Design for Additive Manufacturing (DfAM) is when designers seek to create a product design that takes advantage of the unique capabilities of AM

- re-design for AM is useful because it can yield benefits such as a reduction in the use of material or the consolidation of several parts into one
- encourage engineers and designers to consider the strategic benefits of AM before concentrating on detailed design

Rules for DfAM

- AM should only be used if that part cannot be easily made using some other manufacturing technology (when complexity of a part reaches point where can't be manufactured)
- add useful cosmetic details
- fillet (round) all corners: makes product more ergonomic, comfortable to hold and use, removes risk of sharp edges, reduces stress concentrations which can occur on sharp corners and transitions which could affect the strength of the product
- can't design for AM without thinking of print orientation: quality (strength, material properties, surface quality, amount of support) of every part, with every AM is directly related to print orientation
  - print orientation determines the direction of the anisotropy, which will always be in the Z direction, or vertical print direction
  - if anisotropy is an important factor, part should be orientated so as to have the features requiring maximum strength to be printed horizontally
  - if rounds of holes is also critical, they are best printed in the vertical orientation; holes printed horizontally while suffer from the stair-step effect, and will also be slightly elliptical
  - best print orientation is usually the one that minimizes the total height of the build
- design to minimize large masses of material
- design to minimize support material

Design to Avoid Anisotropy

Anisotropy — the difference in mechanical properties of a part in the vertical direction. When designing, good to have an understanding of the orientation that it will be built in so that it can be designed with the optimal number of features that be subjected to high forces oriented in the X-Y plane

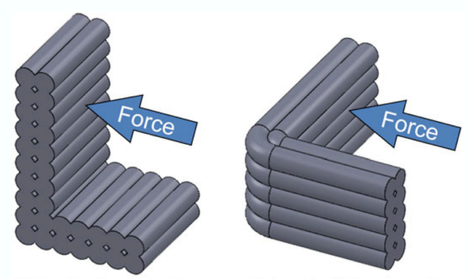

Fig. 3.6 Anisotropy, or the weakness between layers, will cause apart to react differently under different forces

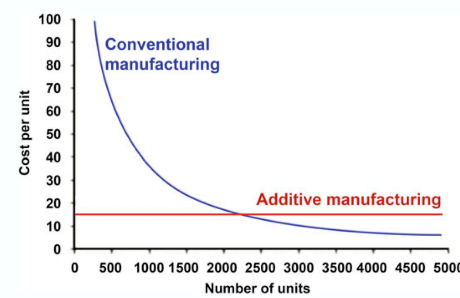

Fig. 3.7 Graph showing how costs decrease as quantity increases with conventional manufacturing but remain constant with AM

Economics of Additive Manufacturing

- If single part takes 10h to print, machine cost for that part is $650. With metal AM, print times often higher (100h, $650 for single part).
- Raw material costs (chrome and titanium most expensive)
- Typically 10% material wastage — support material, powder, etc.
- Post processing costs

Time factors not affected by design

- recoating time metal AM

Design to minimize print time

- use of contour lines and hatch patterns in AM
- shelling to specified wall thickness (bulk of material removed from inside)

- fill solid parts with honey-comb, lattice, or porous material
- may have to compromise print time vs. mechanical properties and geometry properties vs. surface quality vs. support material removal

AM Unique Capabilities

In comparison with most other manufacturing processes…

- shape complexity: possible to build virtually any shape
- hierarchical complexity: features can be designed with shape complexity across multiple size scales (features at one size scale can have smaller features added to them, such as textures added to surfaces of parts)
- functional complexity: functional devices (not just individual pieces) can be produced in one build
- material complexity: material can be processed one point, one layer at a time as a single material or as a combination of materials

— Polymer Design Guidelines —

Designing for Material Extrusion

Material extrusion, aka Fused Deposition Modeling (FDM), is an AM process in which material is selectively dispensed through a nozzle much like a hot-glue gun. Materials used are typically thermoplastic polymers and the part being constructed normally requires support structures for overhanging features.

Material Extrusion Accuracy and Tolerances

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

Layer Thickness

- thinner the layer, better the surface quality, particularly on rounded parts as the stair-step effect is much less visible
- thinner the layer, longer it will take to print (0.1mm will take three times longer than 0.3mm)
- if part made of many curved surfaces, then thin layer thickness preferable in order to achieve curved surfaces that are as smooth as possible

Support Material

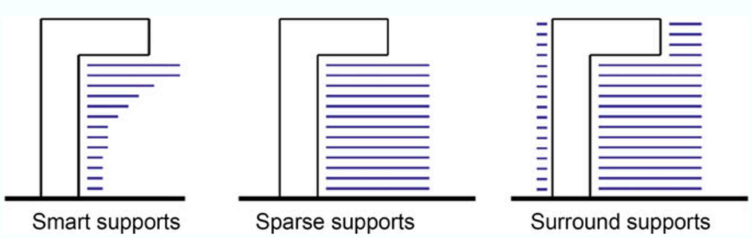

Fig. 8.1 Types of support material

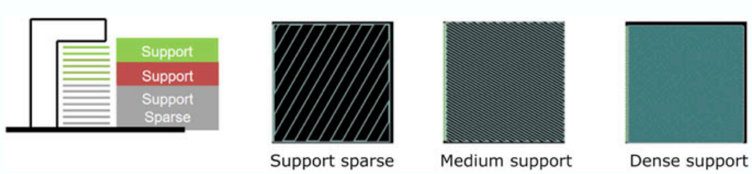

Fig. 8.2 Examples of different support material densities

Fill Style
- infill percentage: how dense scaffold structure should be
- infill percentage of 0% means there is a void (automatic way to let 3D printer know to shell)
- 100% means completely filled through/solid material
- in terms of *mechanical strength*: infill percentages about 50% can have diminishing returns
- parts created using polymer material extrusion usually have anisotropic material properties (part will be weaker in the Z direction than it is in the X and Y directions)
  - therefore, highly stressed features should be built in a horizontal rather than a vertical direction, unless can lower material properties specifications

Other Considerations
- “stair-stepping” on the surface of parts can be reduced by various post-processing techniques
- strand of polymer exiting nozzle is 0.4mm wide but the wall thickness is 0.9mm, software must make a decision as to whether or not to push an extra strand of polymer between two 0.4mm tracks (this is machine brand and model dependent, recommend testing)
- warping may occur with extended lengths of unsupported walls
- avoid sharp transitions, fillet at points where walls join
- even wall thickness is recommended on all walls (both vertical and horizontal)

Designing for Polymer Powder Bed Fusion
- laser sintering, selective laser sintering — an additive process in which thermal energy selectively fuses regions of a powder bed

Designing for Vat Photopolymerization (VP)
- when orienting a part for SLA (particularly bottom-up) biggest concern is the vertical cross-sectional area
- forces involved with a print sticking to the bottom of the tank are directly proportional to the 2D cross-sectional area of the print
  - because of this, part with large cross-sectional areas is often best printed at an angle to the plate
  - minimizing cross-sectional area along the Z-axis is the best way to orient parts for SLA prints
- support material: SLA does require support material for overhanging features because the uncured resin is not viscous enough to support features on its own; support material must be removed post-processing
- isotropy in SLA: one of few processes where parts are relatively isotropic, because the layers chemically bond to one another as they print, resulting in near identical physical properties in the X, Y, and Z-directions
  - whether part is printed horizontally, vertically, or at an angle to the build plate, material properties of the part won’t be noticeably different in any particular direction
- hollowing parts and resin removal in SLA: drainage holes must be added to allow the uncured resin to be removed from the part; if left inside, uncured resin can create a pressure difference within the hollow chamber and can cause what is known as “cupping” —> such failures will cause cracks/holes to propagate throughout the part and eventually cause complete failure or explosion

— **From 3D Hubs** —

Supports in FDM

- FDM extrudes melted filament onto a build surface along a predetermined path; as material extruded, it cools, forming a solid surface providing the foundation for the next layer of material to be built upon
- each layer printed as a set of heated filament threads which adhere to the threads below and around it; when a feature is printed with an overhang beyond 45 degrees it can sag and requires support material beneath to hold it up
- *bridging vs. support*: hot material can be stretched short distances between two points in a method known as bridging, which allows material to be printed without support and with minimal sag; if a bridge is over 5 mm long, generally, support is required to give an accurate surface finish
- the YHT's of FDM support
  - arms of Y printed easily because extend at less than 45 degrees
  - H printed if bridge is 5mm or less
  - T always requires supports for arms of the letter

Two Types of FDM Support

1. flat accordion, or lattice, is most common and best suited for most FDM
2. tree-like preferred by some printers, has less contact with print surface and can result in better surface finish post processing

Dissolvable Support

- if have two print heads, support material can be printed with a dissolvable material that does not tear away from the part but instead dissolves away in a chemical solution that does not affect the main material of the printed model
- result in better surface finish where support in contact with the main material but can be expensive and time consuming
- example: PVA

SLA & DLP Support Structures

Stereolithography (SLA) and Digital Light Processing (DLP) create 3D printed objects from a liquid photopolymer resin by using a light source to solidify the liquid material

- support needed to make sure prints adhere to the print platform and no float around in the vat
- support structures from these printers look like thin ribs, with only small tips actually touching the model to save material and printing time
- removing supports: first isopropyl alcohol (IPA) used to wash liquid resin off completed parts, then support structures broken off surface of model or removed with pliers, then spots where support in contact with object then sanded to remove any remaining marks
- design considerations for supports in SLA & DLP
  - by reorienting, the amount of support and therefore cost of print can be drastically reduced
  - if aesthetic appearance of surface on a component is paramount, orientating the part so that there is little to no support in contact with that area can also be an option

Material Jetting Support Structures

Material Jetting (Stratsys PolyJet for example) technologies are similar to inkjet printing but instead jet layers of liquid photopolymer onto a build tray and cure them instantly using UV light

- require use of support material in all cases where there are overhanging parts, regardless of angle

- supports either water soluble or removed post-processing using pliers, water-jetting, ultrasonic bath, and sandblasting
- unlike FDM, supports not detrimental to looks, surface quality or technical properties of the print

SLS Support Structures

Selective Laser Sintering (SLS) fuses powdered material in a chamber using a laser
- no need for supports since the powder acts as a support when the object is built up layer by layer —> gives a lot of design freedom but also generally increases cost and time to print a part

Metal Printing Support Structures

Metal printing technologies use support structures to keep models fixed to a base plate during building process in all cases, but overhangs with angle greater than 35 degrees can be built without support
- usage of supports does not impact quality of print in any way with proper post processing, all marks can be removed

Rules of Thumb

Printing technology	Support required
FDM	Dependent on model geometry

Printing technology	Support required
SLA & DLP	Always required
Material Jetting	Always, but soluble
SLS	Never
Binder Jetting	Never
Metal printing	Always