at 4:00 pm may mean an idle machine for 11 hours.

Create jobs that fit with the start and stop times of working shifts; jobs that will consume the better part of a weekend; and jobs that will run over a long, holiday weekend. It is inefficient to end a build when there is no staff available for change-over.

The production planning and scheduling tools are:

- Control Center
- Insight

6.1. Packing the build sheet. (*) (Ti)

This operation is basic and routine. Yet, it is important enough to reiterate.

In Control Center, add individual jobs (saved from Insight) to maximize throughput while balancing the build time with the working schedule of those that will remove the parts and start the next build.

Procedure:

6.1.1. Create a pack.

While in the Control Center Pack tab, click Insert CMB and browse for the desired file. Click Open (Figure 80). Repeat until all desired parts are in the pack.

6.1.2. Optimize the pack.

To add duplicates of any part in the pack, click Copy. Next, arrange all parts by clicking and dragging, rotating or Repacking.

6.1.3. Monitor the estimated time in the Pack Details window.

Adjust the time to match the production schedule by adding, copying or removing parts from the pack (Figure 81).

6.2. Nesting parts. (***) (Ti)

Control Center prevents parts from impinging on other parts’ bounding volumes. It also optimizes the travel of the extrusion head for individual parts rather than planning the path for the collection of all parts.
To optimize build packing and extrusion head travel, nest part files in Insight (Figure 84) prior to sending them to Control Center. Using this technique, parts can be stacked, overlapped and nested. One example is the placement of a part in a center void of another, as would be seen with a torus (donut shape).

Procedure:

6.2.1. Open and slice part 1 (Figure 85).

Save the job file (File > Save as > Job).

6.2.2. Open and slice part 2 (Figure 86).

6.2.3. Add part 1 to part 2’s job file (Figure 87).

- **Slice > Combines slice curve files.**
- Click **Slice file** field and select part 1.sjb file.
- Place part 1.
  - Click and drag to a new location or enter coordinates in **Slice curve location fields**.
- Click **OK** to add the part.
  - Note, if placement needs adjustment after the part has been added, use the **Edit > Move, rotate, copy…** function.

6.2.4. Create toolpaths and supports.

6.2.5. Save toolpaths.

- **File > Save as > Toolpath.**

This creates a .cmb.gz file, which is commonly called the “CMB.” To build this set of nested tools later, while in Control Center, select Insert CMB. Then process the build as usual.

7. RECAP - CRITICAL SUCCESS FACTORS

7.1. Leverage advantages of additive manufacturing.

- Complex structures, including part consolidation.
- Design liberties.
- Redesign and revision frequency.
- Automated, lights-out operation.
- Small-lot, on-demand production.
- Lead time and cost reduction.
7.2. Invest time accordingly:

7.2.1. Pilot production.
   Light adjustments.

7.2.2. Bridge-to-production.
   Moderate adjustments and some optimization.

7.2.3. Full production.
   Complete redesign and full optimization.

7.3. Optimize jobs for:

• Strength.
• Aesthetics.
• Maximum throughput/minimal time.
• Lowest piece-part cost.

To obtain a file for the sample part, or to obtain more information on this application, contact:

Stratasys Application Engineering
1-855-693-0073 (toll free)
+1 952-294-3888 (local or international)
ApplicationSupport@Stratasys.com