



TECHNICAL APPLICATION GUIDE

FDM FOR END-USE PARTS

End-use parts include all components, sub-assemblies and products created during both series production and/or custom manufacturing.

Additive manufacturing is a viable alternative to molding, machining, forming and casting of end-use parts. These traditional processes 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.

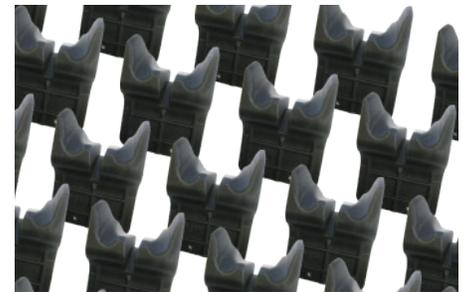
Fused Deposition Modeling™ (FDM®) is an additive manufacturing technology that eliminates tools and dies in order to streamline the production process. Substituting FDM for traditional manufacturing processes substantially reduces lead times and costs. It also makes production a simple, unattended and virtually labor-free operation.

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

Legend:

- * All users
- ** Intermediate users
- *** Advanced users

- De Design optimization
- Ae Aesthetic optimization
- St Strength optimization
- Ti Time optimization



Production lot of Nova Tech Engineering's end-use parts.

APPLICATION COMPATIBILITY

TECHNOLOGY	IDEA	DESIGN	PRODUCTION
FDM	1	2	4
PolyJet	0	1	0

(0 – N/A, 1 – Low, 5 – High)

COMPANION AND REFERENCE MATERIALS

Technical application guide	<ul style="list-style-type: none"> • Document
Application brief	<ul style="list-style-type: none"> • Document
Video	<ul style="list-style-type: none"> • Commercial • Success story • How it's used
Referenced processes	<ul style="list-style-type: none"> • Sparse fill • Metal inserts • Section large parts • Finishing techniques • Optimize seam location • Orienting for smooth part • Part dipping

TABLE OF CONTENTS

FDM FOR END-USE PARTS

1. OVERVIEW	3
1.1. Application	3
1.2. FDM is a best fit	3
1.3. Successful adopter traits	2
1.4. FDM adoption obstacles	4
1.5. Benefits	4
2. ROLES & STRATEGIES	5
2.1. Pilot production	5
2.2. Bridge-to-production	5
2.3. Full production	5
2.4. Bridge-to-end of life	6
3. DESIGN FOR FUNCTION (CAD)	6
3.1. Integrated design (consolidation) (*) (De)	6
3.2. Integrated features (*) (De)	6
3.3. Maximize fill density (**) (St)	7
3.4. Self-supporting angles (***) (Ti)	8
3.5. Variable density (***) (St, Ti)	8
4. SELECTIONS: MATERIAL AND SLICE	10
4.1. Material selection (*) (Ae, St, Ti)	10
4.2. Slice selection (*) (Ae, St, Ti)	12
5. FILE PROCESSING (INSIGHT™ SOFTWARE)	12
5.1. Part orientation (*) (Ae, St, Ti)	12
5.2. Build styles (*) (St, Ti)	13
5.3. Visible surface style (*) (Ae, Ti)	15
5.4. Custom group: fill density/porosity (*) (St)	16
5.5. Custom group: gradual slopes (**) (Ae)	17
5.6. Custom group: supports made with model material (**) (Ti)	18
5.7. Custom group: wide supports (**) (Ti)	19
6. THROUGHPUT & PRODUCTION PLANNING	21
6.1. Packing the build sheet (*) (Ti)	22
6.2. Nesting parts (***) (Ti)	23
7. KEY PROCESS CONSIDERATIONS	24
7.1. Obstacle details	24
7.2. Resolution details	24
8. RECAP - CRITICAL SUCCESS FACTORS	25
8.1. Leverage advantages of additive manufacturing	25
8.2. Invest time accordingly	25
8.3. Optimize jobs	25
8.4. Eliminate adoption obstacles	25

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:

- Time and cost for tooling (or machining) are obstacles.
 - Low, up-front investment and fast turn-time needed to enhance profitability or meet product launch schedules
- On-demand, just-in-time (JIT) manufacturing is used.
 - For small-batch manufacturing
 - For smaller inventories
 - For flexible production schedules
- Design modifications are expected.
 - Custom and configured-to-order products.
 - Products that will be revised throughout the life cycle.
- Optimized designs are needed.
 - Design for function and performance rather than process.
 - Little or no penalty for complex, feature-rich parts.
- Low-volume production is utilized.
 - Generally, practical production quantities are between 1 - 1,000+
 - Quantity is dependent on part size; smaller parts can have larger production quantities.
- Optimal part size
 - Smaller than 300 mm (12 in) for XYZ.
 - Larger parts are possible.
- Materials are compatible.
 - Confirm suitability of thermoplastic materials to ensure:
 - Mechanical and electrical properties
 - Chemical resistance and thermal resistance
- FDM accuracy is suitable.
 - Without secondary machining, > 0.13 mm (0.005 in).
 - With secondary machining, tighter tolerances are possible.

1.3. Successful adopter traits (first iteration and long-term):

- Customer is open to change and seeking innovation.
 - Part design
 - Manufacturing processes
 - Business models
- Start simple, then advance.
 - Tackle straightforward applications first.
 - With experience, move on to more demanding parts.
- Design for performance, time and cost.
 - Start with a fresh design that optimizes part qualities as well as FDM build time and material consumption.



Figure 1: Processing equipment component (PC-ABS).



Figure 2: Electronic surveillance equipment includes FDM housing (ABS-M30).



Figure 3: MRI head coil required compatible materials (PC-ISO).

FDM FOR END-USE PARTS

- Understanding when FDM is the best alternative.
 - Continue to use traditional methods in combination with FDM.

1.4. FDM Adoption Obstacles:

Obstacle	Solution*
Adhering to traditional design practices.	<ul style="list-style-type: none"> • Optimize part design for FDM as outlined in FDM Design Handbook. • Create application specific "best practices" when not defined in FDM Design Handbook.
Applying FDM to high-precision applications.	<ul style="list-style-type: none"> • Offset surfaces in CAD. • Orient critical dimensions in the XY plane. • Use secondary machining operations.
Considering FDM as a complete replacement for existing processes.	<ul style="list-style-type: none"> • Use FDM as an alternative when other processes cannot deliver. • Combine FDM and traditionally manufactured parts.
Focusing only on piece-part cost.	<ul style="list-style-type: none"> • Include cost reductions from all operations (e.g. design to warehouse and transportation). • Include, where applicable, financial gains from optimized designs when put into service.
FDM part requires extensive post-processing.	<ul style="list-style-type: none"> • Leverage automated processes such as mass finishing.
Making large, bulky parts without modification.	<ul style="list-style-type: none"> • Reduce time and material with sparse fills and large rasters. • Reduce time by making supports from model material. • Use evenings and weekends for long builds.
Using green flag (default) process parameters.	<ul style="list-style-type: none"> • Use Insight software to adjust strength, porosity, surface quality and weight.

* Additional solutions may exist.

1.5. Benefits:

- Lead time reduction
 - Average lead time savings: 75% to 90%
 - Shift from prototyping to production in one day.
- Cost reduction
 - Average cost savings: 50% to 90%
 - No up-front cost; eliminate expense of tool design and tool making.
- Digital inventory
 - Utilize just-in-time manufacturing.
- Design freedom
 - Improve product performance.
 - Maximize ergonomics.
 - Consolidate multi-piece assemblies.
- Design flexibility
 - Modify product designs at any time.
- Streamlined operations
 - Few steps and few resources needed for production runs.
- Profitable production of low-volume, high-complexity parts.



Figure 4: Bezel for pharmacy equipment (ABS-M30).

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 or stopgap solution, only minor adjustments may be warranted. Consequently, 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 pilot production, use the part design that suits the conventional production method. The “freedom of design” that FDM 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. The production quantity is also likely to warrant a larger up-front investment of time.

- 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 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.



Figure 5: Gauge pod for custom motorcycle (painted polycarbonate).

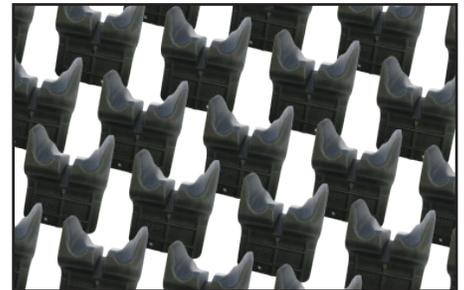


Figure 6: Pilot production: minor process adjustments are acceptable. (PC-ABS)

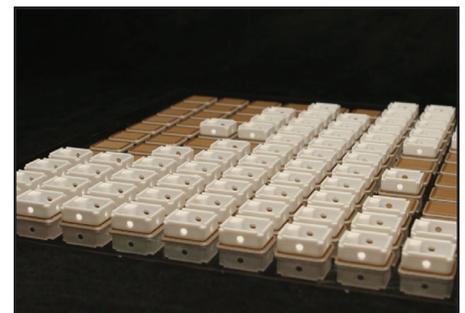


Figure 7: Full production: optimized orientation and pack for throughput (polycarbonate).

2.4. Bridge-to-end of life

As a product nears the end of its life cycle, further investments in repairing or replacing tooling may not be justifiable. As a result, production may be halted prematurely while spare parts are still needed.

FDM may be used to extend a product's life and to manufacture spare parts on an as-ordered basis, thereby eliminating the need for physical inventory. As with full production, optimize both the part design and the FDM process while maintaining fit, finish and performance standards.

- Complete redesign is justifiable.
- Fine-tune process for maximum optimization.

3. Design for Function (CAD)

A common way to apply FDM Technology™ to end-use parts manufacturing is to use existing designs and design rules. However, they may not capitalize on the advantages of FDM. In fact, they may negatively affect performance, time and cost.

Whenever possible, start with a fresh design that builds on the design freedoms offered by FDM 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, integrate as many components as possible into one piece (Figure 10). If designing a new item, create it as one piece. Only split off parts when it is advantageous to the part's operation or maintenance.

Integrated design has many advantages including:

- Improved functionality
 - Focus on the task that the part will perform. Optimize its design for function rather than the process used to make it.
- Eliminate tolerance challenges.
 - Holding tight tolerances is costly. If two mating parts are combined into one, concerns related to controlling the tolerances 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.
- Decrease bill of materials (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.



Figure 8: Part split into pieces to aid in assembly of final product (polycarbonate).



Figure 9: Final product next to FDM component. Material properties allow for MRI functionality (polycarbonate).



Figure 10: Redesigned FDM tool consolidates most components (ABS-M30).

3.2.1. Adding features

With traditional manufacturing processes, there is usually a cost associated with every feature. 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.
- Add ribs, bosses and gussets.
 - This may increase build time and material cost, however the impact is far less than that of a traditional process.
- Add embellishments (Figure 14).
 - Incorporate part numbers, storage locations, alignment aids, logo or operational instructions directly on the part.

3.2.2. Adding 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.

An 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 (i.e. less than 1.52 mm / 0.060 in), there may not be enough space for the internal rasters (Figure 18). This may result in an internal void. These voids can be eliminated in Insight software (see Section 5.3.) or prevented in CAD.

In the CAD model, avoid tapers or draft angles for 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 it is best to include discrete, stepped changes rather than 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) used to make the

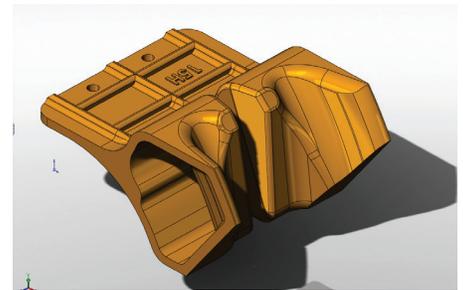


Figure 11: Original design – single component.

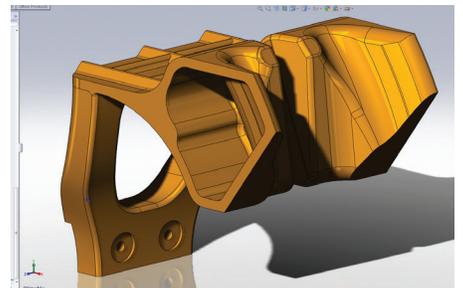


Figure 12: Consolidated design – integrated mounting flange.

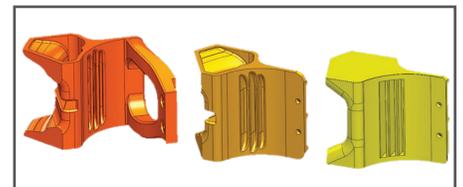


Figure 13: Three design options – adding features improves performance while decreasing time and expense.



Figure 14: Add embellishments to end-use parts such as part numbers (PC-ABS).



Figure 15: Fasteners may be mounted to FDM parts (ABS-M30).

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 are less than the self-supporting angles in the software (Figure 20). Since supports increase build times and material expense, it is advantageous to minimize them. One way to do this 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 6.3 mm (0.25 in) wide in the XY plane, a cylindrical profile may be used (Figure 23).

Alternatively, the self-supporting angle may be adjusted in the software (Support > Setup > Support Parameters). In general, a 3° 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 of this characteristic include varying density for:

- Optimal strength and weight.
- Optimal time and cost.

To some degree, this can be achieved in the software by using custom groups (Figures 24 and 25). For advanced control, make changes to the CAD model so that each region is processed with different toolpath options (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 the software, add a reference feature. Locate this

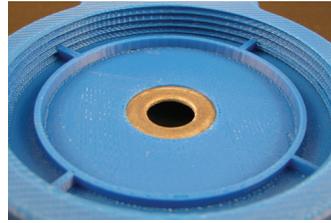


Figure 16: Integrated bushing is embedded while part is building (ABSplus).

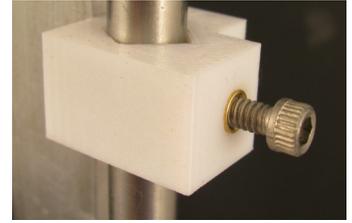


Figure 17: Threaded insert may be added during FDM builds or in secondary operations (ABS-M30).

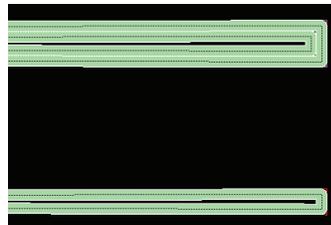


Figure 18: Thin-walled features with taper, may have air gaps between contours.



Figure 19: CAD revisions produce a constant wall thickness that has a solid, dense fill.

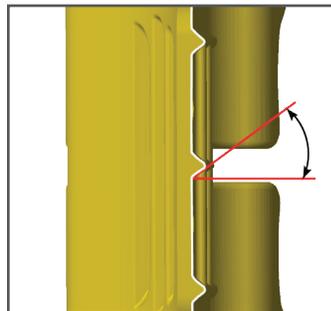


Figure 20: Insight software's self-supporting angle specifies when support structure is needed.



Figure 21: Review CAD model for features that may avoid supports.

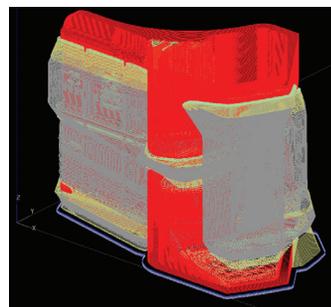


Figure 22: Supports on back of part where ribbing is below the self-supporting angle.

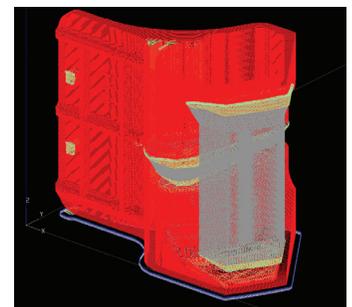


Figure 23: With minor alteration to rib design, supports are no longer needed.

feature at the origin (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: The reference feature must be retained.

Export this file as a high resolution STL.

3.5.1.3. Extract second region (Figure 30).

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

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

Export the file as a high resolution STL.

Note: Large facets can create regions of overlapping curves between parts. This can create problems during toolpath generation.

To avoid this problem, use small facets and visually inspect all layers after toolpath generation in the software.

Repeat this step for all other regions of the tool.

3.5.2. Process regions in Insight software.

3.5.2.1. Process first region.

Open the STL file for the first region (Figure 31). Select the model material and slice height. Then, orient and slice the file. 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

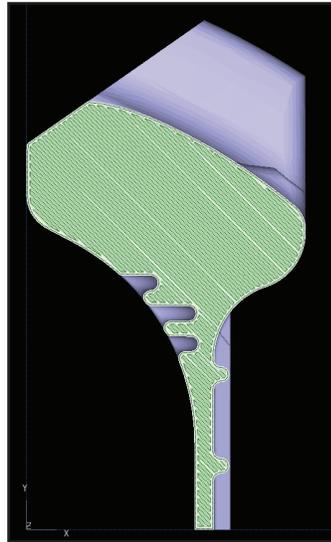


Figure 24: Solid-normal fill throughout part.

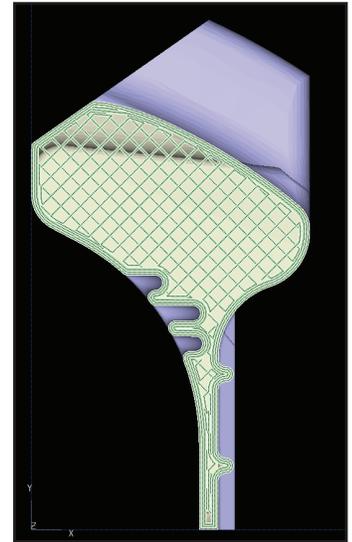


Figure 25: Sparse – double dense fill in narrow section when custom groups are used.

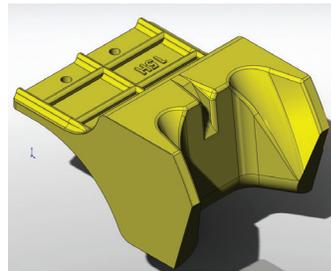


Figure 26: CAD model of part.

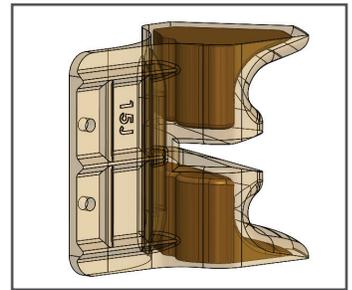


Figure 27: Transparency shows two regions of variable density.

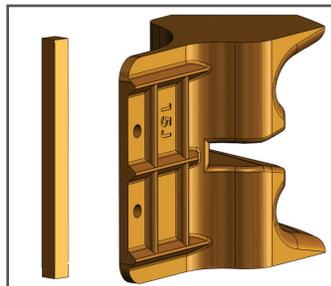


Figure 28: CAD model with reference feature at 0,0,0.

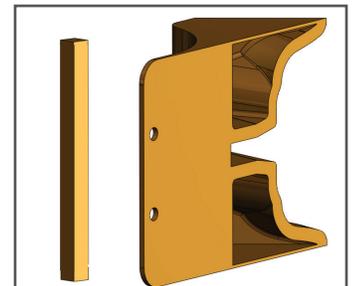


Figure 29: Cross-section showing region 1 with region 2 extracted.

style” to “Sparse-double dense”. This must be done through custom groups (Toolpaths > Custom group > Add).

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

3.5.2.2.Repeat.

For all but the last region, repeat step 3.5.2.1. 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 file and apply the desired toolpaths by using custom groups.

Save the job and keep the file open (File > Save as > Job). Next, add the previous regions (Slice > Combine slice curve files). The reference feature will force all regions to align to each other (Figure 32).

After all regions have been added, select and delete the curves for all of the reference features (Edit > Delete > Curves).

Save the job and continue with the usual file preparation procedures (File > Save as > Job).

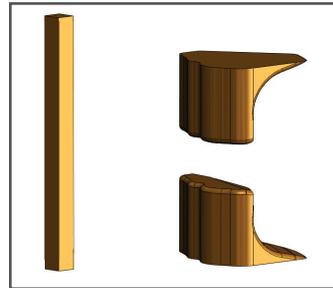


Figure 30: CAD model of region 2 with reference feature.

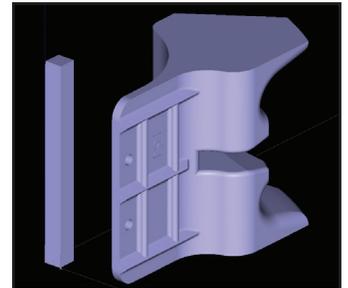


Figure 31: Open first region in Insight software.

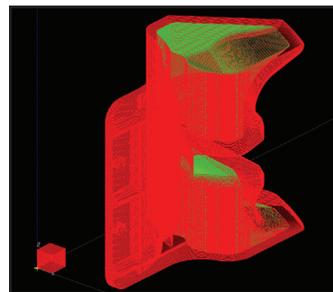


Figure 32: Combine files for both region 1 (red) and region 2 (green).

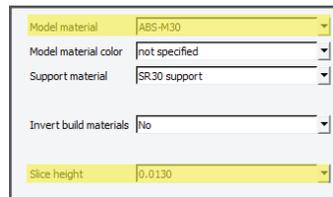


Figure 34: Select materials first and then select slice height.

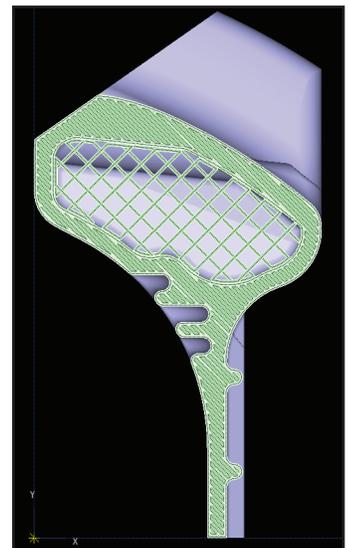


Figure 33: Result of variable density.

4. Selections: Material and Slice

When producing 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 the software; they are fundamental to the product’s performance, as well as the efficiency of the operation.

Start with the material decision (Figure 34). Then, move on to slice height selection.

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

The following are the top three selection criteria listed in order of importance:

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

Mechanical Properties	English	Metric
Tensile Strength	5,200 psi	36 MPa
Tensile Modulus	350,000 psi	2,413 MPa
Tensile Elongation	4%	4%
Flexural Strength	8,800 psi	61 MPa
Flexural Modulus	336,000 psi	2,317 MPa
IZOD Impact, notched	2.6 ft-lb/in	139 J/m
IZOD Impact, un-notched	5.3 ft-lb/in	283 J/m

Figure 35: Review FDM material data sheets to find the desired properties. (Build orientation is on long side.)

- Support material

FDM offers two support styles: breakaway and soluble. The material selection will dictate which styles are available so consider if the advantages of soluble supports (e.g. automated, labor-free and fewer concerns over access) are required. If so, choose a material that offers the appropriate type of support (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).

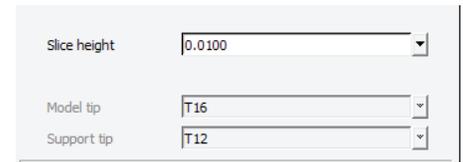


Figure 36: Slice height selection establishes which tips to use.

FDM Material	Support Style	Slice Heights mm (in)
ABSplus	Breakaway or Soluble	0.127 (0.005)** 0.176 (0.007) 0.254 (0.010)
ABSi	Soluble	0.127 (0.005)** 0.176 (0.007) 0.254 (0.010) 0.330 (0.013)
ABS-M30	Soluble	0.127 (0.005)** 0.176 (0.007) 0.254 (0.010) 0.330 (0.013)
ABS-M30i	Soluble	0.127 (0.005)** 0.176 (0.007) 0.254 (0.010) 0.330 (0.013)
ABS-ESD7	Soluble	0.176 (0.007) 0.254 (0.010)
PC-ABS	Soluble	0.127 (0.005)** 0.176 (0.007) 0.254 (0.010) 0.330 (0.013)
PC	Breakaway or Soluble	0.127 (0.005)* / ** 0.176 (0.007) 0.254 (0.010) 0.330 (0.013)
PC-ISO	Breakaway	0.176 (0.007) 0.254 (0.010) 0.330 (0.013)
PPSF	Breakaway	0.254 (0.010) 0.330 (0.013)**
ULTEM™ 9085 thermoplastic resin	Breakaway	0.254 (0.010) 0.330 (0.013)***

Table 1: FDM material options.

* Soluble only

** Fortus® Production System 400mc only

*** Fortus® Production System 900mc only

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. This will affect surface resolution and stairstepping. It also determines which extrusion tip to use, which in turn, dictates the range of raster and contour widths (Table 2).

Important: Slice selection influences build time, surface quality, feature resolution and part strength. Therefore, take into account each of these factors:

Thinner slices:

- Thinner layers = smoother surfaces
- More layers = increased build time
- Narrow toolpaths (rasters and contours) =
 - Finer feature resolution
 - Denser fills for thin walls

Thicker slices:

- Thicker layers = faster builds
- Fewer layers = increased stairstepping
- Wider toolpaths (rasters and contours):
 - Faster deposition and shorter builds
 - Stronger parts

Slice Height	Tip	Minimum Toolpath mm (in)	Maximum Toolpath mm (in)
0.127 (0.005)	T10	0.20 (0.008)	0.58 (0.022)
0.176 (0.007)	T12	0.30 (0.012)	0.73 (0.028)
*PC & PC-ISO	T12	0.25 (0.010)	0.68 (0.026)
0.254 (0.010)	T16	0.41 (0.016)	0.83 (0.032)
*PC-ABS & PC-ISO	T16	0.41 (0.016)	0.88 (0.034)
*PPSF & ULTEM	T16	0.41 (0.016)	0.78 (0.030)
0.330 (0.013)	T20	0.46 (0.018)	0.98 (0.038)

Table 2: Slice height variables.

* Material configured for individual toolpath settings.

5. File Processing (Insight)

Although Insight software's "green flag" defaults will produce acceptable, and usually very good results, there are many control options to consider. These options apply to both the model and the support, and are used to improve part quality and process throughput. The following file processing actions are commonly used to improve end-use part designs while maximizing process speed.

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

Part orientation is one of the few required actions for every part. Whether using the default toolpath parameters or customizing toolpaths manually, it is the first step. Although orienting to minimize the Z-height, and possibly build time may be a viable strategy, the following are equally important considerations (Figures 37 - 39).

5.1.1. Considerations:

- Strength
 - Orient the part so that forces and loads are perpendicular to the slices.
- Surface finish
 - Orient the part so that the part's critical 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 material and support material.
- Time and ease of support removal
 - Orient the part to minimize the amount of supports and to make support removal easy.
 - Small cavities are oriented so that supports have access for removal.
- Airflow
 - Ensure uniform airflow on parts, especially for higher temperature materials (ULTEM thermoplastic resin and PPSF).

5.1.2. Software settings

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

- (STL > Orient by selected facet)
- or
- (STL > Rotate)

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

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

- Strength
- Weight
- Material consumption
- Build time

Part interior fill style may be selected from:

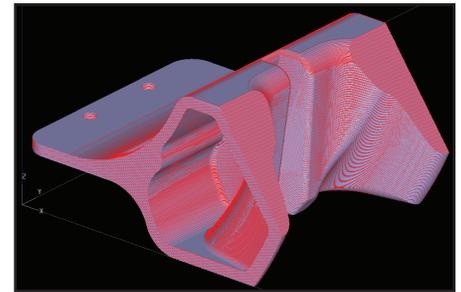


Figure 37: Default orientation when opened in Insight software.

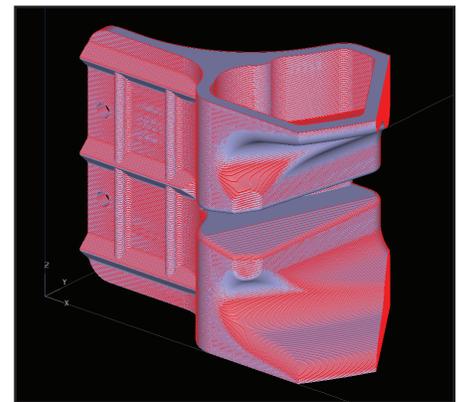


Figure 38: Orientation for strength and minimal supports.

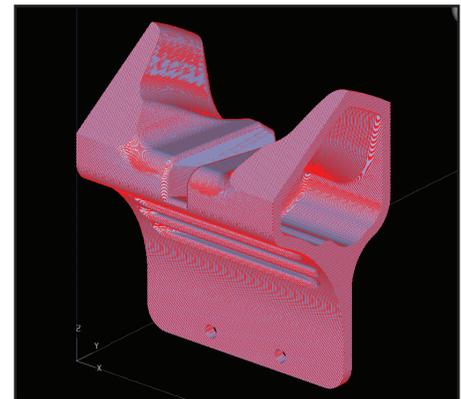


Figure 39: Alternative orientation.

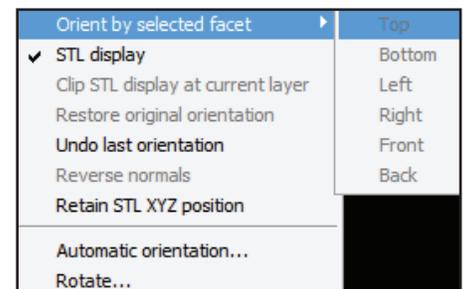


Figure 40: Orientation dropdown menu offering Rotate and Orient by selected facet.

- (Modeler > Setup)
or
- (Toolpaths > Setup)

The following are descriptions and characteristics of the interior fill styles.

5.2.1. Solid-normal (Figure 41)

- Dense fill with no gap between adjacent rasters.
 - Rasters run perpendicular to those on the preceding 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 gaps between rasters
 - Uni-directional rasters on each layer (Figure 43)
 - Raster angle alternates between layers
- Characteristics:
 - Weakest
 - Lightest
 - Lowest model material consumption
 - Shortest build time

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 compared to Sparse fill:
 - 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. Adjustments are made through either

- (Toolpaths > Setup > Toolpath Parameters)
or
- (Toolpaths > Custom Groups > New/Modify)

The most frequently changed settings are (Figure 46):

- Number of internal contours:
 - Default is the thickness of one contour. Increase The number of interval contours to add strength to the surface

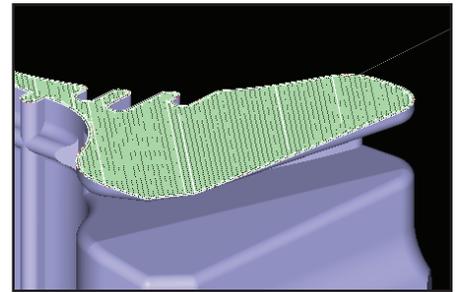


Figure 41: Part with Solid-normal fill.

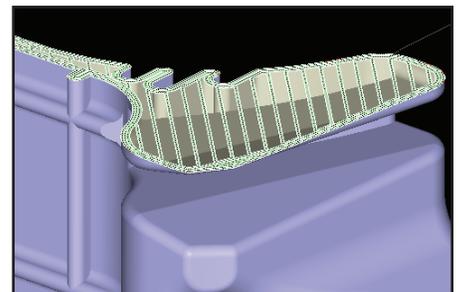


Figure 42: Part with Sparse fill.

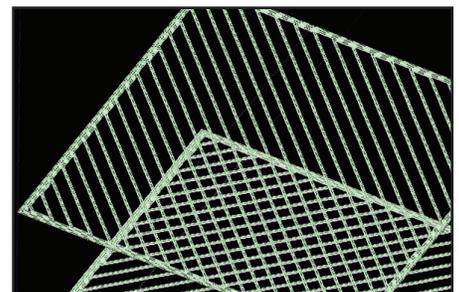


Figure 43: Sparse fill alternates raster direction for each layer.

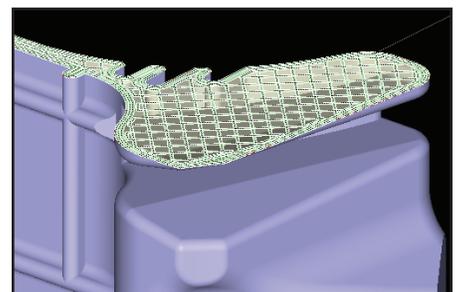


Figure 44: Part with Sparse – double dense fill.

- of the part.
- Note: The total perimeter thickness will be the “number of internal contours” value plus two default contours. In addition to the internal contours, there will be two contours on the outside of the part.
- Part sparse solid layers: number of slices above and below the sparse fill that will have a “Solid-normal” fill style.
 - Increase part sparse solid layers to add strength to the top and bottom surfaces.
- Part sparse fill air gap: the distance between rasters.
 - Default, which ranges from 1.52 mm to 2.54 mm (0.060 in to 0.100 in), is dependent on material and tip size.
 - Increase to reduce build time and material consumption.
 - Large values are possible but adjustments may be needed for other parameters.

5.3. Visible surface style (*) (Ae, Ti)

Set visible surface style to “Enhanced” (Modeler > Setup).

Enhanced mode uses 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 (Toolpaths > Setup > Advanced settings) (Figure 49). They include:

- Visible surface raster (width): Reducing the surface raster’s toolpath width eliminates porosity and improves the overall appearance of the part.

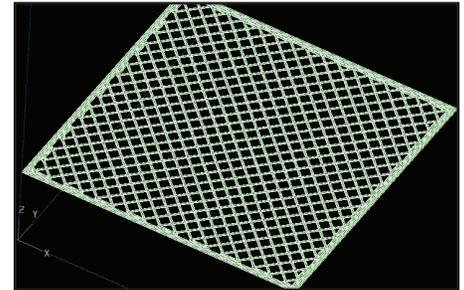


Figure 45: Sparse – double dense fill uses crossing rasters on each layer.

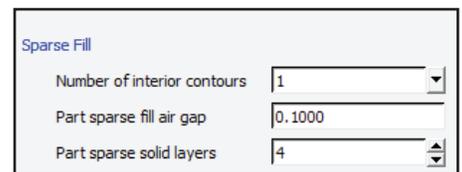


Figure 46: Advanced parameters for sparse fills include number of interior contours, solid layers and size of air gaps.

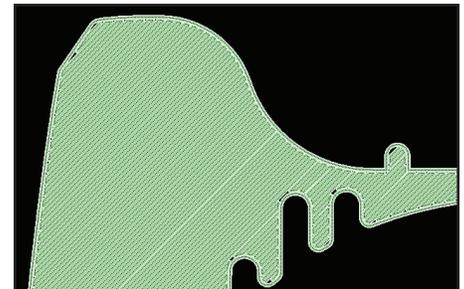


Figure 47: Enhanced mode produces narrow rasters with a small negative air gap for smooth surfaces.

Material	Tip Size	Visible Raster mm (in)	Visible Raster Air Gap mm (in)	Internal Raster mm (in)	Internal Raster Air Gap mm (in)
ABS-M30, ABS-M30i, ABSi, PC-ABS	T-10	0.23 (0.009)	-0.03 (-0.001)	0.41 (0.016)	0.000
	T-12	0.30 (0.012)	-0.03 (-0.001)	0.51 (0.020)	0.000
	T-16	0.43 (0.017)	-0.03 (-0.001)	0.61 (0.024)	0.000
	T-20	0.56 (0.022)	-0.03 (-0.001)	0.81 (0.032)	0.000
ABS-ESD7	T-12	0.30 (0.012)	-0.03 (-0.001)	0.51 (0.020)	0.000
	T-16	0.43 (0.017)	-0.03 (-0.001)	0.61 (0.024)	0.000
PC-ISO PC	T-12	0.30 (0.012)	0.000	0.51 (0.020)	0.000
	T-16	0.43 (0.017)	0.000	0.61 (0.024)	0.000
	T-20	0.56 (0.022)	0.000	0.81 (0.032)	0.000
PPSF	T-16	0.43 (0.017)	-0.03 (-0.001)	0.61 (0.024)	0.000
	T-20	0.56 (0.022)	-0.03 (-0.001)	0.81 (0.032)	0.000
ULTEM 9085	T-16	0.43 (0.017)	-0.03 (-0.001)	0.61 (0.024)	0.000
	T-20	0.56 (0.022)	-0.03 (-0.001)	0.81 (0.032)	0.000

Table 3: Enhanced mode default settings.

- Visible surface rasters air gap: Using a negative air gap on surface rasters can eliminate slight air gaps between rasters, thereby increasing density and improving surface finish.
- Internal raster (width): Increasing the internal raster's toolpath width will reduce build time and increase vertical layer strength.
- Internal raster air gap: Increasing the internal raster's air gap will decrease build time.

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, use custom groups to apply user-defined values to the style and width of the outer shell (the contours) of individual features. The first approach uses the Multiple Contours 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, enter a contour width that is an even multiple of the feature's thickness because all curves must be closed. For example, a 2.0 mm (0.080 in) wall could potentially have a contour of 1.0 mm (0.040 in), 0.5 mm (0.020 in), 0.3 mm (0.013 in) or 0.25 mm (0.010 in). So, begin by measuring the feature and dividing its value by even numbers (Figure 54).

Next, confirm that the desired width is available for the job's tip size. For a T16 tip, the 0.5 mm (0.020 in) value is the default 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.

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 "Multiple contours," which provides a user-definable perimeter

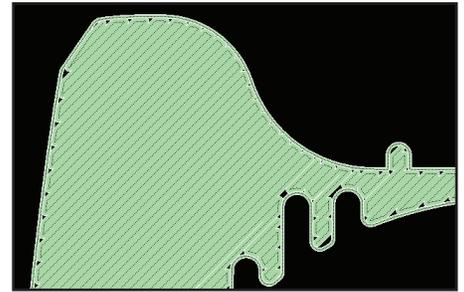


Figure 48: Internally, rasters are wider which reduces build time.

Raster angle	45.0
Visible surface style	Enhanced
Visible surface rasters	0.0170
Visible surface raster air gap	-0.0010
Surface max contours	3
Internal rasters	0.0240
Internal raster air gap	0.0000

Figure 49: Sample parameters for Enhanced mode.

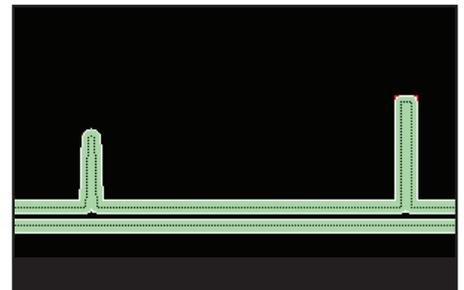


Figure 50: Defaults may produce voids in thin walls when rasters cannot fill between contours.

thickness made from one or more contours.

Next, set the value for contour width which was 0.5 mm (0.020 in) in the above example. Then, set the number of contours to two.

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

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 and view them (Toolpaths > Shade toolpaths or right click > Shade toolpaths). 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 multiple contours. The next approach to improve strength is very similar, but it uses the “Single contour 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 55).

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

Note: This method will not work with a 2.0 mm (0.080 in) feature using a T16 tip. The necessary value (1.0 mm /0.040 in) 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 56).

The last step is to generate toolpaths, shade and view them (Toolpaths > Shade toolpaths) or (Right click > Shade toolpaths). 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)

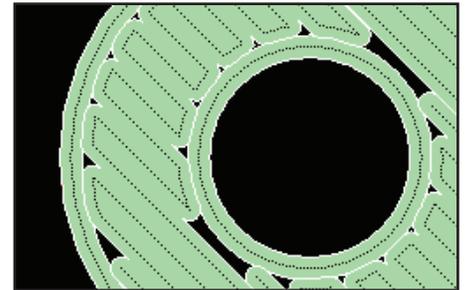


Figure 51: Defaults may produce porosity at the point of turnaround.

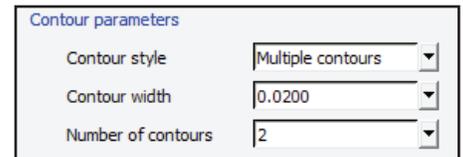


Figure 52: Custom group settings for solid fills using “Multiple contours.”

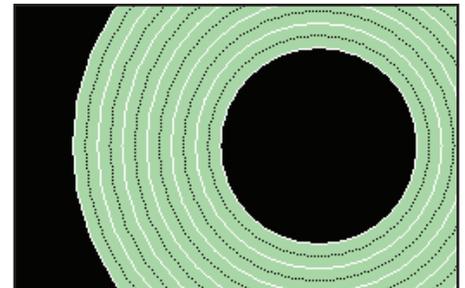


Figure 53: Result – solid fill for boss.

Steps:

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

Figure 54: Measuring procedure within Insight software.

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 57). To conceal the raster turnarounds and have a solid external surface, create and apply a custom group to the affected slices (Figure 58).

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 so that it overlaps with the contour 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 to cover the rasters below the contours for the next layer.

5.5.3. Add curves to custom group (Figure 59).

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

5.5.4. Generate toolpaths.

After generating toolpaths, shade and view all affected layers (Toolpaths > Shade Toolpaths). Use the “View layer plus layer above/below” setting to see if all raster turnarounds are concealed by the layer contours (Figures 60 and 61). If adjustments are necessary, return to the “Modify” function and adjust the “Number 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 increases.

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

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

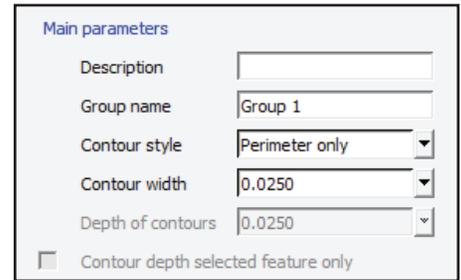


Figure 55: Custom group settings for solid fills using Perimeter only.

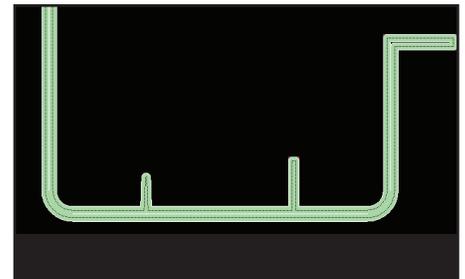


Figure 56: Result – solid fill for thin walls.

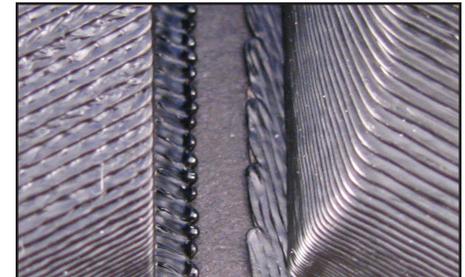


Figure 57: On shallow slopes, raster turnarounds may be visible (left). Custom groups can correct this (right).

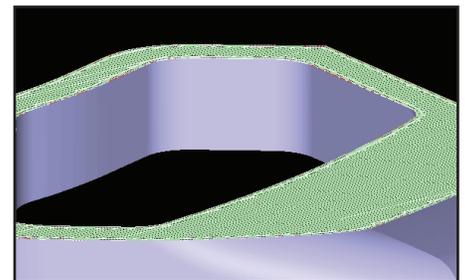


Figure 58: Area of interest (gradual slope) shown in this section view.

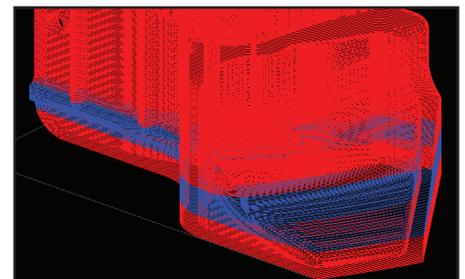


Figure 59: Select the curves in the area of gradual slopes (blue) and apply custom group.

- Generate supports
 - After slicing the file, set support style to “SMART” or “Sparse” (Support > Setup). Then click the “Create supports” icon.
- View slices (Figure 63).
 - Scroll through all the slices. Look for:
 - Layers with only model (red) and “sparse/SMART” supports (dark gray).
 - Support curves that can easily be removed.

Avoid applying this technique to supports that are surrounded by model curves. With either condition, the supports will be difficult to remove without damaging the part.

To swap model material 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 64). 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 65).

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 66 and 67).

5.7. Custom group: wide supports (**) (T1)

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

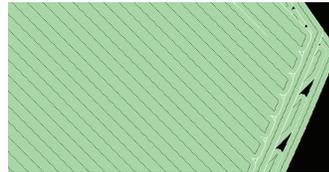


Figure 60: View toolpaths using the two-layer view to see if raster turnarounds are concealed. In this illustration, they are not.



Figure 61: With the correct “Multiple contours” setting, the turnarounds are concealed.



Figure 62: Visible turnarounds (left) from default settings. Concealed turnarounds (right) with custom group.

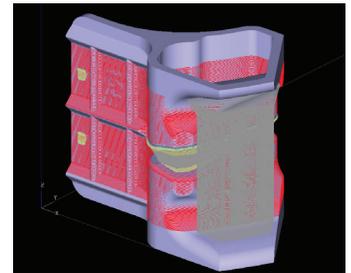


Figure 63: View slices to find those with only support (gray) and model (red).



Figure 64: Create custom group using a template for SupportSparse.

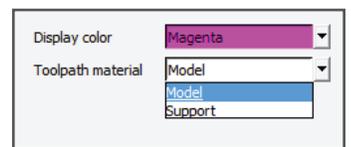


Figure 65: Change Toolpath material to “Model.”

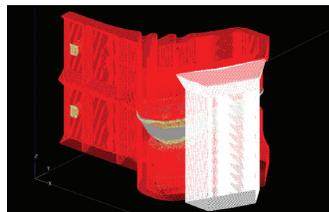


Figure 66: Select the curves for the supports that will be made from model (white) material.

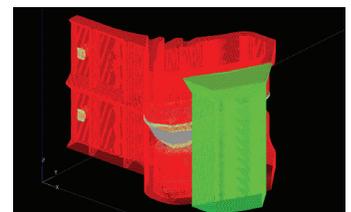


Figure 67: Apply the custom group (green) to build supports from model material.



Figure 68: Support tower with wide air gaps to decrease build time.



Figure 69: Top-down view shows wide spacing between support rasters.

gap between each support raster (Figures 68 and 69).

The wide support procedure is defined in a custom group so that it increases the air gap and replaces support material with model material (Figures 70 and 71).



Figure 70: Default Sparse support toolpaths (right).

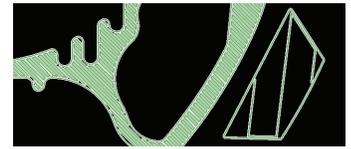


Figure 71: Wide air gap supports (right).

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 72).

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.

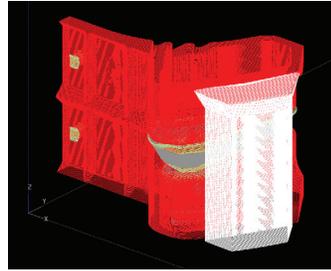


Figure 72: Select support towers that are accessible and do not surround part features.

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

5.7.2. Modify toolpath settings (Figure 73).

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 19.1 mm (0.75 in) 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 74).

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 deltas to 99. These settings place a layer of

Figure 73: Settings for Custom Group 1.

Figure 74: Custom Group 2 adjustments.

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 75):

- Bottom 10 layers: Custom Group 1 (-30° raster angle)
 - The wide supports cut across the default supports (if parallel to them, they would “fall through the cracks”). (Figure 76).
- 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
 - Default supports are required because this is 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 77) that appear in the custom group pane (e.g. by group name, color or curve type).

6. Throughput & Production Planning

When compared 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.

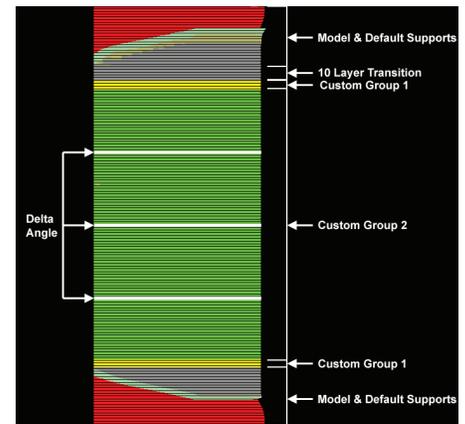


Figure 75: Toolpath regions after applying Custom Groups 1 and 2.



Figure 76: Wide supports in Custom Group 1 run perpendicular to default supports.

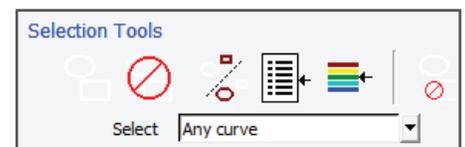


Figure 77: Use the selection tool when selecting curves.

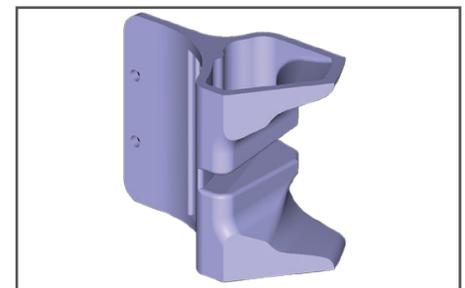


Figure 78: Build small lots of parts, even one-off, without concern for setup and changeover.

- It is an unattended operation.

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

- No setup (changeover) time required (Figure 78).
 - Small batch runs and one-off builds are economical and efficient. Parts can be built one at a time and as needed.
- Production mix is unlimited (Figure 79).
 - 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.
 - The tallest part will carry most of the process overhead.
- Builds can be matched to shift schedules (Figure 80).
 - To maximize throughput and minimize downtime, consider the shift schedule when laying out builds, launching runs and planning the production schedule.
 - Match jobs with the start and stop times of shifts. Avoid scheduling jobs that will consume the better part of a weekend and jobs that will run over a long holiday. It is inefficient to end a build when there is no staff available for changeover.

The production planning and scheduling tools are:

- Control Center
- Insight software

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

This operation is basic and routine, but important.

In Control Center, add individual jobs saved from the software to maximize throughput. Additionally, plan the production of parts in coordination with the work schedules of those who will remove them and start the next job to maximize efficiency.

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 81). Repeat until all desired parts are in the pack.

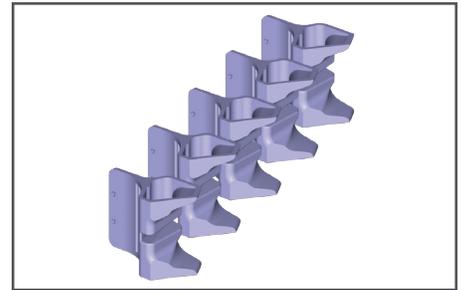


Figure 79: Combine multiple parts, in any combination, for building.

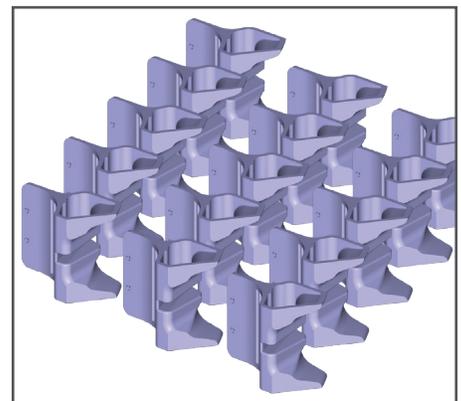


Figure 80: Maximize throughput with multi-part builds.

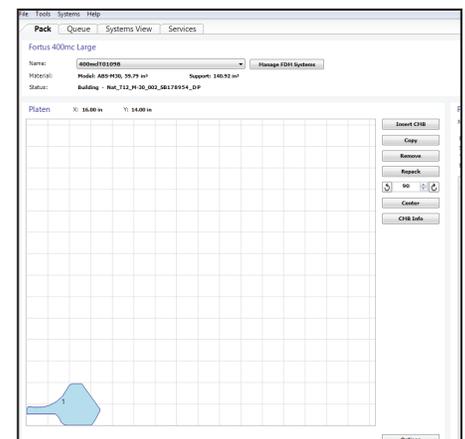


Figure 81: In Control Center, insert CMBs to create a pack.

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.

NOTE: The incremental time reduction will decline as parts are added (Figure 84).

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

Adjust the time to match the production schedule (Figure 83) by adding, copying or removing parts from the pack (Figure 82).

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

Control Center prevents parts from impeding on the bounding volumes of other parts. 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 the software prior to sending them to the Control Center (Figure 85). Using this technique, parts can be stacked, overlapped and nested. One example is the placement of a part in a center void of another, like a torus (donut shape).

Procedure:

6.2.1. Open and slice Part 1 (Figure 86).

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

6.2.2. Open and slice Part 2 (Figure 87).

6.2.3. Add Part 1 to Part 2’s job file (Figure 88).

- (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 that is commonly called the “CMB.” To build this set of nested tools at a later date, select “Insert CMB” while in the Control Center. Then, process the build as usual.

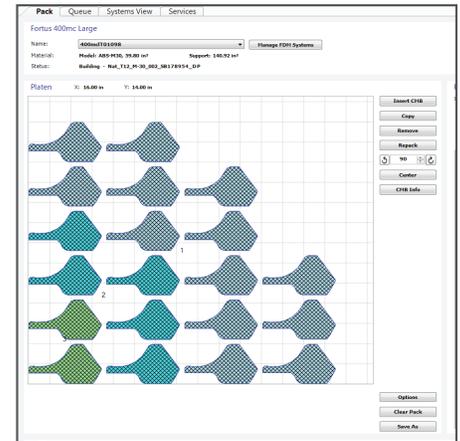


Figure 82: Duplicate or add parts and then place them in the pack.

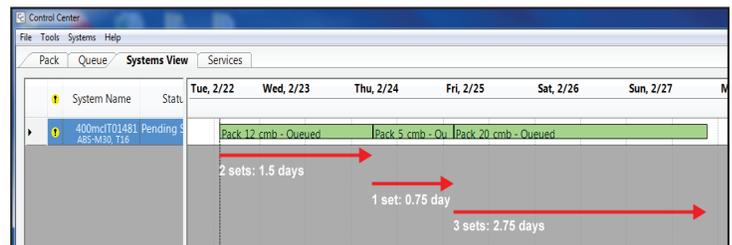


Figure 83: Sample production schedule that accounts for shift schedules and weekends.

7. KEY PROCESS CONSIDERATIONS

7.1 Obstacle details:

The following table presents common obstacles to end-use part production and recommended solutions.

OBSTACLE		RESOLUTION					
		Machine Operations	Design for FDM	Process Control	Secondary Processes	Part Orientation	Material Selection
Build Time	Throughput not competitive with other manufacturing processes.	✓	✓	✓		✓	
Porosity	Internal porosity impacts performance.		✓	✓	✓		
Operating Conditions	Demanding requirements when in service (e.g., mechanical loads, thermal exposure, and chemical exposure).		✓	✓			✓
Accuracy	Dimensional accuracy does not meet operation specifications.		✓		✓	✓	
Surface Finish	Visible layers and toolpaths impact performance or appearance			✓	✓	✓	
Material Cost	Higher cost than other manufacturing materials.		✓	✓			✓
Small Features	Resolution is too coarse to replicate.		✓	✓	✓		
Part Properties	Available materials do not offer properties that meet operational specifications.				✓		✓

Table 4: Common obstacles and resolutions

7.2 Resolution details:

- Machine operations
 - Increase throughput and efficiency by managing job schedules to leverage “lights-out,” automated operations.
 - Group parts into a single build.
- Design for FDM
 - Redesign parts to optimize the FDM process: self-supporting angles, offset surfaces, material removal and wall thicknesses.
- Process control
 - Use advanced Insight software tools for fill styles, custom groups (e.g. strength, porosity and build time).
 - Select appropriate slice heights for feature size, surface finish and build time.
- Secondary processes
 - For surface smoothness: mass finishing, vapor smoothing, secondary machining or sanding.
 - For accuracy and flatness: secondary machining.
 - For porosity: sealing or vapor smoothing.
- Part orientation
 - Position part to improve feature accuracy, surface finish, strength, and build time.
- Material selection

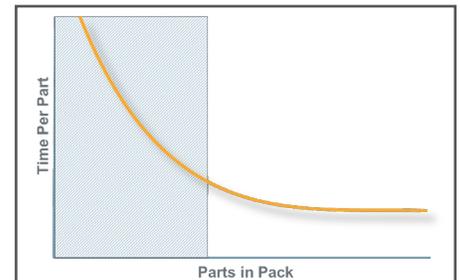


Figure 84: Note that the incremental time reduction will decline as parts are added.



Figure 85: CAD model of a nested part.

- Select the best material to meet the performance requirements (e.g. mechanical, thermal) instead of defaulting to what is in the machine.

8. Recap - Critical Success Factors

8.1. Leverage the advantages of additive manufacturing by using good FDM practices.

- Complex structures, including part consolidation
- Design freedoms
- Redesign and revision frequency
- Automated, lights-out operation
- Small-lot, on-demand production
- Lead time and cost reduction

8.2. Optimize parts for FDM.

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

8.3. Invest time accordingly.

- Pilot production – light adjustments
- Bridge-to-production – moderate adjustments; some optimization
- Full production – complete redesign; full optimization
- End-of-life production – moderate adjustments; full optimization

8.4. Eliminate adoption obstacles.

- Adopt FDM best practices.

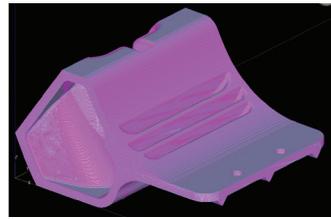


Figure 86: Open and slice the first part.

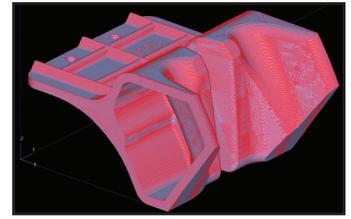


Figure 87: Open and slice the second part. (Example duplicates a single part.)

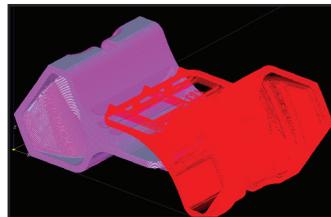


Figure 88: Combine the job files.



Figure 89: A redesigned end-use part from Nova Tech Engineering (PC-ABS).



Figure 90: Production assembly utilizing FDM end-use parts (ABS-M30).

CONTACT

For questions about the information contained in this document, contact Stratasys at www.stratasys.com/contact-us/contact-stratasys.



E info@stratasys.com / STRATASYS.COM

HEADQUARTERS

7665 Commerce Way, Eden Prairie, MN 55344
+1 888 480 3548 (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