Design: A Processing View — Part II

Cris Belmont

General Electric Company
1 Plastics Avenue
Pittsfield, MA 01201

Floyd Hester

General Electric Company
17000 Gale Ave.
City of Industry, CA 91745

The following dialogue between designer and molder was taped by Plastics Design Forum at the SPI Structural Foam Division Conference in Norfolk, VA, May, 1979. Thanks are due Ed Galli of PDF for editing this taped version:

Designer:

Looking at the problems that face the designer, the end user, the toolmaker, the molder and the finisher, we find that no one group can remain an independent island in the overall program. It is necessary for every group to have a complete understanding of each other's problems and methods, so that the optimum design is reached at the minimum cost, minimizing delays, attaining the quality demanded today and meeting the production schedule.

Molder:

We often find that an engineering group will come up with a new program – turn it over to the design group, which in turn goes to purchasing, which then selects a molder and perhaps even a toolmaker. This isn't the way to go about it. There's no backflow of information. When the design is still in soft copy it's not difficult to get changes made in the print, based on information from the processor. Once prints are made and turned over to purchasing, however, it can take several months to get any kind of realistic design changes made.

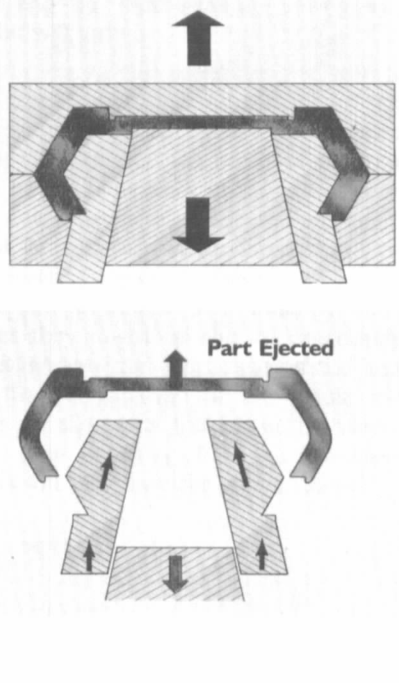
Designer:

So if the processer is included in the initial stages of the project, he can save time and money, and help in designing a better SF part. That's an important point because even relatively simple designs have hidden problems – in flow lengths, poor surface, warpage, overpacking, control of tolerances, sink, molding cycles, and so forth. The same theme, the importance of the designer understanding the processing parameters, and working with the molder and toolmaker at an early stage, it becomes even more important when we get to the more complicated designs.

To start us off, here's a good SF application, a keyboard housing. Industrial designers gave us the outside shape, with the curving back that bows back in at the bottom. We've got a problem here... we're not allowed to have a witness line... we can't make this housing in two pieces, split at the waist. As a result we've ended up with a pretty thick section. I know it's possible to keep that wall at a constant thickness, but that would involve an undercut. Our volume isn't large enough to justify a lot of complicated hydraulics – motion and cams and our molders would prefer a simpler one. Any comment on how we can avoid this problem?

Molder:

There is one fairly simple way – most people in the industry call it "advancing cores." What it amounts to, as the sketch shows, is that the cores are mounted on an angle and fastened to the ejector plate. It's a standard technique, used by a lot of molders, but unfortunately a lot of people don't know about it. It's a very simple concept, with no hydraulics required. It is a mechanical function, tied in with ejection; the ejector plate comes forward, the cores collapse and allow for removal of the undercut.



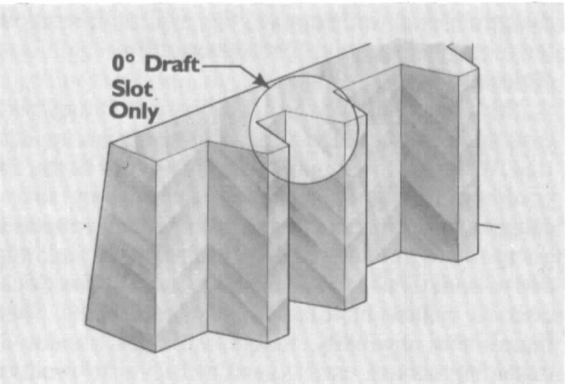
This approach can be used quite successfully even in areas of negative draft. For instance, where heavy bosses that can add to the molding cycle are present, a portion of the boss can be eliminated by using advancing cores. Sometimes designers say, "Well, we'll make the part a little heavier because otherwise the tooling cost will go up several thousand dollars, but in any decent volume job I think the cost would be more than justified." And you'll be making a better quality part, and the cycles will definitely be much shorter.

Designer:

You mentioned zero degree draft. A lot of processors are molding card cages right into their housing bases, saving a lot of money. Since the cards have to be rectangular, the card slots need zero draft. Is that how you mold them, with fancy cores and a lot of complicated cam action?

Molder:

Not really. Normally these parts involve only a small draft area. In SF it is quite feasible to pull zero degree draft. In injection molding it can be a problem; because of the increased pressure, the plastic material really wants to hang onto the cores, and torn parts result. But if some weight reduction is accomplished in SF, zero degree draft isn't very difficult. You might want to put a bit of lead at the top, to lead your cards in, and that would give a little bit of draft. Then we can pull zero degree draft for the rest of it.



Designer:

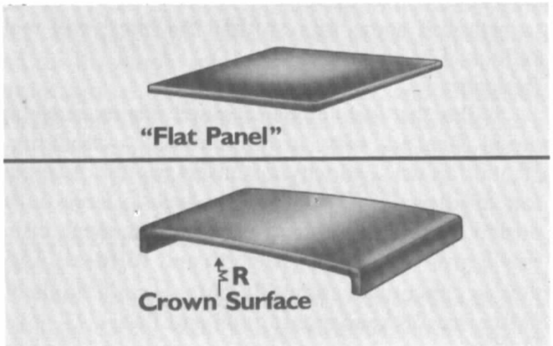
So in small areas like card slots we don't need any tool action? That's really low cost. That adds a lot more design freedom to SF.

Molder:

Perhaps you can reciprocate and help me out with a problem. Here's an application I'd like to design in SF. This desk unit is in sheet metal, but when I tried to tell the manufacturer that it would look great in SF, he said he wasn't even satisfied with some of the sheet metal cases. They're concave, they look weak... how can we make a plastic part look better than a steel one? And of course steel is much stronger. Have any suggestions on how I can approach that market?

Designer:

That's a tough discussion; I've been there. What you can do – industrial designers use this many times, it's an optical illusion – in order to make a panel appear flat when the light is reflecting off it, *don't* make it flat. They'll look caved in, and if they have any concavity at all, they'll really look bad. The trick is to slightly crown the surface using a very large radius – perhaps 200 to 300 inches. Do it in two different directions, and if you blend them properly, by crowning the surface it will give the appearance of being flat.



Also, by crowning the surface, oilcanning will be eliminated – that's a problem in metal panels. It also improves the strength. If you're looking at the top of a printer or a long CRT unit, it should have a straight styling line. Arch that slightly, and lean on it – it's going to have a lot more strength and stiffness because you're changing the centroid when the load goes into compression; But there's a problem because I'm sure it's going to complicate the tooling, having to produce the curved surfaces.

Molder:

We will have to duplicate the cavity. We're probably talking about six parts, and at about two thousand dollars per cavity, it shouldn't be too bad.

Designer:

Another design advantage offered by SF, over injection molding – is the ease in which metal inserts can be molded in. They can add additional function to the part. This is a good example – a paper drawer that was molded with an insert steel channel for the drawer slides, to give a good metal surface for the rollers and extenders to ride on. A good example of where the molded-in insert creates an additional function at low cost.

Molder:

One important consideration if you're molding inserts like this: in machines with vertical-acting clamp, gravity will normally hold the impact in place. But in a horizontal machine – especially if there's any vibration – you have to be very careful and either have some positive way of locking in the inserts or some sort of monitoring system so that the mold doesn't close on an insert that's fallen out. That can result in quite serious mold damage.

Another thing to consider is how the insert locked into the part. If the plastic doesn't shrink around it, there will have to be some sort of method to establish some bite so that the insert will really be locked in place.

Designer:

I like the idea of using metal inserts more and more. Steel's a very strong material – modulus of 30 million psi – so why not insert mold it into parts for additional stiffness. If you have a large unsupported tabletop, just put a piece of steel inside, that'll give you a lot of strength and stiffness. And if you're concerned about flatness, I can't think of a better way to keep a part good and flat.

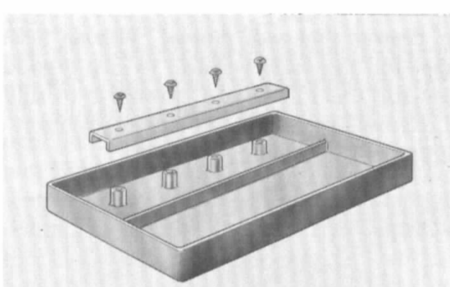
Molder:

I'm afraid this design will create difficulties. There's only a small amount of plastic over the top of that insert. If the total part is exposed to any kind of forces at all, that metal insert will shear right through the plastic.

There are other things to consider, too. Because the shrinkage factor is so different between steel and plastic, you'll find the plastic on the bottom side is going to shrink much more, and a warped part will result. You put the steel in the top, and it will keep the top from shrinking, but the bottom will shrink normally. Ideally the insert should be located right in the middle, to equalize shrinkage.

Designer:

That doesn't do me a bit of good. If I put the steel right at the centroid of the part, the contribution to the moment of inertia is almost zero – it's right at the point where it doesn't give me any stiffness.



Molder:

If it can't be done any other way, perhaps you'd consider fastening it on the outside. It could be attached ultrasonically to studs that are left on the bosses, or with some sort of metal fastener. If there's no reasonable way to put a stiffener *in* the part, consider adding it to the outside.

Designer:

Well, if metal inserts can create more problems than they can solve, there are always ribs. Here's a cover, an appearance part, that needs strength because things will probably be stacked on it, or someone will lean on it. What do you think about the rib design?

Molder:

The rib design I like best is no ribs. The approach I advocate is, if there is doubt about whether the part is going to be stiff enough, leave the ribs out. Ribs can create sink, and all kinds of flow problems – let's mold some parts without ribs and see what we've got. Ribs can always be machined into the tool after the part has been sampled, because there's always some touching up to be done later. It won't cost any more; ribs are normally right out in the open and it's simply a matter of going through with a taper cutter and polishing them out.

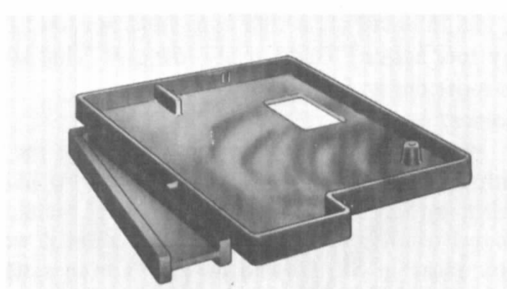
Designer:

I guess in retrospect we designers are kind of guilty, we don't always go through a lot of calculations – we tend to be conservative and just throw in some ribs because it looks like they ought to be there. I actually don't really know what the deflection load on this part is going to be.

Let's turn to a more complicated part. This is a base that's a good concept for SF. I've designed-in a lot of function – bosses, standoffs, ribs, and over in the corner is a top support. this will replace about ten metal parts... good cost reduction.

Molder:

You may actually be wasting some money with that design, especially if it's a low volume application. With that top support sticking way up in the air, you're going to require a very large cavity to get deep enough to mold that part. The extra metal is going to cost money, and it's expensive to sink that cavity and then core out the center section. Could even be a little bit of a problem in molding. Then there's the possibility that I can't get a piece of metal big enough, so I'll need a forging to produce the mold, which could add to lead time besides cost.



But since that section just has to support an end thrust pressure, why not mold it as a separate part off to the side. That would produce a considerable savings in most cost, because all I'd need would be a 4-inch block of aluminum instead of a 14-inch block. The part could be solvent bonded or mechanically fastened after molding, probably at the machine, so it wouldn't cost any more to mold it.

Designer:

If I'm paying an extra $15,000 just to mold that as one part, it's obviously going to be much more cost effective by doing a two-piece design.

Molder:

If you have a lot of complexity and a lot of high volume parts it could be a different story, and the choice of polymer could make a difference too. Adding that part on may require a bit of assembly time, but if your volumes are relatively low, you'll enjoy a cost savings this way.

Designer:

That brings up another project. Here's a cabinet that requires deep cavity tooling, a lot of complex work... we're not a big company, we don't have a lot of money for tooling. I guess I should design this as an assembly – a base, a top, two doors and a couple of sides, and as you just pointed out, this will save me 35–40 percent in tooling costs.

Molder:

Each job should be studied individually. This particular one would involve about six molds to make six different parts; that's a tool cost in the range of a hundred thousand dollars, and depending on whether a single or multiple nozzle is used, anywhere from three to six molding cycles. Then, with a part of this complex, you would have to include assembly slots, and it couldn't be assembled at the machine, so assembly labor would also be included. So if we look at doing it in multiple pieces, I think the total part cost, excluding material, would likely be about thirty dollars.

If you look at it as one piece, it's interesting. Because there would be only one ejector system for the large part, instead of five, and only one core and one cavity, there isn't much of a difference in tool cost. Depending on whether it's run by itself or both parts are run together, we have only one or two molding cycles... and there's no assembly cost. So for this particular unit I think it would be foolish to go with multiple molds.

Designer:

I guess the difference is that in this part that deep cavity does a lot for us; it forms all four sides at one time, where in the previous part it just delivered one post.

Molder:

While we're looking at enclosures, here's a problem that I've encountered. I've been talking to a company that uses a lot of enclosures, but they must have 25 different panels, in many sizes, and often there are only 200 or 300 units on an order. I tell them that SF is cheaper, but they say they can't afford the tooling costs for so many different panels. How can I get this business with that kind of problem?

Designer:

This is a common problem especially in the business machine industry. These people grew up in sheet metal. They make any size they need, one at a time, it doesn't matter how big. The way to attack this is to go to the concept of standardized design; combine all those parts into a few sizes and get the volume up to ten or twenty thousand a year. Admittedly they're not going to make every product in exactly one size, but if you design the units properly, and use them in various combinations, many times you can completely eliminate all of the metal panels and do the whole job in SF. You can get the styling advantages of SF with witness lines, and crown the surface for better stiffness. Then, by using inserts in the mold, you can provide for the whole variety of cut-outs for control panels, cooling louvers, windows. Of course, I don't know what it would cost to go to mold inserts.

Molder:

I can see there are a lot of possibilities there. I would probably want to go with steel tools because of the wear involved with the inserts, and I'd have to carefully figure where to put water lines and ejector pins. With the low volumes we'd need a way of changing those inserts quickly, with some sort of locking device outside the mold on the side, or access through windows. If I'm only running a few hundred parts I certainly wouldn't want to pull the mold every time I had to change the inserts. Basically what we've got is an adjustable mold... almost like an erector set.

Designer:

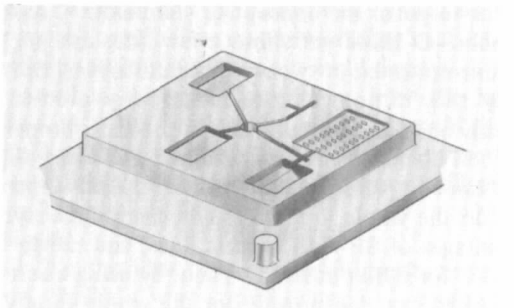
Okay, there's lot of good work to be done by SF in big parts like cabinets But what about the small parts? We had a management review last week. Engineering management had a couple questions: Why are you designing small parts in SF? SF uses big machines. Shouldn't the small parts be injection molded? Why tie up a machine with a capacity of 30, 50, even 100 pounds to make those little widgets?

I have some answers ready. One is appearance. If you injection mold or vacuum form a keyboard panel, or the bezel for a CRT, and not paint it, there will be a finish problem trying to match up with the painted SF cabinet. If you mold all the parts in SF, they'll have the same feel, the same paint, the same texture. And they'll probably all be done at the same molder, who can then assemble them.

Then there are little parts with heavy sections, like some housings and blower controls. Heavy sections really utilize SF to its best advantage. You can get much better stiffness with SF, too.

Molder:

For a lot of these small parts it would be helpful if they could be nested in a family mold. The mold has to be designed with gating set up to progressively mold the most difficult part first. Even with multiple nozzle machines, a small part can cause big problems. But when quite a few parts are molded together in a common mold base, molding is much easier.



Designer:

Talking about multiple part molding, here's a question for you. Here's a job with two parts: housing, and its access door. They're molded together in a two-cavity tool, on a multiple-nozzle machine. And sometimes they vary in size. How can that happen when they're made in the same tool?

Molder:

Part size can be related to density; it's possible that the parts being received vary in weight. The larger part is the more difficult one to mold, so the machine is set up to handle it, and the small one just comes along automatically.

Designer:

So the denser part is the bigger one?

Molder:

Not necessarily. It's an unusual curve; it isn't a straight line. A part that comes out undersize is definitely going to be heavy in SF. In injection molding it's just the reverse. The more plastic packed into a part, the larger the part is going to be. A poorly packed part in injection molding will be undersize, and that is probably the same range in cavity pressure that would be seen with a very heavy foamed part. What happens is that it shrinks too much. By decreasing the weight of the SF part, you can make it larger. There is a range where you can come up with an ideal density. An oversize part can be either too heavy or too light, but an undersized SF part will always be too heavy.

Designer:

Since the density of the part, or the weight of the part, is related to the dimensions or tolerances of the part, what do I specify on the drawing if these two parts are related? If I calculate the weight of the part, and specify it on the drawing, am I guaranteed of getting good parts?

Molder:

You really can't do that because different configurations will have different behaviors. Part density can be added to the drawing, but only after parts are molded and the optimum density range is developed. You have to work together with the molder to settle on a realistic density range and realistic dimensional tolerances. Once that is accomplished, it can become a QC criterion. The weight of a part, once it has been established this way, can be a good QC tool that is predictable.

Designer:

Good. Here's a very complex part that we're having a lot of problems with. It has a lot of critical dimensions. Very tight tolerance controls. Unfortunately, the dimensions that are critical are very hard to measure, requiring a lot of stack-up and transfer dimensions. Are you saying that once the weight of the part is established, all I have to do is weigh them to find out if the dimensions are correct?

Molder:

It isn't quite that easy. Because of the density variation you can't use just weight of the part. It can be an indicator, but if this part is as critical as you say, some dimensional tolerances should be set up, as I've indicated on the sketch.

Designer:

But those aren't the dimensions I need to measure.

Molder:

As long as they're longer than your dimensions, they will be good indicators. They are points that give you good coordinates to measure – and incidentally, sometimes it's useful to mold in a boss or two just for this purpose, to make measuring easier. Set up long dimensions that can be measured measured with gauges or fixtures, and check them against your critical dimensions; if they prove out, you can save a lot of time in your QC checks.

Designer:

So if we combine the two techniques – weigh the part, and if it isn't way off, so that we know it's a bad part – then check these dimensions we've set up, we have an easier way of running QC. In fact, we can do a better job because this is easier than what we had to go through to check those hard-to get-at critical dimensions.

That sounds like a time saver, but I'm not sure it's going to work, because here's a part that seems to contradict your theory. When we get these covers in and weigh them, they're all within a percent or two of the right weight, but the corners are way oversize. How come?

Molder:

Looking at the part, I suspect the problem happened in handling or storage or shipping. I suspect the part was processed properly, but if somebody stacked these parts up, it doesn't take much imagination to figure out what happened... the sides spread on some of them, especially the ones on the bottom with the biggest load on them. You can solve the problem in various ways: pack the covers separately, or develop some sort of packing system that won't spread the sides, or you could even mold a bar across the front – it could be helpful as well during handling and finishing Then remove the bar just before assembly. Even a runner system can be left on some parts to add additional support.

Designer:

That's the last of the problems I had to discuss with you. I think it's quickly become clear that if we talk things out at the beginning of the project, we can save time and money and a lot of headaches later. The designer's discussions with the molder are going to influence the whole product design, how the parts are going to be made and assembled. All the way through the planning stages, talk to the molder, the toolmaker, the finisher.

Molder:

Absolutely. It helps you out, and it helps me out. We'll have to get together and have another of these talks soon.