Design for Additive Manufacturing

Whitepaper

Dr. TR Kannan

Contents

Introduction

Additive Manufacturing has become a buzz word in today's manufacturing world. It has gone through tremendous improvements over the past few decades and has matured from simple prototyping to actual manufacturing and tooling. Various methods have emerged like Fused Deposition Modeling (FDM), Stereolithography (SLA), PolyJet (3DP), Selective Laser Sintering (SLS), Direct Metal Laser Sintering (DMLS), etc. All these methods manufacture the part through addition of materials layer by layer and hence this technology is referred as additive manufacturing. There are many other synonymous terms used like rapid prototyping, rapid manufacturing, 3D printing, etc.

Applications of additive manufacturing cover a vast variety of industries like automotive, consumer goods, medical devices, aerospace, defense, etc. Though in principle, any component can be manufactured by either subtractive manufacturing or additive manufacturing techniques, various design features pose completely different challenges in both methods. With a growing number of parts manufactured directly by additive manufacturing techniques, it is important to lay down design principles suitable for such manufacturing processes and to ensure parts are designed for additive manufacturing. There are several factors that are to be considered at the design stage for effective manufacturing of parts using additive manufacturing. Few such design issues in additive manufacturing are discussed in this paper.

Maximum Part Size

Parts can be either manufactured with in-house additive manufacturing machines or can be outsourced. The maximum size of parts is generally constrained by the additive machines available for manufacturing. If the part is of bigger size than the maximum machine capacity, then the following methods can be adopted to tackle this. (a) If the end use is only to create a prototype, then the part can be scaled such that the maximum dimensions fit the machine. But due to scaling, some finer details could be lost depending on the scaling factor and the feature dimensions. This could require some editing or clean up in the scaled 3D model. (b) The part can be redesigned to fit the machine if the mismatch is small in dimension or the part can be redesigned in to multi-piece assembly. (c) If the model is too large then the model could also be spilt into two or more pieces to be glued or welded later. This could also require some modifications or addition of few features to increase the bond strength.

Faces Requiring Support

As additive manufacturing methods build the part layer by layer, some designs might require additional support due to the nature of additive manufacturing. Features such as negative drafts, overhangs and undercut features as shown in figure 1 require support in FDM/SLA/3DP. Such features can be avoided wherever possible, as they require supports which increases the part creation time and in turn cost of manufacturing. Parts requiring supports also might require secondary processes like removal of support, cleaning and sanding of the part at support joints.

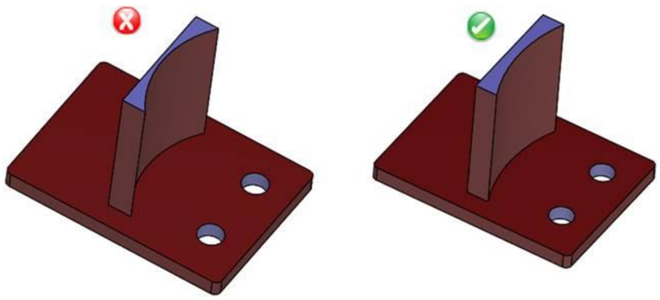

Figure 1. Faces requiring support

Minimum Wall Thickness and Rigidity

The minimum wall thickness of a part is generally constrained by the additive manufacturing method, machine resolution, etc. Very thin walls could make the part very fragile and hence a minimum wall thickness has to be maintained to provide sufficient strength and rigidity to the part. Apart from this, the part has to be strong enough to withstand the stress caused while removing the support material. This necessitates a minimum wall thickness to be maintained for faces requiring supports such as negative drafts, overhangs and undercuts.

Rigidity of a part can also be increased by adding ribs. Ribs are added as protrusions, which can stiffen and strengthen the part and in turn facilitate having thinner walls. However, tall ribs or long ribs can also create some problems. Ribs need to be designed in correct proportions of length, height and thickness to provide the required strength. Rib thickness should be large enough to be manufactured by additive manufacturing. If ribs are too long or too tall, supporting ribs may be required. It is better to use a number of smaller ribs instead of one large rib. For large surfaces, it is advisable to add rib networks. For additive manufacturing, it is generally recommended to increase the wall thickness rather than adding ribs to avoid thin walled structures. But thick walled structures increase the weight of the part as well as cost of additive manufacturing, hence it is generally a tradeoff between design requirement and cost.

Generally part designs may have bosses, which serve as points for attachment and assembly. The most common boss designs consist of cylindrical projections with or without holes. Holes in bosses are designed to receive screws, threaded inserts, or other types of fastening hardware. Under service conditions, bosses are often subjected to stresses not encountered in other sections of a component. So, bosses are generally designed with a draft to increase the strength at the bottom. In injection molding, drafts in bosses also facilitate easy removal of parts from core and cavity of the mold. In additive manufacturing, drafts in boss outer surface serve as boss stiffeners. A fillet of certain radius is also provided at the base of boss to reduce stress. The Undercuts Negative Draft Overhang Filling Direction radius at the base of boss should be larger than a certain minimum value depending on the additive manufacturing machine. Tall and slender bosses should be avoided. While designing the bosses, correct proportions of height, outer radius, hole radius and hole depth should be provided to achieve the required strength.

Minimum Feature Size and Manufacturing Quality

The minimum feature size of various features like holes (blind or through), pockets (depression texts or symbols, cutouts), islands (protrusion texts or symbols, bosses, pins) in a part is generally constrained by the additive manufacturing method, machine resolution, wall thickness, whether the feature is in vertical or horizontal wall, etc. The minimum feature size is constrained by the bead width in FDM and laser in SLS. Machine manufacturers recommend that the minimum feature size in any section i.e. XY plane should be greater than or equal to four times the resolution, whereas the minimum feature size in Z direction should be greater than or equal to the resolution. So, features should be designed to have any dimension greater than the minimum feature size to get more accurate parts from additive manufacturing. Also all sharp corners in the XY section plane should be filleted or chamfered to accommodate the natural radius inherent to the manufacturing process and to reduce the stresses. Fillet radii should be greater than the minimum natural radius, which is generally four times the resolution. Similarly any knife edges with zero thickness at the edge will get manufactured with some thickness hence it is advisable to flatten such knife edges to a minimum thickness.

There are specific design considerations to be taken into account while designing various features.

- Quality of holes depends on the wall thickness and the diameter of the hole. Minimum diameter that can be used increases with the wall thickness. In other words, ratio of hole diameter to hole depth should be higher than a minimum specified value.
- The quality of pockets or cutouts depends on the wall thickness and the feature dimensions. Quality is generally better in thin walls for dimensions, which are perpendicular to the wall thickness. Minimum value of such dimensions increase with the wall thickness. In other words, ratio of min dimension perpendicular to wall thickness to pocket depth or wall thickness should be higher than a minimum specified value. While designing such features, a minimum inter-feature distance and a minimum feature to edge distance have to be always maintained.
- Texts are similar to pockets and islands. Texts should be larger than a minimum font size and the manufacturing quality is generally better when they are placed in vertical walls than in horizontal walls.
- Bosses or pins should be larger than a minimum diameter depending on the additive manufacturing machine.
- Bosses or pins should be larger than a minimum diameter depending on the machine.

Generally 3D models are converted to STL file format before manufacturing. STL files store the surface geometry of a 3D model by tessellating into triangulated surfaces, which introduces an approximation error for curved surfaces. Such errors are less prominent with finer tessellation.

Hence it should be ensured that the tessellation quality is set as per the allowable deviation as it plays a major role in quality of parts manufactured.

Geometric DFX

For additive manufacturing, though all the design rules might appear simple, verifying all these rules manually in 3D models is very difficult and time consuming. Manual verification can be error prone as there are chances of missing few checks. An automated system can largely help in speeding up this process and bringing a standard way of verifying parts to avoid manufacturability issues.

Geometric DFX is a design for manufacturing product that takes 3D parts as input and automatically analyses all the manufacturability issues based on certain predefined rules. DFX highlights the rule failures directly in the 3D model and also generates xml and xl reports. The rules are configurable based on the machine and the type of additive manufacturing. A variety of CAD formats can be handled like Pro/E, NX, CATIA, SolidWorks, Inventor, STEP, IGES, Parasolid, ACIS, etc. DFX checks for additive manufacturing reduces multiple design iterations and helps to ensure right designs to be submitted for manufacturing. DFX reduces design to manufacturing lead time, reduces multiple trials and in turn cost of manufacturing. Apart from additive manufacturing module, there are many other modules in DFX like milling, turning, sheet metal, injection molding, casting and assembly.

DFX Rules for Additive Manufacturing

General design rules that should be ensured before sending a part for additive manufacturing are given below.

1. **Maximum part size check** – Compares part size with allowable maximum part size and shows a failure if the part is larger.

2. Minimum wall thickness check - Compares wall thickness of the part and highlights the regions where thickness is lesser than the allowable minimum thickness. This rule also helps to check for minimum distance between generic pockets (Hole/cutout/pocket) and minimum distance from edge to generic pockets.

3. Faces requiring support rule (negative draft/ overhang/ undercut recognition) – Recognizes faces requiring support and highlights those faces.

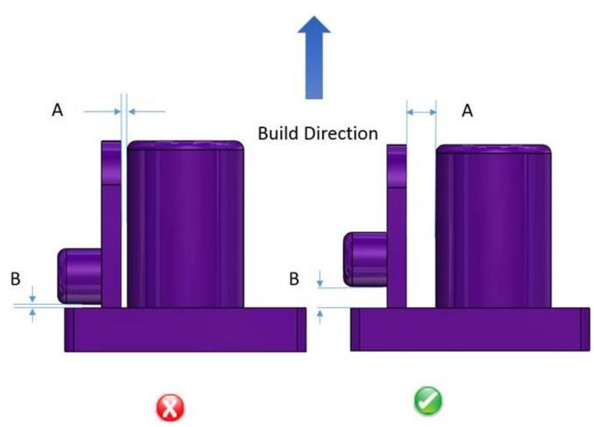

Fig 2: Faces Requiring Support

4. Minimum thickness of faces requiring support rule (negative draft/ overhang/ undercut recognition) – Compares the thickness of faces requiring support with that of the allowable minimum thickness and highlights the faces that fail.

5. Minimum feature size (Pocket/Island/Text) – Compares the feature sizes with that of the allowable minimum feature size and highlights the features that fail.

6. Recommended rib parameters – Recognizes ribs and compares the ratios of (a) rib-base thickness to nominal wall thickness and (b) rib height to nominal wall thickness with that of the maximum allowable ratio.

7. Rib reinforcement check - Compares the ratios of (a) rib area to nominal wall thickness and (b) rib width to nominal wall thickness with that of the allowable maximum ratio and highlights the features that fail.

8. Boss ID to OD ratio - Recognizes bosses and compares the ratio of inner diameter to outer diameter with that of the allowable minimum ratio and highlights the features that fail.

9. Boss height to OD ratio - Compares the ratio of boss height to outer diameter with that of the maximum allowable ratio and highlights the features that fail.

10. Minimum hole diameter to thickness or depth ratio – Recognizes holes and compares actual diameter to thickness (depth) ratio with that of the allowable minimum ratio and highlights the features that fail.

11. Knife edge - Recognizes knife edges and highlights them.

12. Recommended corner radius – Recognizes fillets and compares actual diameter with that of the allowable minimum radius and highlights the features that fail. Also recognizes sharp edges and highlights them.

13. XYZ slice dimensions – Checks whether all XY dimensions are exact multiples of 4 times resolution and Z dimensions are exact multiples of resolution and highlights the regions that fail.

Conclusion

To stay competitive in the era of globalization, it is very important to look for all means to reduce manufacturing lead time, cost and time to market. It is generally agreed that design change requests that come at various stages affect cost and lead time. Changes made in the early design stage are less costly and hence concurrent engineering concepts got in place where experts from different teams analyze to reduce known issues in the design. Even this process requires multiple iterations and for additive manufacturing processes, the availability of expertise is rare. By incorporating Geometric DFX, organizations can reap considerable benefits to achieve first time right designs for additive manufacturing which in turn will reduce manufacturing cost and time.

References

1. http://www.shapeways.com/tutorials/design_rules_for_3d_printing

2. http://www.solidconcepts.com/resources/dg/selective-laser-sintering-sls-designguidelines/

3. www.3dsystems.com

4. www.eos.info/en

5. www.stratasys.com/

6. www.makerbot.com/

About the Author

Dr. Kannan has over 20 years of R&D experience in CAD/CAM, engineering software development, and manufacturing automation. He has a Ph.D. in computer integrated manufacturing and process planning. He has published multiple research papers in renowned international journals and conferences in related areas. His area of expertise includes product management and R&D for next generation CAD/CAM software products. He is currently responsible for research and development of various products like DFMPro, Feature Recognition and Nestlib.

About HCL Technologies

HCL Technologies is a leading services company that covers the entire gamut of technology solutions and services including infrastructure management, application development, BPO and engineering and R&D services. Over the years, HCL Technologies has demonstrated remarkable growth inspite of economic downturn and is emerging as one of only eight 21st century listed technology companies in the world to cross $1bn in net profit, $5bn in revenue and $15bn in market capitalization.

As a $6 billion global company, HCL Technologies brings IT and engineering services expertise under one roof to solve complex business problems for its clients. Leveraging extensive global off-shore infrastructure and a network of offices in 31 countries, HCL provides holistic, multi-service delivery in industries such as financial services, manufacturing, consumer services, public services and healthcare.

