PLASTIC PRODUCT DESIGN FOR ECONOMICAL INJECTION MOLDING - Glenn L. Beall

Part 3: Process Limitation and the Three Design Elements

The first two articles in this series dwelt with the history of product design and the creative process that culminated in hypothetical new injection molded product sketches. The next challenge is to reduce the concept sketches to detailed piece-part drawings or CAD data bases. However, before starting that time consuming process it is highly desirable to determine whether or not the parts of the new product are moldable and what the molds and molded parts will cost.

The sketches, with their chosen material and approximate overall size, shape, and estimated wall thickness, can be used to secure tentative tooling and molded part cost estimates. These preliminary estimates will indicate whether or not the product is moldable and is within an acceptable cost range.

If the concept passes this crucial test it is then safe to proceed with the detailed design of the individual parts that combine to produce the new product. This is a critical part of the design process which defines the information the mold maker will use to build the cavities that will produce the required plastic parts. The molded parts can only be as good as the cavity. The cavity cannot be better than the part drawing. This is why part design is just as important as product design. However, it is in this area of part design where many perfectly good product designs begin to fail. The product designer and the part designer may or may not be the same person. If not, there is a possibility of incomplete transfer of important information and misunderstandings.

As the project progresses from the product design phase to the part design phase, the designer's emphasis will change. All through the product design phase, the designer was working in the creative realm of searching for a structure, process, and material combination that would satisfy the product's functional requirements within an acceptable cost. This free-thinking part of the project is basically undisciplined and there are few, if any, rules to guide or restrict the designer's creative thinking. The only important rule is that the resulting product concept must be commercially acceptable and is hopefully superior to competitive products.

Piece-part design, on the other hand, is highly disciplined. By trial, error, and experience each of the industry's different molding processes have evolved their own part design guidelines. By following these guidelines, it is possible for both experienced and novice designers to produce an acceptable part design.

DESIGNING FOR THE PROCESS

The injection molding process can be defined as a closed mold, melt flow, high temperature, high pressure, capital intense process that is capable of producing large quantities of complex, precision parts at a relatively low cost. This is a wonderful process which has become the product design community's favorite plastic molding process. The process does however have its limitations.

For example, a designer could have designed a 7.600 in. long nylon part with a nominal wall thickness of 0.012 in. This is twice the thickness required to provide the required electrical insulation. Regrettably, the injection molding process cannot cause the nylon to flow that far at that thickness. Other non-melt flow processes such as thermoforming and blow molding could produce even larger parts at that thin wall thickness.

Faced with this problem some molders would attempt to force the nylon to fill the cavity by increasing the injection pressure and speed. Standard injection molding machines are capable of developing cavity packing pressures at 20,000 PSI. Some special machines intended specifically for thin wall molding are capable of developing pressures of 40,000 PSI and higher.

The molecules that make up plastic materials have a comfortable intermolecular spacing. In the melt phase these high pressures reduce these intermolecular spaces. Packing pressures will be the highest near the gate and the lowest at the periphery of the cavity. The molded part will exhibit more shrinkage at the periphery of the part than near the gate. Parts with non-uniform shrinkage will have increased levels of molded in stress. Internal stress encourages warpage and make it difficult to maintain precision dimensions. Stress can also result in a decline of heat deflection temperature and resistance to aggressive chemicals. It is good to remember that between two parts equal in every way except one was molded with higher injection pressure than the other the one molded with the higher pressure will always contain a higher level of molded-in stress.

A different molder faced with the same problem might attempt to fill that cavity by increasing the melt temperature. Generally speaking the higher the melt temperature, the easier or further the material will flow through a cavity.

Overheating a thermoplastic material or holding it in an injection cylinder for a prolonged period of time will result in breaking the long molecular chains that form plastic material. The longer the molecular chains, the stronger the plastic material will be. We spend a lot of money for these high class engineering materials. It is detrimental to overheat them and reduce their impressive physical properties.

Higher melt temperatures also result in longer molding cycles and higher part cost. This is due to the added time required to cool the hotter plastic material until it has regained strength enough to retain its shape.

The ideal part design is one which has been proportioned so that it can be molded at a modest injection pressure and melt temperature. This combination of molding conditions is different for each type of plastic material and part design.

A new product development project could call for the creation of a no frills, low cost in hotel room coffee maker. Eleven of the coffee maker's components are to be injection molded polypropylene.

Another project might result in a one piece, low cost polypropylene lawn chair. At this stage of the project the lawn chair and the coffee maker exist only as a design check list and/or a specification sheet and some sketches. In order to proceed it is now necessary to finalize the design and prepare detailed engineering drawings or CAD data bases.

The coffee maker components and the lawn chair are both injection molded polypropylene. Their design requirements are however, very different. The coffee maker is an assembly of plastic and metal parts. These parts have to fit together to provide a desired function. Some of the parts will require close tolerances for water tight fitments. The design must allow for efficient assembly with metal fasteners, snap-fits, and welding. Easy disassembly for repair or recycling may be required. Coffee makers are not normally subjected to heavy loads, except perhaps by being dropped or by the stresses created during assembly. It is not likely that the coffee maker will be used out-of-doors. The assembly will however be subjected to elevated temperatures. Some of the parts will be in contact with hot water, coffee, cleaning fluids, and maybe sugar, cream and their substitutes.

The lawn chair has very different design requirements. As a one piece product there is no consideration of ease of assembly or co-operation between mating parts. Precision dimensions are not required. The only dimensional considerations are that the chair be of a size to comfortably accommodate a human being. Another desirable feature is that the chairs be stackable to minimize shipping cost and reduce storage space for both the manufacturer and the end user.

These chairs can be used indoors and out-of-doors where they will be subjected to ultraviolet light. They are at risk of coming into contact with all sorts of lawn and garden chemicals and insect repellent sprays that are stress cracking agents for some plastics.

The primary challenge in designing lawn chairs is to create a structure that meets the market's demand for relatively low cost while providing the necessary strength to safely support a full grown man with a wiggling child in his lap. If these chairs are used on grass, or a stone or brick patio, the bottom free end of the chair legs tend to be held in there as molded positions. If the same chair is used on a waxed tile recreation room floor, the free ends of the chair legs can easily spread apart when heavily loaded. The back legs of a chair are subjected to the highest loads. If they fail, the person in the chair falls backwards. This has resulted in severe injuries when the back of that person's head strikes the floor.

Design engineers have been successful in designing the shapes of these chair legs to maximize the stiffness of a non-engineering material like mineral filled polypropylene. The Business and Institutional Furniture Manufacturers Association has established recommended strength values and test procedures for chairs. From a liability perspective, chairs should be designed to meet or exceed these recommendations. Unfortunately some manufacturers have been unable to resist the temptation to ignore these recommendations and are marketing lower cost, dangerously weak chairs.

The coffee maker also presents opportunities for personal injury. Burns from contact with hot surfaces or hot water. Every effort must be made to protect the user from electrical shock throughout the life of the product.

Both products require careful attention to appearance design and human engineering.

Both the coffee maker components and lawn chair are to be made of injection molded polypropylene. Is this a suitable combination of process and material? An observant plastic product designer should already know that injection molded polypropylene is the first choice for one piece lawn chairs and coffee makers and hot water kettles. It is a safe bet that injection molded polypropylene is a good combination for these two new products.

Using the same material and process combination as a competitor is not very creative, but it is very efficient. Several months or years of success in the market place is a better indicator of the suitability of a material and process than any preproduction laboratory testing.

The lawn chair and the coffee maker share many common part design characteristics. However, from a product design point of view they are very different from each other. The coffee maker is made up of a multiplicity of relatively small, thin walled, precision parts. the lawn chair is a one piece, relatively large, thick walled part with no precision dimensions. In spite of these differences in product design, the part design guidelines will be exactly the same for both products.

From the design engineer's perspective, all plastic parts, no matter how complex they may be, can be subdivided into three basic design elements: The part's nominal wall, projection off of that nominal wall, and depression into the nominal wall.

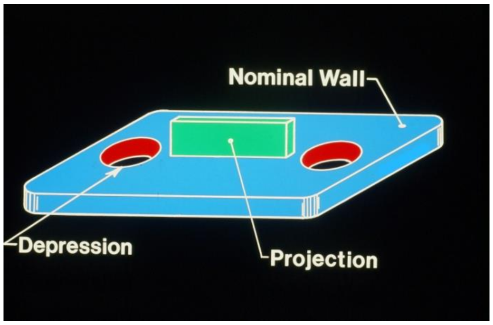

Figure 3a

In finalizing the design of a complex part, it is helpful to concentrate on designing each of these elements individually. This will help prevent overlooking an obscure, but important, detail.

The nominal wall is a part's single most important element. It forms the backbone or framework to which are added all other details on the part. The nominal wall is what gives the part its basic shape, size and overall structural integrities.

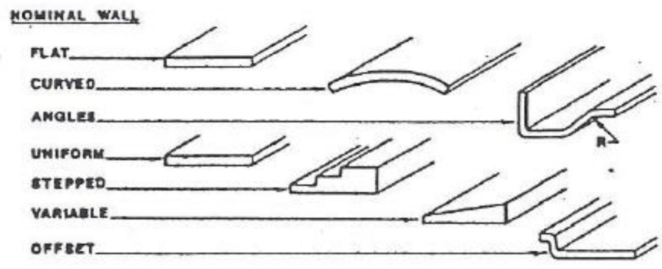

Figure 3b

A nominal wall can be a two-dimensional, flat or curved shape. It can go around corners to create complex, three-dimensional shapes.

Once the part's nominal wall has been established, other functional features can then be provided by adding projections. A projection is any shape that is a non-linear extension off of the nominal wall.

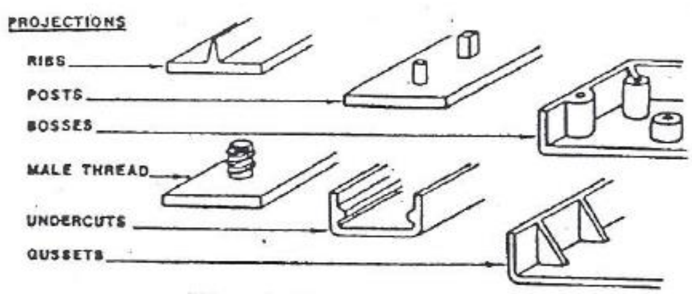

Figure 3c

Interestingly enough, all projections have much in common, from a design point of view. Reinforcing ribs, pegs and gussets are all designed according to the same design guidelines which will be defined in future articles in this Series.

Holes or depressions can also be positioned in the nominal wall or the projections to provide additional functional features on the part.

Round holes, square holes, blind holes and through holes are actually all designed accordingly to the same design guidelines.

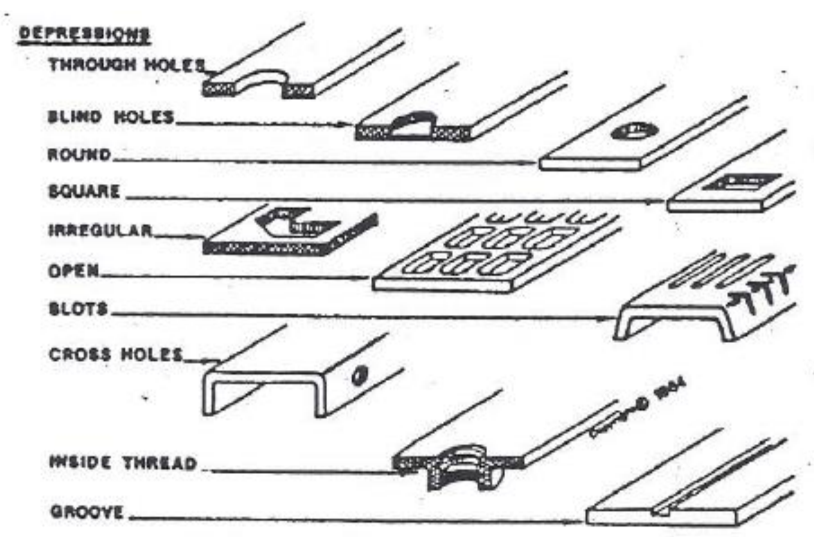

Figure 3d

If the designer understands how to design these three simple elements, he or she will then know how to design any injection molded part, no matter how complex it may appear as a whole.

As an example, the functional requirement of a new product might result in a part like that shown in Figure 3e.

This is not a very complex part but it is a good example of how subdividing a part into its nominal walls, projections and depressions can simplify the design of the part.

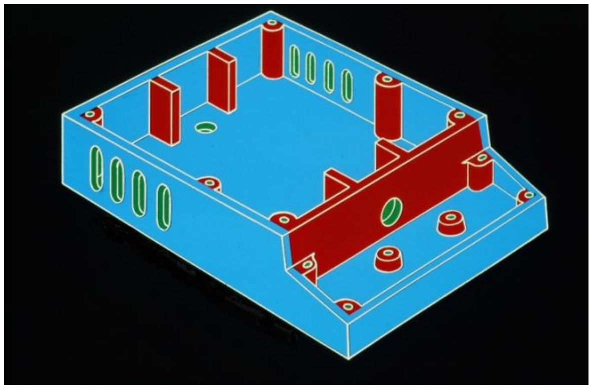

Figure 3e

A plastic part's nominal wall is its single most important element. It is therefore a good idea to design the nominal wall before considering the part's other elements.

Removing the projection, the depression and the standing wall near the front of the part would leave only the nominal wall shown in Figure 3f.

The nominal wall now appears to be much simpler than the same nominal wall shown in whole Figure 3e. This nominal wall could be made even simpler as shown in the exploded view at Figure 3g.

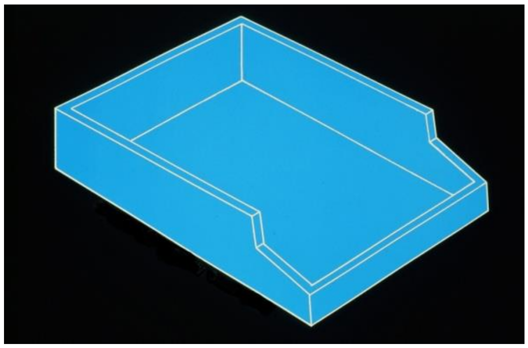

Figure 3f

Viewed in this way the nominal wall has been reduced to five simple, two-dimensional flat plates. The only design guideline for a flat plate is "maintain a uniform wall thickness".

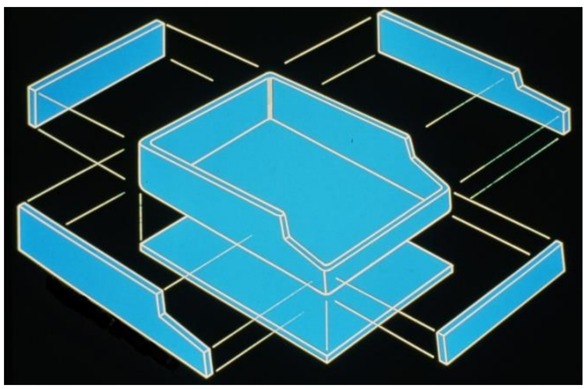

Figure 3g

These five, two-dimensional flat plates can however be attached together to create the three-dimensional part shown in Figure 3f. If each of the five, two-dimensional plates are designed correctly, the three-dimensional part will be properly proportioned for the injection molding process.

As a two-dimensional part transitions to a three-dimensional part, radii should be provided on the parts corners.

All of these surfaces perpendicular to the mold's parting line should be provided with molding draft angles.

Providing a uniform wall thickness, with radiused corners and draft angles, are the only guidelines for a part's most important element which is its nominal wall.

Once the design of the nominal wall has been finalized, additional functional features can be provided by adding projections as shown in Figure 3h.

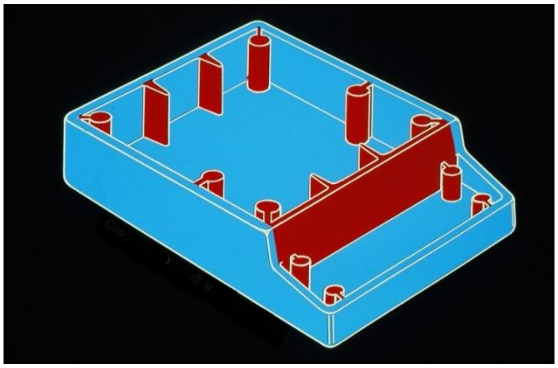

Figure 3h

In this view the ribs have been thinned down and provided with draft angles. The bosses have been moved off of the nominal wall to avoid an increase in thickness at the junction of the boss and the nominal wall. This will avoid sink marks on the outside appearance surface while reducing molded-in stress.

The standing wall which at first glance appears to be a portion of the part's nominal wall is a non-linear extension. It is actually a projection off of the nominal wall and must be designed like a large rib.

The part shown in Figure 3h can be provided with even more functional features by removing material in the form of holes or depressions as shown in Figure 3i.

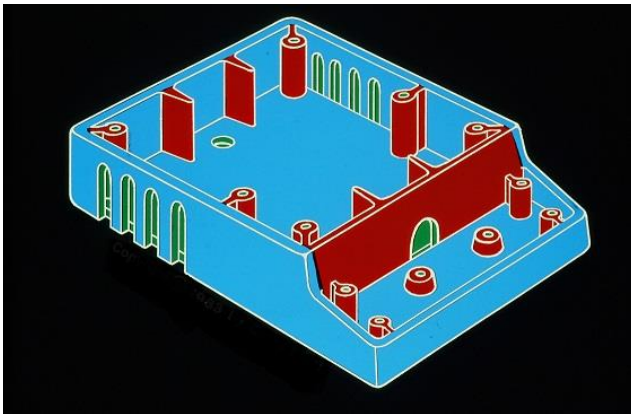

Figure 3i

The holes in the bosses are for assembly and the attachment of internal components. The louvers on the two long side walls provide cooling. The hole through the standing wall is an electric wiring access hole.

The original round access hole (Figure 3c) and the louvers are troublesome as they require three side acting mold mechanisms. These tooling features increase the complexity of the mold design, and the cost of building and maintaining the mold.

These side actions may or may not increase the molding cycle time.

Another consideration is that side acting molds are often larger in overall size than the same mold without these mechanisms. Larger molds require bigger molding machines. Larger machines cost more to operate.

These disadvantages could be eliminated with the following redesign. If the louvers were extended down to the bottom surface of the part, they could be molded on raised cores on the cavity that kissed off on the core of the mold. This would eliminate two side acting mechanisms. It would also change the appearance of the part. This change in the part's appearance may or may not be acceptable. However, the advantages of the redesign are too good to ignore.

The same thing could be done if the round hole in the standing wall was changed to a "D" shape resting on its flat side. A hole of that shape could be molded on a standing core pin on the cavity that kissed off on the sidewalls of the standing wall cavities.

It is interesting to note that the part's shown in Figures 3e and 3i are the same part accept for a change in the appearance of the louvers. Both parts will provide the same function. However, the part shown in Figure 3i can be produced with a lower cost mold that is easier to maintain. The redesigned part has a more uniform wall thickness and can be produced on a shorter molding cycle. The elimination of thick sections will result in less molded-in stress and a reduced tendency to warp.

In this example applying the design guidelines for nominal walls, projections and depressions resulted in improved quality at a reduced cost.

The following articles in this series will cover the important subject of how to design nominal walls, projections, and depressions on parts to be produced by the injection molding process.

All past parts of this series of articles can be found on the SPE Milwaukee Section website at milwaukeespe.org.