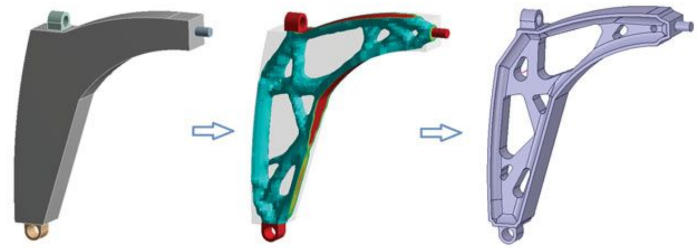
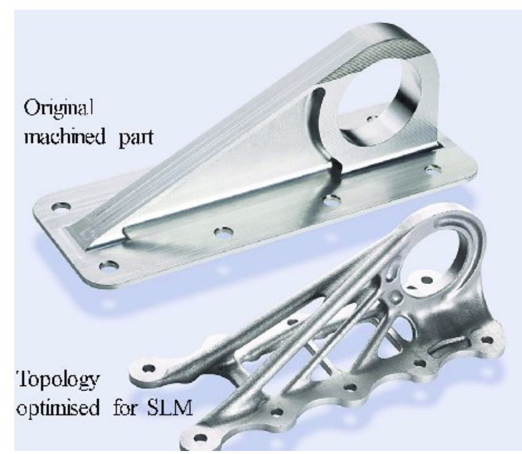
香港中文大學
The Chinese University of Hong Kong

MAEG5160: Design for Additive Manufacturing

Lecture 2: Design for polymer AM



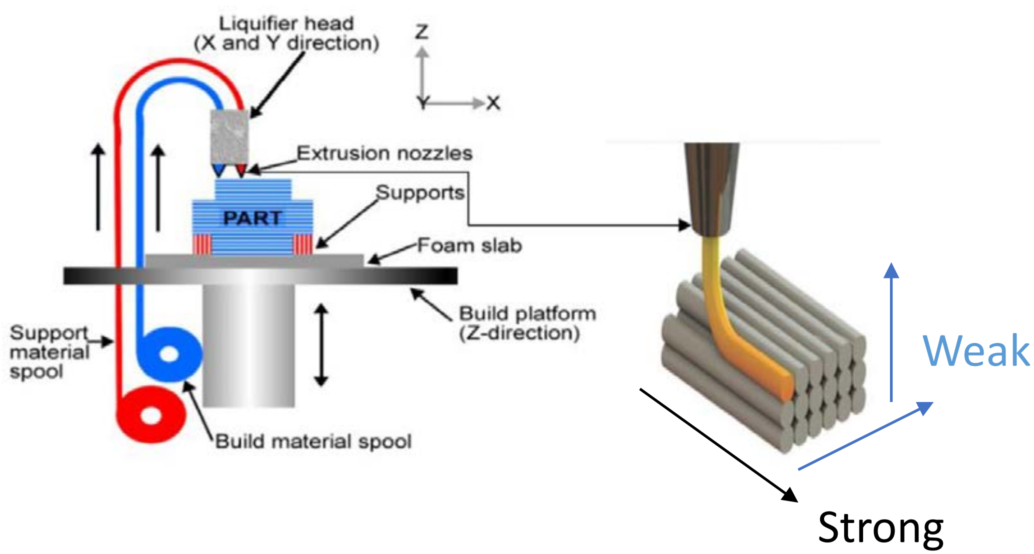


Prof SONG Xu
Department of Mechanical and Automation Engineering,
The Chinese University of Hong Kong.

Design for polymer AM: general guideline

1. Anisotropy

Anisotropy is the term used to describe the properties of a part in which the mechanical properties of the part are not the same in all directions. With all additive manufacturing technologies, there is always a certain amount of anisotropy in the vertical direction, between the layers. This is because the mechanical strength of the bond between each layer can be somewhat weaker than the mechanical strength within the layer itself.

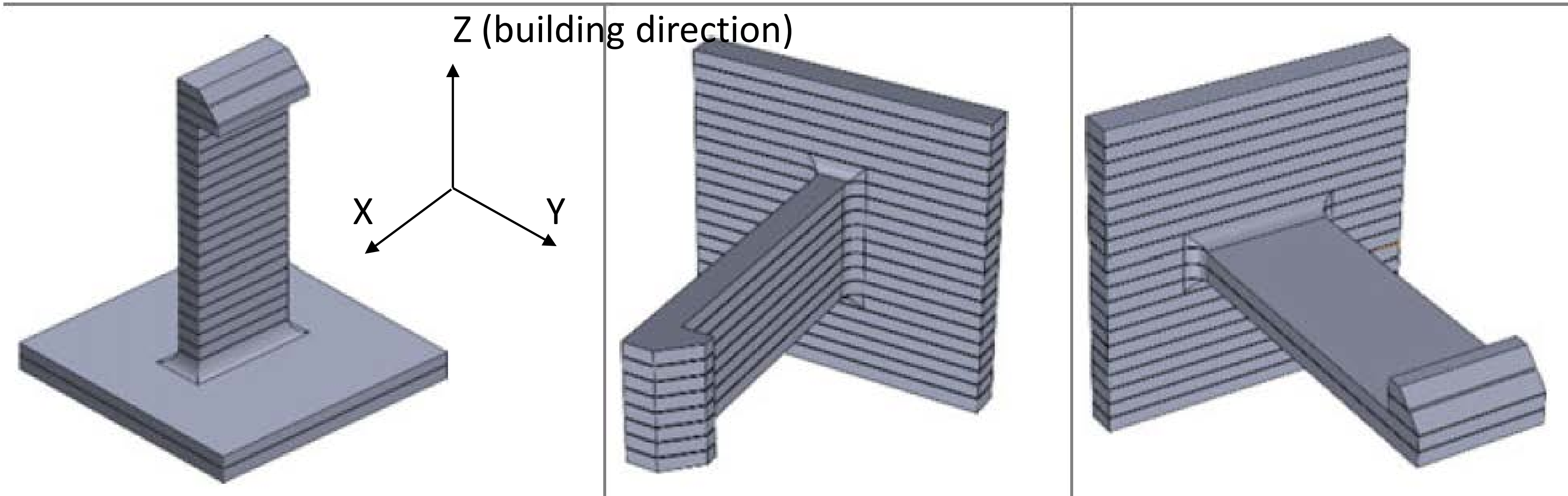


Elastic property is almost the same, but the non-linear behaviour can be different significantly

Design for polymer AM: general guideline

1. Anisotropy

Implication/example



Clip will be weak and, almost certainly, break	Good compromise clip, with decent spring and strong hook	Clip has the best spring strength and flexibility but a weak hook

Design for polymer AM: general guideline

2. Wall thickness

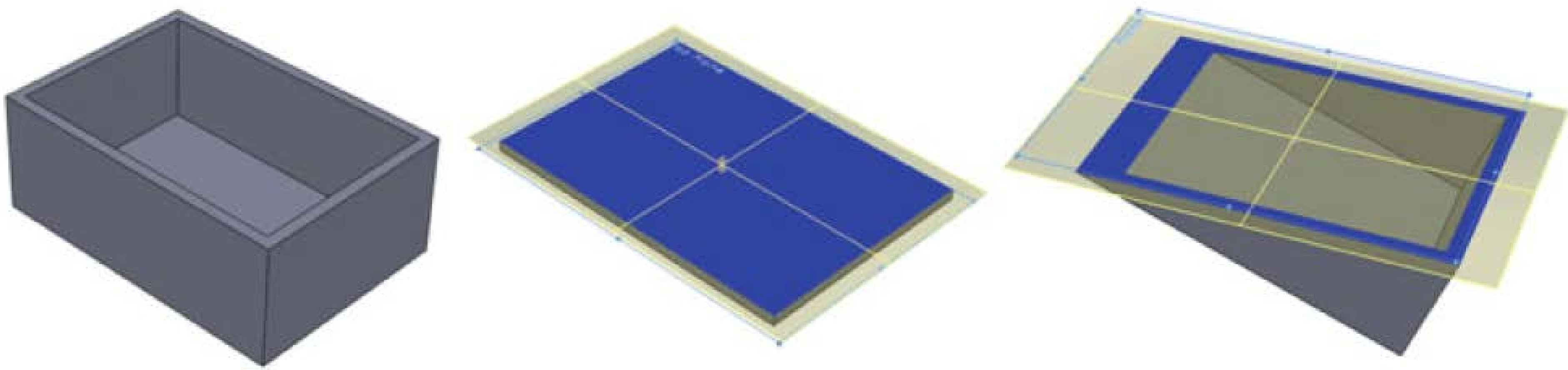
In general, for light-weight consumer products, this ranges from around 0.6–2.5 mm, and for more industrial heavy-duty industrial products, this can range from 3 to 5 mm. Though it is possible to create thinner walls, how successfully they will print will depend on the surface area of the wall, and the unsupported width to height ratio.

Large surface area flat thin walls will be hard to print without distortion and, depending on the AM technology used, may delaminate. A simple technique to avoid this problem, if the wall cannot be made thicker, is using ribs to reinforce the wall. A general rule of thumb is to use even wall thicknesses throughout the parts, as uneven wall thicknesses can create part distortion.

Design for polymer AM: general guideline

2. Wall thickness

In some case, the orientation in which the part is printed can be used to prevent large flat walls from warping. In the simple box example below, if the part is printed in the horizontal position, then there will be a large thin ‘sheet’ of polymer that gets melted, and may try to curl up and cause warping, or could even cause the machine to crash. In contrast, printing the part at a slight angle, typically above 10°, removes such large flat areas and can drastically reduce the risk of distortion. But, as with all AM guidelines, this too is a compromise, as printing the part at an angle may result in poorer surface finish than if it is printed straight.

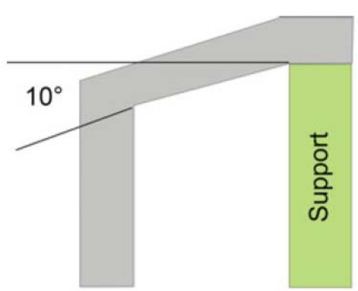


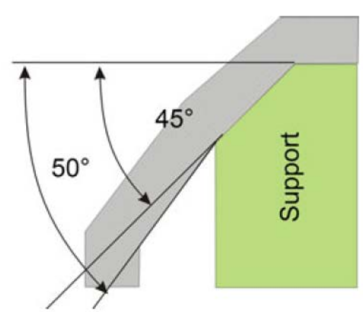
Design for polymer AM: general guideline

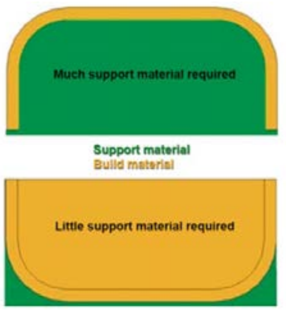
7.3 Overhangs and Support Material

Almost all polymer AM technologies, with the exception of powder bed fusion, and some binder jetting technologies, the printed parts require support material to support any overhanging features. Support material is a sacrificial material that is utilized during the printing process to allow any features that overhang, because it is not possible to print in air without the material collapsing, and is removed after the part has finished printing

There is usually a support 'angle' option in your 3D printing software that determines the angle in which the part requires support material (a very rough starter is 45 degrees). Some printers measure this angle from the vertical, while others measure it from the horizontal. It is therefore important to be aware of how each particular printer takes this measurement.





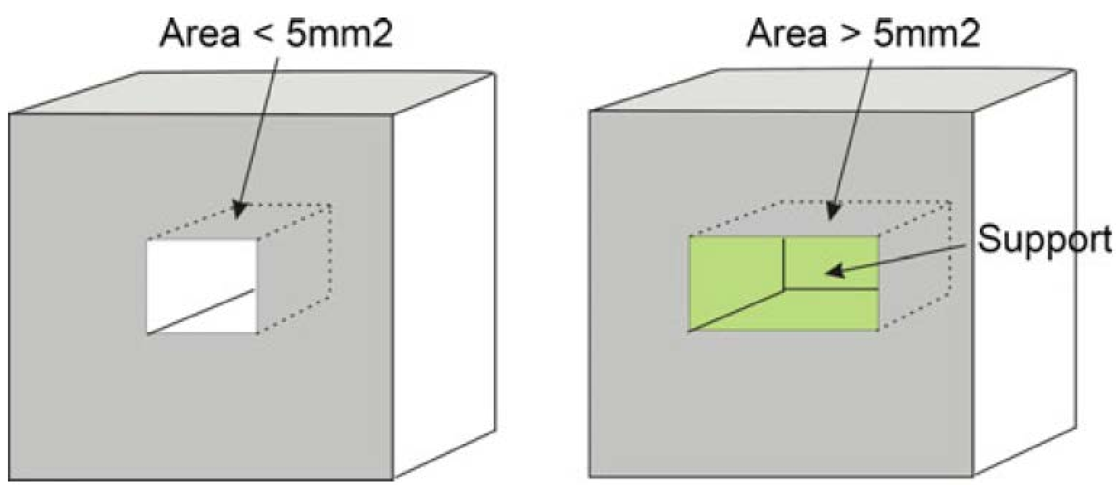


Design for polymer AM: general guideline

7.4 Holes

With AM technologies, print orientation can greatly affect the roundness of holes. To achieve the roundest possible holes, it is always best to have the holes printed in the vertical direction. Holes printed in the horizontal position will suffer both from the stair-step effect, and from some sagging that may make the holes slightly elliptical.

Some AM software also allows you to set the surface area below which no support material is required. The benefit of this is that some material is saved and a slightly shorter print time is achieved. Moreover, this approach can also result in less support material to remove from holes. The risk is that, if the surface area is too high, there may be some sagging of material on the top surfaces of the overhangs.



Design for polymer AM: general guideline

7.5 Ribs

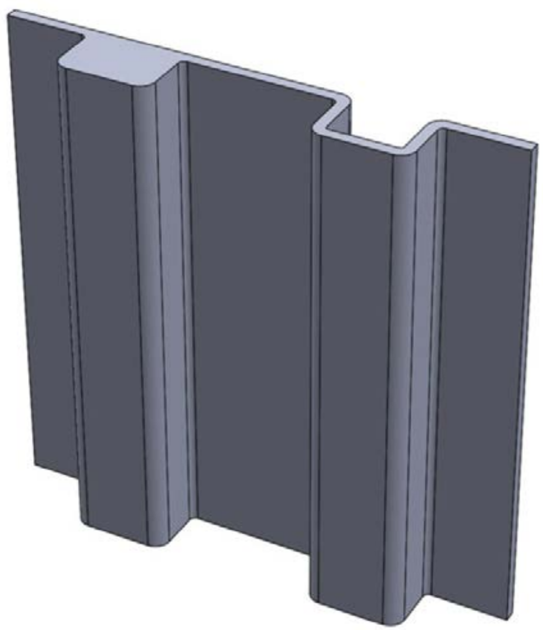
Most polymer AM materials are slightly less rigid than their injection molded counterparts. This means that large surface areas and walls can be quite flexible, and can sometimes develop some distortion during the printing or cooling down process. The simplest way to make walls more rigid, and to minimize the risk of distortion is to design the part with ribs to reinforce large thin areas.

Thickness of ribs: 75% of wall thickness	
Height of ribs: <3× thickness	
Rib spacing: >2× thickness	
Always fillet the point where ribs meet the wall	

Design for polymer AM: general guideline

7.6 Avoid unnecessary material

When designing AM parts, it is therefore important to avoid having large masses of material that serve no functional purpose, as they slow down production time, increase part weight, and can cause part deformation. The very simplest approach to this, also described in the economics of AM, is to ‘shell’ the thicker sections of the part. This will minimise print time and cost. But a decision also needs to be made as to whether to leave the excess material (unsintered powder, liquid photopolymer resin, or support material, etc.) inside the shelled part, or to design in ‘salt-shaker’ holes so that the excess material can be removed. To remove internal support material, larger holes may be necessary.

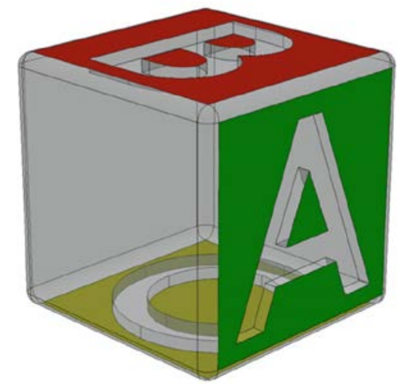


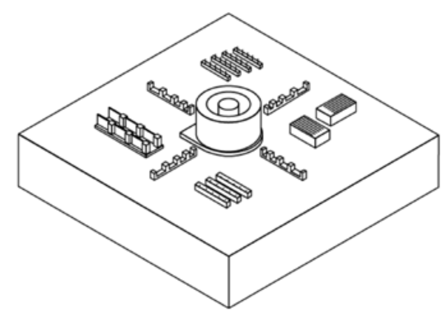
Design for polymer AM: general guideline

7.7 Font size and small details

Typically, for most polymer AM technologies, details are visible down to about 0.5 mm (though in some cases they can be as small as 0.2 mm high × 0.2 mm wide), but this must be tested for each model of printer. Also, surfaces that are in contact with support material may not be possible to reproduce with as fine a detail as those surfaces that do not require support material.

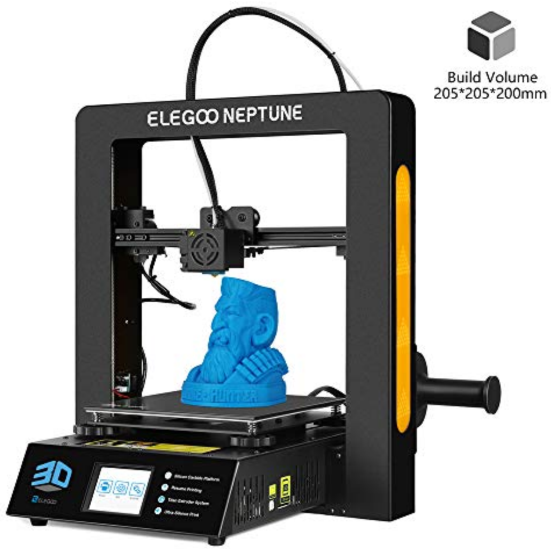
Fonts, and other small details, can either be sunk into the wall (debossed) of the part or can protrude from the wall of the part (embossed). In general, it may be preferable to have them sunk into the walls of the part for two reasons: Firstly, it removes material form the part which means a slightly reduced print time and, second, it reduces the risk of the font or details being sanded off during post-processing. However, there is no problem with using embossed fonts, if required. But greater care may need to be taken during the part post-processing. A font size that usually works on all surfaces is 14pt, and at least 0.4mm(0.016 in.) in depth. On vertical surfaces one can go down to about an 8pt font.





ISO/ASTM 52902:2019 Additive manufacturing — Test artifacts — Geometric capability assessment of additive manufacturing systems

Design for polymer AM: material extrusion



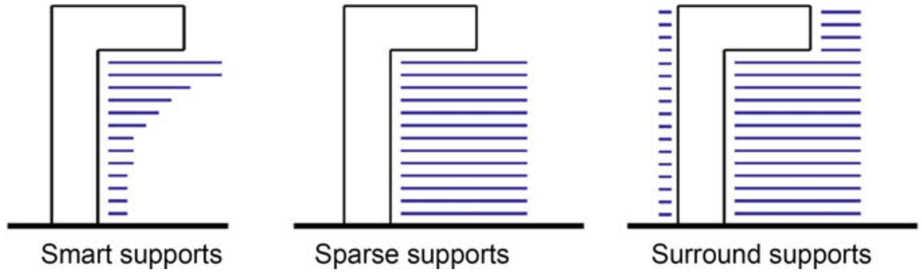
There is a vast difference in accuracy and tolerance between different material extrusion systems. They also vary depending on geometric features and print orientation. The only sure where to know the accuracy and tolerance of any particular system is to print a test reference part and to measure it.

- Accuracy is how close the part is to the CAD model data.
- Tolerance is the acceptable degree of variation.

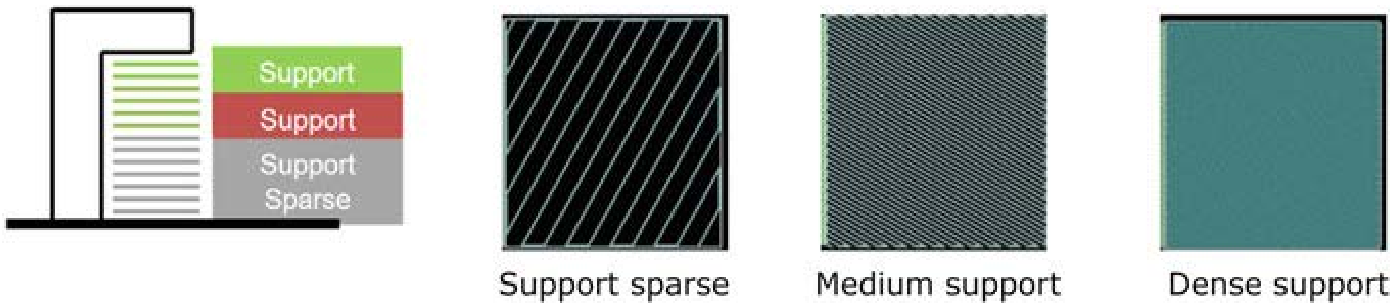
Layer thickness	0.1–0.3 mm (0.005–0.013 in.)
Accuracy	±0.1 or ±0.03 mm per 25 mm (±0.005 in. or ±0.0015 in. per inch), whichever is greater
Tolerance	Reality rule of thumb for Material Extrusion: typically 0.25 mm (0.01 in.)
Smallest feature size	Around 1 mm (0.04 in.)

The first decision that has to be made when printing material extrusion parts is what layer thickness to use. In general, the thinner the layer, the better the surface quality, particularly on rounded parts, as the stair-step effect will be much less visible. However, the thinner the layer, the longer the part will take to print. A 0.1 mm layer thickness will take three times longer to print than a 0.3 mm layer thickness. If a part is mainly composed of flat geometric features in the vertical direction, then printing it with a thicker layer thickness will not produce much worse surface finishes that a thin layer thickness but will print much faster. If the part is made up of many curved surfaces, then a thin layer thickness may be preferable in order to achieve curved surfaces that are as smooth as possible.

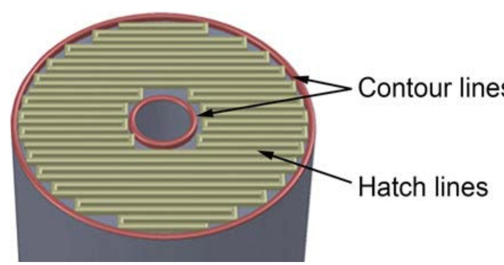
Design for polymer AM: material extrusion



Support for the overhang



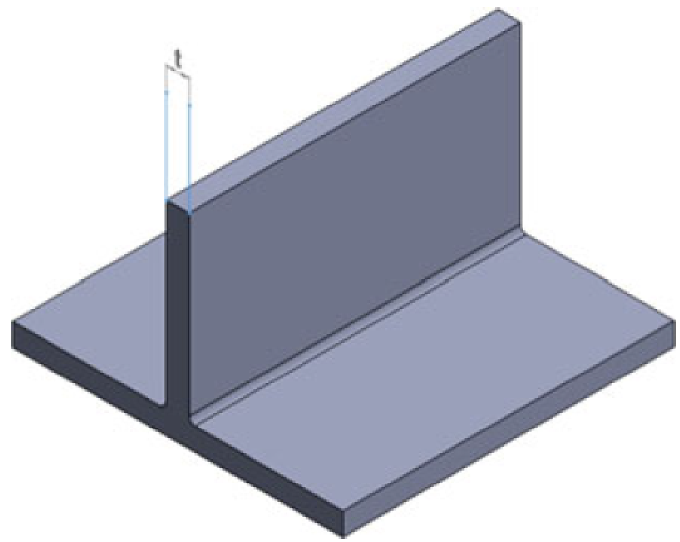
Support density, too dense is not easy to remove; too sparse cannot support



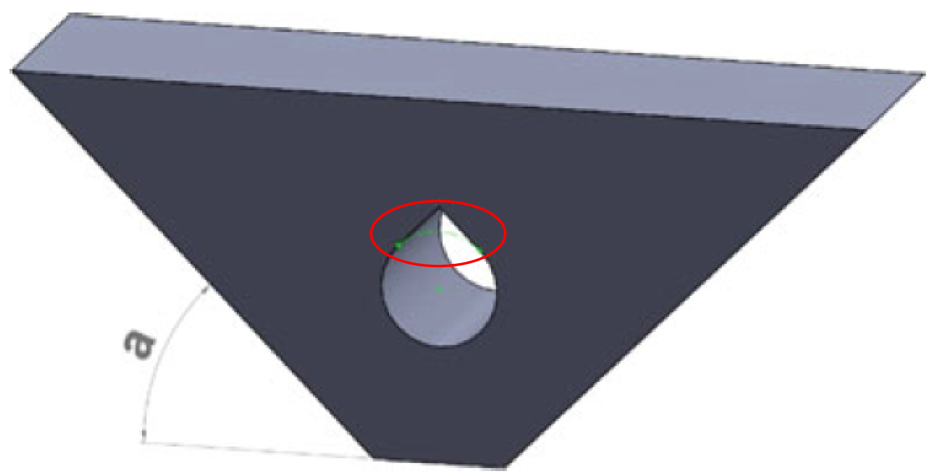
Interior filling option

Design for polymer AM: material extrusion

Process variable	Wall thickness (t)	
Layer thickness	Minimum	Recommended minimum
0.18 mm (0.0071 in.)	0.36 mm (0.014 in.)	0.72 mm (0.028 in.)
0.25 mm (0.0098 in.)	**0.50 mm (0.02 in.)**	**1.00 mm (0.039 in.)**
0.33 mm (0.013 in.)	0.66 mm (0.026 in.)	1.32 mm (0.052 in.)
	2 layers	*4 layers*



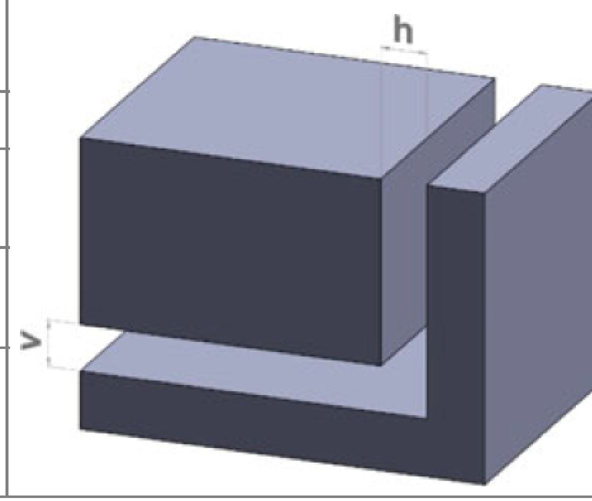
Maximum overhang angle (a)
45° This is a safe default number. But the angle can vary greatly from printer brand to printer brand, and depends on the desired surface quality



Horizontal holes (e.g., cooling channel profiles) can often be modified into teardrop or ovals shapes to minimize the need for internal supports that are hard to remove.

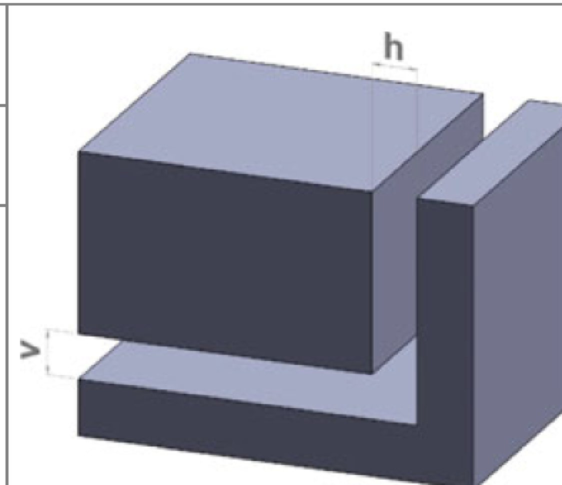
Design for polymer AM: material extrusion

Process variable	Minimum clearance	
Layer thickness	Horizontal (h)	Vertical (v)
0.18 mm (0.0071 in.)	0.36 mm (0.014 in.)	0.18 mm (0.0071 in.)
0.25 mm (0.0098 in.)	**0.50 mm (0.02 in.)**	**0.25 mm (0.0098 in.)**
0.33 mm (0.013 in.)	0.66 mm (0.026 in.) *2 layers*	0.33 mm (0.013 in.) *1 layer*



Feature Type: Clearances Between Moving Parts with Soluble Supports

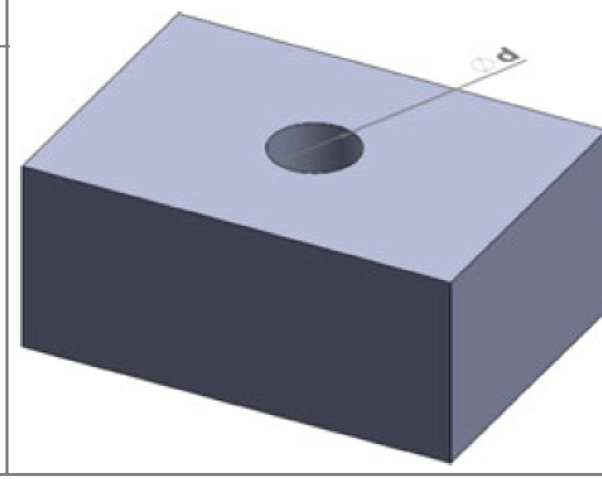
Process variable	Minimum clearance	
Layer thickness	Horizontal (h)	Vertical (v)
0.18 mm (0.0071 in.)	0.36 mm (0.014 in.)	Adequate access to facilitate supports removal *2 layers*
0.25 mm (0.0098 in.)	**0.50 mm (0.02 in.)**	
0.33 mm (0.013 in.)	0.66 mm (0.026 in.)	



Feature Type: Clearance Between Moving Parts with Break-Away Support Material

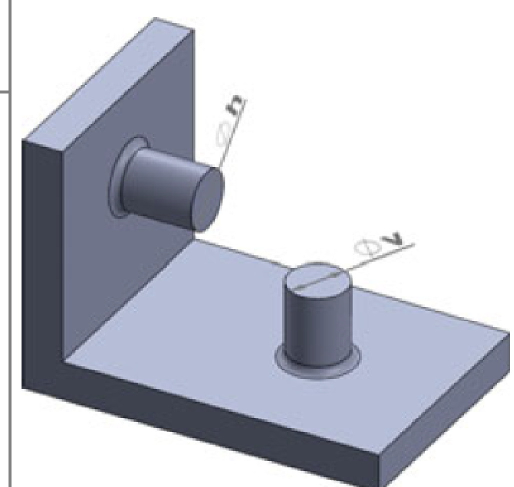
Design for polymer AM: material extrusion

Required diameter (d)	CAD model diameter
5.0 mm (0.197 in.) 10.0 mm (0.394 in.) 15.0 mm (0.591 in.) 20.0 mm (0.787 in.)	5.2 mm (0.205 in.) 10.2 mm (0.402 in.) 15.2 mm (0.598 in.) 20.2 mm (0.795 in.)



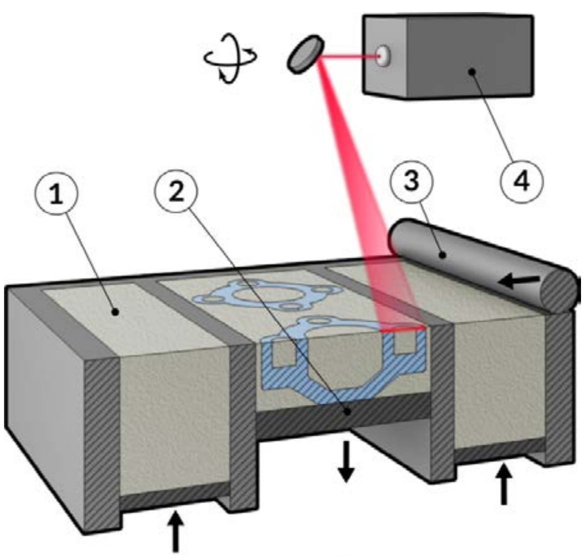
Vertical Circular Hole

Minimum diameter for vertical pins (v)	Minimum diameter for horizontal pins (h)
2.0 mm (0.079 in.)	2.0 mm (0.079 in.)

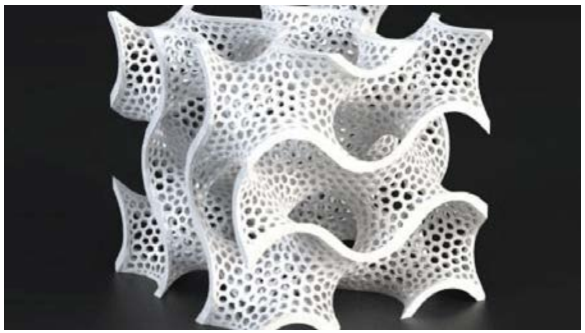


Circular Pin

Design for polymer AM: Polymer Powder Bed Fusion



PA (nylon)



Use a mix of virgin powder and used powder. Typically, the ratio of new to old powder ranges from 20/80 to 35/65.

The part being constructed does not normally require supports, since the unfused powder surrounding the part provides sufficient support. This gives the designer greater freedom than most other AM system.

Parts created using polymer powder bed fusion usually have some degree of anisotropy in their material properties, particularly for small features that are less than about 25mm in surface area in the vertical direction.

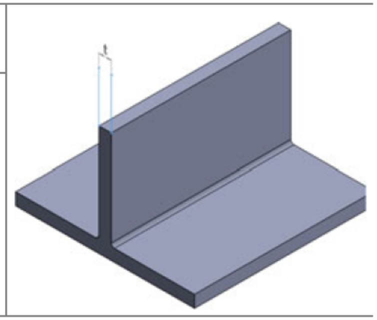
Another characteristic of this process is a pronounced granular roughness on the surface of parts. This can be reduced by various post-processing techniques (such as tumbling with abrasive media), but this will have an impact on part accuracy.

Avoid large masses of material, and the most common technique for removing large masses of material is to shell

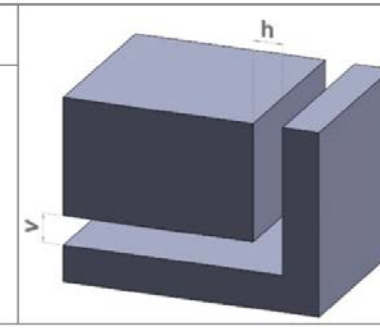
Layer thickness	0.1 mm (0.005 in.)
Accuracy	±0.3% lower limit of ±0.3 mm (0.010 in.)
Tolerance	±0.25 mm (0.010 in.) or ±0.0015 mm/mm (0.0015 in./in.)—whichever is greater
Smallest feature size	Around 0.5 mm (0.04 in.)

Design for polymer AM: Polymer Powder Bed Fusion

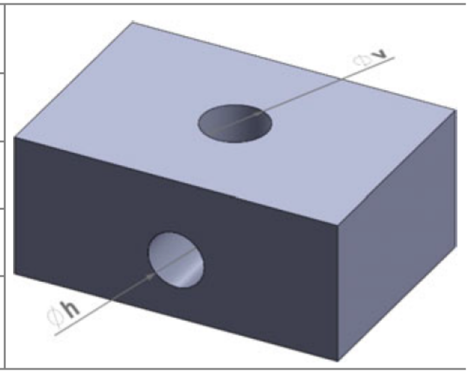
Minimum wall thickness (t)	Recommended minimum wall thickness (t)
0.6–0.8 mm (0.031 in.)	1.0 mm (0.039 in.)



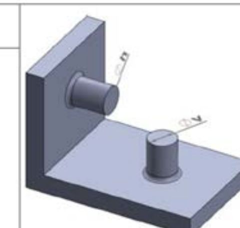
Minimum horizontal clearance (h)	Minimum vertical clearance (v)
0.5 mm (0.02 in.)	0.5 mm (0.02 in.)



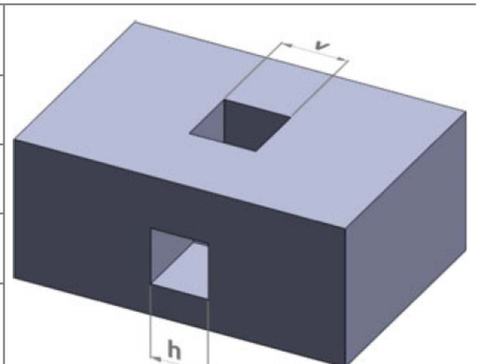
Process variable	Minimum diameter	
Wall thickness	Vertical hole (v)	Horizontal hole (h)
1 mm (0.039 in.)	0.5 mm (0.019 in.)	1.3 mm (0.051 in.)
4 mm (0.157 in.)	0.8 mm (0.031 in.)	1.75 mm (0.069 in.)
8 mm (0.314 in.)	1.5 mm (0.059 in.)	2.0 mm (0.079 in.)



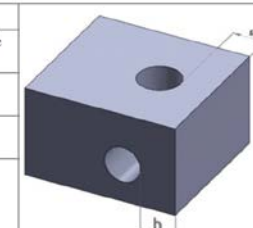
Minimum diameter for vertical pins (v)	Minimum diameter for horizontal pins (h)
0.8 mm (0.031 in.)	0.8 mm (0.031 in.)



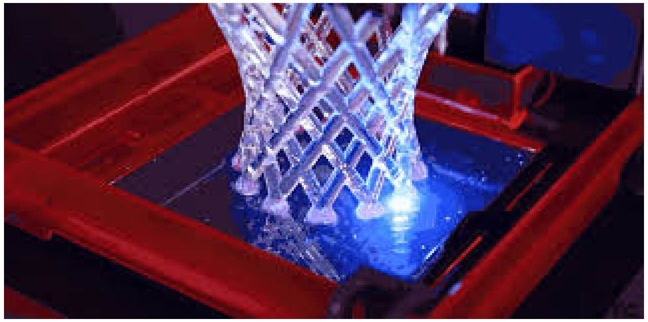
Process variable	Minimum diameter	
Wall thickness	Vertical hole (v)	Horizontal hole (h)
1 mm (0.039 in.)	0.5 mm (0.019 in.)	0.8 mm (0.031 in.)
4 mm (0.157 in.)	0.8 mm (0.031 in.)	1.2 mm (0.047 in.)
8 mm (0.314 in.)	1.5 mm (0.059 in.)	1.3 mm (0.051 in.)



Design variable	Minimum distance to edge	
Hole diameter	Vertical hole (v)	Horizontal hole (h)
2.5 mm (0.098 in.)	0.8 mm (0.031 in.)	0.8 mm (0.031 in.)
5.0 mm (0.197 in.)	0.9 mm (0.035 in.)	0.95 mm (0.037 in.)
10.0 mm (0.394 in.)	1.05 mm (0.041 in.)	1.0 mm (0.039 in.)

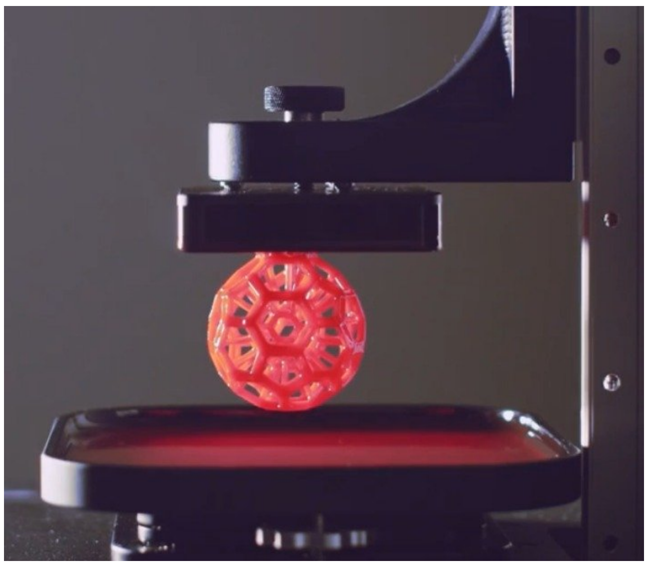


Design for polymer AM: Vat Photopolymerisation



SLA resolution in the XY-direction is dependent on the laser spot size and can range anywhere from 50 to 200 µm. Resolution in the Z-direction varies from 25 to 200 µm depending on the choices of layer thickness allowed by the machine.

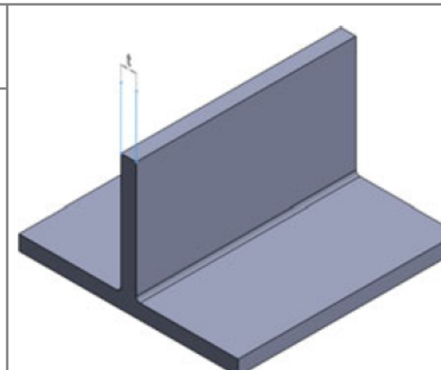
SLA does require support material for overhanging features. This is because the uncured resin is not viscous enough to support features on its own. Any unsupported overhangs must be kept less than 1.0 mm in length and at least 20° from horizontal.



SLA is one of the few processes where the parts are relatively isotropic. This is because the layers chemically bond to one another as they print, resulting in near identical physical properties in the X, Y, and Z-direction.

Design for polymer AM: Vat Photopolymerisation

SLA machines can print solid, dense models but, if the print is not intended to be a functional part, ***shelling the model to be hollow*** can significantly reduce the amount of material needed as well as reduce the print time. It is recommended that the ***walls of the hollowed part be at least 2mm thick*** to reduce the risk of failure during printing. If printing a hollow part, ***drainage holes must be added*** to allow the uncured resin to be removed from the part. If left inside the part, the uncured resin can create a pressure difference within the hollow chamber, and can cause what is known as "cupping". Small failures such as cracks or holes propagate throughout the part and will, eventually, cause complete failure, or part explosion, if not corrected. ***Drain holes should be at least 3.5 mm in diameter***, and at least one hole must be included per hollow section, although two holes can make the resin much easier to remove.

Minimum wall thickness for supported walls (t)	Minimum wall thickness for unsupported walls (t)	
0.4 mm (0.016 in.)	0.6 mm (0.023 in.)	

Wall thickness

Design for polymer AM: Vat Photopolymerisation

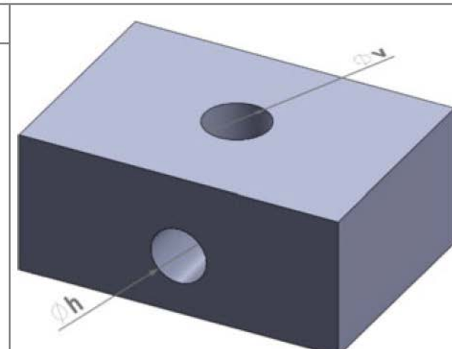
Embossed details (including text) include any features on the model that are raised slightly above the surfaces around them. These must be ***at least 0.1 mm in height above the surface*** of the print to ensure the details will be visible.
Engraved details (including text) include any features which are recessed into the model. These details are at risk of fusing with the rest of the model while printing if they are too small, so ***these details must be at least 0.4 mm wide and at least 0.4 mm deep***.

If parts are being made that need to connect together, it is always best have a certain tolerance between the parts that fit together. For SLA, these tolerances are:

- 0.2 mm clearance for assembly connections.
- 0.1 mm clearance will give a good push or snug fit.

If interlocked moving parts are being printed, then the tolerance should be 0.5mm between the moving parts.

Minimum diameter h and v
0.5 mm (0.019 in.)



Minimal hole size

Thank you for your attention

and don’t forget ink (material) jetting and binder jetting~