Designing Parts for Direct Digital Manufacturing

Jim Comb
Stratasys Inc.
Minneapolis, Minn.

Direct digital manufacturing gives the freedom to design without all the rules used in traditional methods.

Traditional manufacturing methods, like machining and injection molding have many rules, restrictions, and limitations. However, the same rules don’t apply when using direct digital manufacturing (DDM); designers are free to concentrate on the best design and do not need to be concerned with manufacturability.

An often-touted fundamental advantage of DDM is “freedom of design,” but how far does this freedom reach? What can a design engineer do with it, and what does he or she need to know?

Essentially, design for manufacturability (DFM) rules are discarded—design is no longer constrained by the limitations of conventional manufacturing processes. This frees the product development team to design the “perfect” part for the application. Following are just a few examples of typical injection molding rules that don’t apply to DDM.

Although DDM can be implemented without changing existing design principles, a little education will go a long way in getting the maximum value from the process. To get started on the road to DDM success, consider these helpful design tips.

**Design Methodology**

*Forget DFM:* DDM rewrites product design doctrine, so the most critical step is to not think about designing for traditional manufacturing processes. Complex design can be fast, inexpensive, and practical when made with additive manufacturing technologies.

*Focus on function:* The design process begins with a focus on function; engineers should design the parts to achieve the best performance possible. You can make the parts as complex, intricate, and detailed as they need to be. For industrial designers, the converse holds true: focus on form and let fit and function follow.

Because DDM uses an additive process to manufacture the parts, cost and time are no longer a function of complexity as with conventional manufacturing methods. For the same reason, design features are rarely impossible to reproduce. While there are a few process constraints, which depend upon the particular brand of additive manufacturing equipment used, they can be addressed at the production stage.

*Iterate:* Increase the frequency of design iterations and plan to continue design refinement later in the product development cycle. Continue to hone the design up to the day that the product is launched.

Although all additive manufacturing processes are suitable for rapid prototyping, not all are suitable for DDM. Some processes can make beautiful models, but not durable parts for end use. The Stratasys FDM process creates durable parts from various formulations of engineering thermoplastics like ABS (acrylonitrile butadiene styrene), polycarbonate, sulfone, and blends.

Assuming you employ a rapid prototyping technology that is also suitable for DDM, you can use the same equipment for both functions making prototyping and production processes the same.

Each is completed with little effort, minimal cost, and no delay. The only difference between the final prototype and a production part is its intended use. This low-risk, rapid-turn cycle allows the design team to be creative and to push the envelope. There is no penalty for design revisions late in the product development cycle.

*Refine the design:* DDM can be performed using various additive technologies, so it’s important to have a good understanding of the

---

<table>
<thead>
<tr>
<th>Injection molding</th>
<th>Direct digital manufacturing (DDM)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Draft angles must be included in the tool or parts won’t eject properly.</td>
<td>No need for a draft on the part. You can even have “negative (back) draft” on a DDM part.</td>
</tr>
<tr>
<td>Uniform wall thickness is required to minimize warpage and sink marks.</td>
<td>Wall thickness can be varied throughout a DDM part and thick-wall to thin-wall transitions cause no problems.</td>
</tr>
<tr>
<td>Radiused (rounded) corners are required to reduce stress concentrations and improve plastic flow during molding. Radii are required on most inside and outside corners of a molded part.</td>
<td>With DDM, you can have sharp corners wherever desirable.</td>
</tr>
<tr>
<td>Since each material has a specific shrink rate, it may not be feasible to change materials once a tool has been made.</td>
<td>With DDM, you can change your material with each new build.</td>
</tr>
</tbody>
</table>
one you will use. Each technology has different specifications in areas such as minimum wall thickness, tolerance, surface finish, and material properties. Refine the product’s design to accommodate these characteristics. If you cannot produce part qualities using the additive manufacturing process at your disposal, either outsource production or purchase a system that can produce the product. Fortunately, purchasing an additive manufacturing system for DDM often comes with a fast ROI, so it may be easy to justify the purchase.

Question tradition: Do not let past practices, old traditions or previous decisions dictate design options, and process selections. Question everything. For example, a part previously made of sheet metal may be an ideal candidate for plastic because the rationale for the original decision may no longer hold true. Sheet metal may have been selected as a practical, but not preferred, manufacturing process because of low production volumes and high cost for injection molds. With DDM, a sheet metal enclosure can be converted to a sophisticated, stylized plastic part since there is no tooling to amortize over a small production run.

Design Techniques

Make it feature rich: With traditional manufacturing methods, each feature adds cost because that feature must be machined into the part, mold, or die. This is not true with DDM. Consequently, never simplify designs for any reason other than product performance or aesthetic value.

Rethink wall thickness: Many manufacturing methods have a narrow range of recommended wall thicknesses. For example, the sweet spot for injection molding is 0.40 to 0.80 in. When designing parts for DDM, the only consideration is to stay above the minimum wall thickness needed for the part to perform as specified. So a part can have walls as thick or thin as desired, rather than being dependent on manufacturing restrictions. There is also no need to maintain a consistent wall thickness. To minimize weight, consider making the walls hollow. In the FDM process this construction style is called sparse fill. A lattice structure is skinned with bounding surfaces to yield mechanical strength while decreasing material volume. Although the volume reduction can range greatly, a typical application might yield a 60% volume reduction. Leaving features hollow has the added benefit of reducing material cost and part construction time.

Consolidate or segment: Part consolidation is a big advantage of DDM that should be considered at all times. Rather than producing a multi-piece subassembly, the entire unit may be consolidated into a single component. By consolidating parts, the assembly process is eliminated and inventory management is simplified,

As part of a product redesign, Bell & Howell engineers reduced the number of components in this assembly from 26 to 13 and eliminated the need for screws. Consolidating components is one of the advantages of direct digital manufacturing.
Part consolidation may also be used to overcome an overly tight tolerance specification. For example, a tight-tolerance interface can be avoided by simply consolidating mating parts.

The converse also holds true. A single piece can be segmented into several components without a significant increase in cost. When using traditional processes, dissecting a component may not be justifiable because doing so may require more molds, translating to higher expenses. The ability to create a subassembly, rather than a single piece, can be an asset when addressing product design considerations like serviceability and replacement cost.

Fill the envelope: Use every nook and cranny of your product’s available space. Twist, turn, and contort to maximize the use of space and minimize the size of the product. Since machining and molding limitations are removed, think organically and let the design flow.

Forget the details: When the design is complete there is no need to invest time to adapt it to process-specific requirements like those commonly used for machining or injection molding. For example, designers do not need to spend time defining parting lines, adding draft angles, and determining how to incorporate them without changing form, fit, and function. There is no need to resolve undesirable sink marks, ejector marks, or knit lines. These types of process constraints no longer exist.

Forget the details and unlearn the past. DDM presents a radical departure in design practices, techniques, and methodologies. The shift is so fundamental that the impact will not be fully recognized for years to come. Industry has only started to appreciate all that DDM can offer. So, adopt the basic techniques, implement the new methodology, and continue to advance the practice of direct digital manufacturing.

Above all else, be creative. Stretch the skill set, push the design envelope, and challenge conventional wisdom. Never settle for a direct process substitution because much of DDM’s value will be lost. Always allow the time necessary to design a part, subassembly, or product in order to capitalize on the unique capabilities of DDM.

Finally, never stop redesigning. Equally powerful to the freedom of design is direct digital manufacturing’s freedom to redesign. There is no commitment to tooling and little investment of manpower, so a design is never frozen. It is perpetually fluid. Capitalize on this by continually refining designs to satisfy the customer, maximize manufacturing efficiencies, and minimize production costs.

For more information: Jim Comb is a systems engineer at Stratasys Inc.; Eden Prairie, Minn.; 800/937-3010; jjcomb@stratsys.com; www.stratasys.com