

Influence Factors on the Dimensional Accuracy of the Plastic Parts

DANIEL STAN*, AUREL TULCAN, LILIANA TULCAN, TUDOR ICLANZAN

Univ. Politehnica din Timisoara, Facultatea de Mecanica, Bd. M. Viteazul, Nr.1, 300222, Timisoara, Romania

The paper deals with the shrinkage and warpage phenomenon briefly presenting the most important influence factors and their interaction involved in dimensional changes of injected plastic parts. The predictable tolerances, design strategy and the main required action are debated for one final purpose: maximum and yet realistic accuracy of dimensional precision, in a cheapest, easy manufacturability and easy operating injection tool. Finally, there are presented some useful considerations for minimizing the effect of shrinkage and warpage in dimensional variance of injected plastic parts.

Keywords: injection moulding, tolerance, shrinkage, warpage

The design history of any engineering product reveals that most designs are not right at the first time and, inevitably, the main problems result from poor tolerance capability and manufacturing capability.

Generally in industrial manufacturing dimensional tolerance specification will govern the part cost and manufacturability and particularly at part and mould design, a special attention must be paid to a realistic prescription of tolerances.

According to the industrial standards (e.g. DIN 16901), in terms of general tolerances and dimensions, a distinction is generally made between three mould quality classes [1]:

- for "general-purpose" injection-moulding, with fast production cycles in simple working conditions, requiring a low level of quality control and low reject rates;

- for a medium level technical injection moulding, considerably more costly since it imposes higher demands on the mould and specific production conditions; requires frequent quality control checks, having an increased reject rates ;

- for high-precision injection moulding, requires advanced manufacturing precision of the moulds, special production conditions and continuous quality control.

The influence of mould cost on the individual parts costs is largely dependent on proper correlation between the assessed production volume, the time interval over which the moulded part will become obsolete and damping time for the mould. Normally moulds are amortized over 1 – 3 years, involving several million parts.

Beside part shape and design and wall dimensions, the dimensional tolerance is also one of the most important influence factors on mould cost. Unnecessarily tight tolerances for example, can greatly increase costs by generating a large quantity of defective mouldings.

Technically, taking into account mould dimensions the following dimensional tolerances are associated with good moulding practice:

- dimensions up to 150 mm:
 - ±0.15% for precision moulding
 - ±0.3% for technical moulding
- dimensions above 150 mm:
 - ±0.25% for precision moulding
 - ±0.4% for technical moulding

Mould manufacturing tolerances

Thermoplastics, due to their specific behaviour (typically have high elongation and elasticity), do not permit the close tolerances that are specified for metals with their high rigidity, low elongation and low elasticity.

For multicavity moulds, the tool making tolerances are important having a direct effect on the dimensional tolerance of the part.

As an example, in an ordinary mould, for a mould dimension of 30 mm manufactured to within $\pm 0,01$ mm, experience has shown that dimensional consistency better than $\pm 0,03$ to $\pm 0,04$ mm cannot be expected for parts from different cavities in a single shot.

The designers play a key role not only in defining the material and part design but also in determining the costs of an injection-moulded part, and they must also ensure commercially viable tolerance. For this, it is important to avoid excessively close tolerances for plastic components and to define the correct tolerance range of the final product, that should not be as tight as possible but as tight as necessary (fig. 1), and the following important factors must be considered, [2] :

- tolerances in mould manufacturing ;
- additional tolerances given by processing method ;
- additional dimensional variations induced by the moulding conditions (mould design, parameters settings, cooling conditions, part geometry) ;
- warpage due to mould shrinkage ;

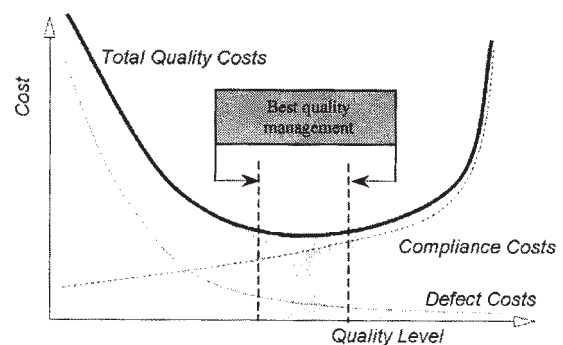


Fig. 1. Quality costs in injection molding [3]

* email: davostan@yahoo.fr

-it is also important to decide whether only a production tolerance is required or whether an operating tolerance is also necessary if the part dimensions are affected by service environment (moisture, thermal expansion, contact with chemicals).

A commercially acceptable value for a production tolerance would be 0.25 to 0.3% deviation from the nominal dimension, but this must be checked against application requirements. When close tolerances are needed, it is important to consult with the injection moulder or material supplier to see if the required tolerances are technically feasible and commercially appropriate.

Choosing a plastic for a specific use can be a daunting task. Designers face a seemingly endless variety of resins and a lot of properties that define them. However, each market usually needs a unique set of properties for the plastics used in it, each product family needs particular plastics and additives.

It has been assumed that for each process there is a fundamental level of inherent variability associated with processing the "ideal" design in realistic manufacturing conditions.

The ITG refers to the *International Tolerance Grade* of an industrial process and identifies what tolerances a given process can produce for a given dimension. For injection moulding, typically procedure and common mould design, the ITG is about 9 to 14, [1,4], and the following formula:

$$T = 10^{0.2(ITG-1)} \cdot (0.45 \cdot D^{0.333} + 0.001 \cdot D), \quad (1)$$

where T is the tolerance, [μm], D is the dimension, [mm] gives an expected tolerance not smaller than $T = 22 \mu\text{m}$, for a nominal dimension $D = 10 \text{ mm}$ and $\text{ITG} = 9$, the shrinkage being responsible for this limitation. The effects of shrinkage grow as part dimensions grow and the precision is smaller.

Thus, holding a tolerance to $\pm(0.02 \div 0.05) \text{ mm}$ is a realistic goal only for a small dimensions part ($D < 30 \text{ mm}$), but could be a real problem for bigger dimensions, leading to special mould design and special working conditions. Under these circumstances, for small plastic parts, the mould equipment would be theoretically manufactured at a tenth of calculated value, at about $T_m = 2 \div 5 \mu\text{m}$.

Technically is accepted that plastic mould making equipment is not proper to generate dimensions with tolerances less than $1 \dots 2 \mu\text{m}$. Even if its technological equipment is not designed to hold these small tolerances, toolmakers sometimes makes "micro tooling" at

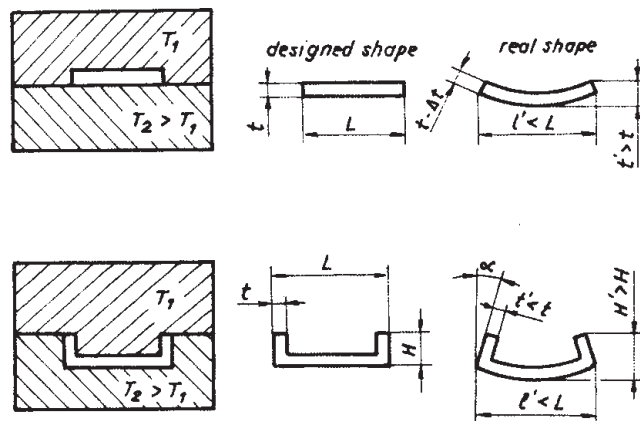


Fig. 2. Shrinkage and warpage of the plastic parts

production-repeatability up to $2 \dots 5 \mu\text{m}$, the lack in mouldmaking equipment precision being compensated by inventively and professional experience. Electro-discharge manufacturing and several other processes, including stereo-lithography, precision laser machining, chemical etching, metal spray and silicon wafer technology, are available to make such tooling.

Shrinkage and warpage of the injected plastic parts

Shrinkage or mould shrinkage reflects the rate of reduction from the mould cavity dimensions to the corresponding plastic part dimensions, due to the stress induced in material, and it's an important information for concurrent design, for material substitution and for specific applications.

Normally, the most important shrinkage and stress of the plastic part material will be noticed along the wall direction. Frequent flow direction changes and a complex geometry of the moulded part could however lead to unexpected material stress having an effect on shrinkage and hence on tolerances and geometric precision. This non-uniform shrinkage can cause warpage and plastic part-distortion of the 3D part geometry (fig. 2), that can be significantly greater than the in-plane mould shrinkage value.

Basically, material shrinkage, as physical property of every plastic material, is ascertained by the phase changes, particularly microstructure and the different density of the polymer from the processing to the ambient temperature. Semi-crystalline materials are particularly prone to

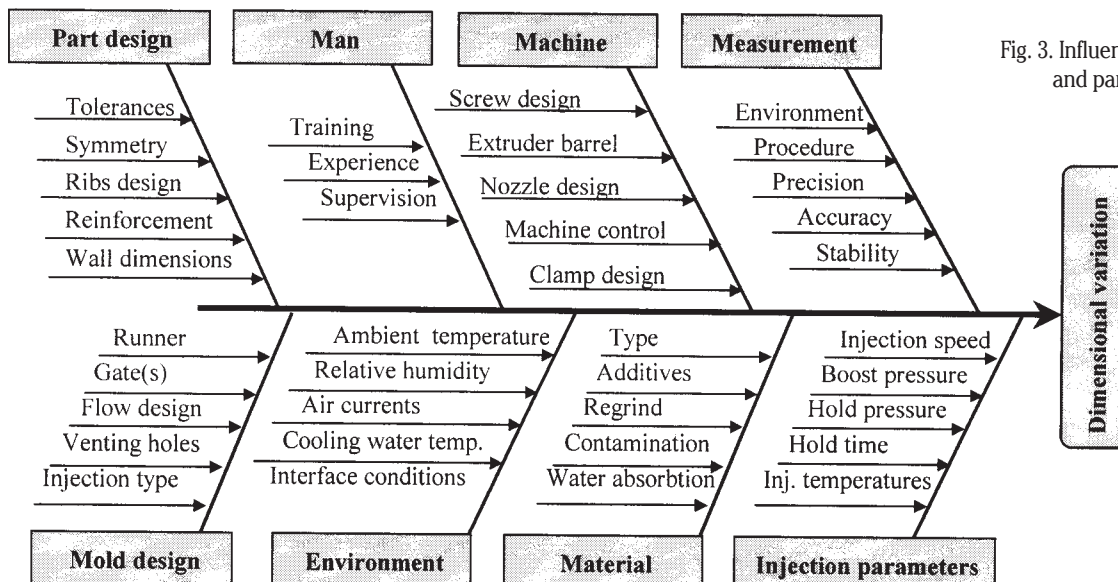
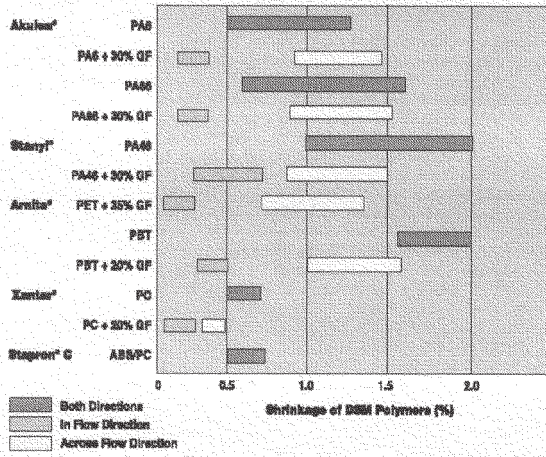
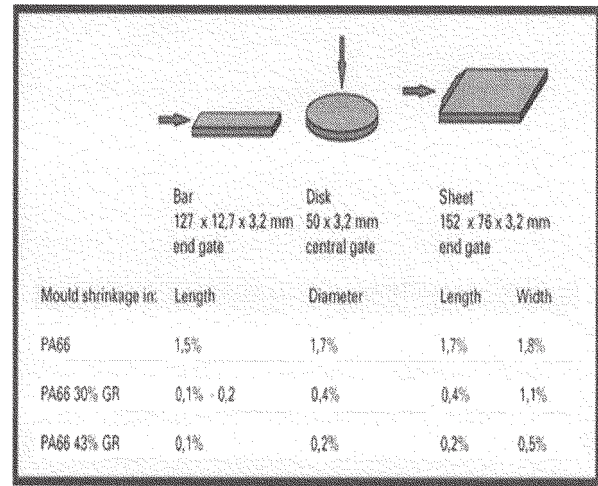


Fig. 3. Influence factors for shrinkage and part warpage [3, 5-7]



a. Materials shrinkage DSM specifications [8]



b. Du Pont shrinkage data sheet [1]

Fig. 4. Variance of shrinkage with flow direction

relatively high thermal shrinkage and amorphous materials tend to shrink less.

One of the main problems related to shrinkages is the lack-in-precision definition of these and the great number of influence factors (fig. 3), that could affect the real part dimensions, [3, 5-7]. Therefore, the maximum and minimum values for various thermoplastics as indicated by the material suppliers (fig. 4), are guidelines only.

For a specific polymer grade, a given part, mould and nest design, it is not possible to predict exact shrinkage values because this is extremely dependent also on additives, on the part/mould design, and the processing conditions used to mould the part, on the geometry of the part and the flow pattern of the resin, so that in production an acceptable control of the final dimension(s) and warpage could be provided only by the variation of several injection parameters :

- mould and melt temperature: it is too high, results in heat sinks due to shrinkage and if it is too cold, leads to moulded-in stresses that may contribute to part warpage;
- nest cooling speed;
- hold time and injection hold pressure.

If regrinding material, a special attention must be paid to the small changes in viscosity, density, or composition that may occur when regrind is mixed with virgin material, if a material is used after it has been stored over an extended period of time, or a switch is made between different batches of the same material grade and producer [3]. Small changes in lot-to-lot material properties can also induce dimensional instability, or inconsistencies in part weight among a batch of moulded parts.

Shrinkage behaviour is also strongly affected by fiber reinforcement. For the basic material, the aligned macromolecular chains shrink more in the flow direction, while for reinforced materials the orientation of the glass fibres along the flow direction is responsible by substantially smaller shrinkage in flow direction than across flow direction (fig. 4).

More over, the material shrinkage subject is not closed at demoulding moment. As a consequence of continued crystallisation and relaxation of moulded-in stresses, where the resin moves towards a more stable state, in the next 16÷24 h after the part has been ejected it follows a post-moulding shrinkage that must be taken into consideration and the dimensional control should be done long enough after injection moulding (48 h recommended).

Mould design considerations

Mould shrinkage starts when the polymer is injected into the mould cavity, so optimum mould design will choose the best gate(s) position, runner diameter, the smoothest flow path and best cycle time in order to prevent or to minimize the shrinkages and warpings across the whole part. Other elements to be considered include cavity walls slope, cavity number, ribs, slides, cooling channels, placement of parting lines where the mould opens, ejectors and even the metal used for the mould.

Runner and gate design should allow the molten plastic to flow smoothly to the limits of the cavity. For conventional cold runner tools, full-round runners are best because they provide less resistance to flow, less contact surface area,

Table 1
RECOMMENDED GATE TYPES [9]

Gate type	Advantages	Disadvantages
Edge / pin gate	- appropriate for flat parts - easy to machine and modify	- post mould gate/runner removal
Submarine (tunnel) gate	- auto gate removal - minimal gate vestige	- technologic difficulties in machining - more difficult to extract
Back gating (trough pin and hole in substrate)	- no part vestige on front side of part	- more complexity - post mould trimming - surface sinks risks
Diaphragm gate	- good concentricity - no knit lines	- appropriate for round parts - post-moulding gate removal
Valve gate (hot runner system, mono- and multi-point injection)	- minimal gate vestige - positive shut-off - minimized post pack	- higher tool costs - higher maintenance costs

cooling is minimized and therefore keep the material molten longer the shrinkage and post moulding induced stress being minimized.

Next, designers decide the gate type, adjust the size and position taking into account factors such as cavity pressure, mould-filling time, resin molecules and fibers alignment tendency.

Gates can also be positioned to hide flow lines that form on a part surface as resin passes through a gate or to relocate weld lines (weak areas where two or more melt streams meet after flowing around a core) to noncritical areas. The type of gate and the location of this relative to the part, can also affect the following:

- part packing;
- gate removal or vestige;
- part aesthetics;
- part dimensions, including substrate warpage

A minimal set of elementary good-practice mould design considerations are to be taken into consideration :

- start with small injection gates and the firsts injection tests will reveal the minimum cosmetic gate vestige to obtain best cavity fill. Large gates should be avoided.

Generally the gate depth should be around 35% of the part's wall thickness at the gate entrance, less for easy-flow material and maximum 50% for a viscous melt.

A good starting point for the gate width should be 1.0-1.5 times the gate depth. As a design tip, the gate area in the mould may be included as a removable insert to facilitate gate maintenance or modification ;

- gates should be located at the heaviest cross section and/or so that the main flow direction will be aligned with the long axis.

Generally the gate would be located at the thickest wall section, in order to facilitate part packing and minimize voids and sink, in a region where potential residual moulded-in stresses around the gate will not affect part function or aesthetics, trying also to:

- minimize obstructions in the flow path (flow around cores or pins);
- minimize jetting (thin-walled components);
- minimize flow marks in critical cosmetic areas;
- minimize potential knit lines (particularly in components with two or more gates);
- allow easy manual or automatic degating.

Furthermore, the shrinkage of the material in the direction of the flow will be different from that perpendicular to the flow. As a result, a rotating part will be somewhat elliptical rather than round. In order to eliminate this problem, "diaphragm gating" [7, 9] can be used which will cause the injection of material in all directions at the same time. The best, but most elaborate way is "multi-pin-gating" injection at several places symmetrically located. This will assure reasonable viscosity of the melt, creating as well as minimal and uniform shrinkage in all directions.

For multi-cavity tools or/and multi-point injection, the cavity layout should be physically balanced so that the melt flows to each cavity in equal times under uniform pressure. An unbalanced runner lead to inconsistent part weights and dimensional variability of the part, even the nests are dimensionally appropriate.

The wall thickness should be as uniform as possible to obtain the best moulding cycle time. Wall thickness ranging from 0,5 mm to 3 mm will ensure good rheological conditions in most injection applications. If the part requires the use of thick wall sections, they should be cored out both to minimize shrinkage problems and reduce the part weight and lower cycle time. Transitions between

different wall thickness should be gradual to reduce flow problems, such as turbulences, back fills and gas traps.

The use of radii (0.5 mm minimum) in sharp corners also reduces localized stresses.

A specific wall draft (0.5÷2) will be applied for the walls along the opening direction. Properly designed deep undercuts are possible if:

- the part does not have sharp adjacent corners
- advancing core is used when the mould opens
- at ejection temperature the material is elastic enough and capable to deflect as it is ejected.

If in service the stress or the deflection of the part under load are high, for a strengthen structure without thickening walls, ribs or other reinforcing features can be added. The primary purpose of ribs in plastic design is to improve the stiffness of the structure by increasing sectional properties. Rib design (fig. 5) can affect part weight, cosmetics, warpage and moldability.

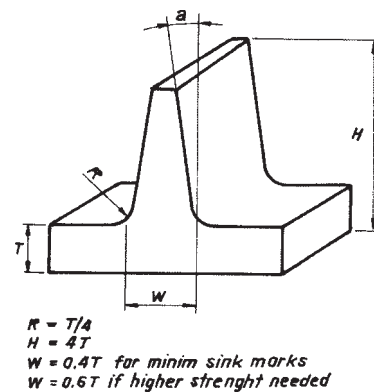


Fig. 5. Exemple of ribs design [8, 9]

Thick ribs can cause internal voids, shrinkage and tendency to warp, as well as sink marks on the opposite part surface. For the same structural effect and added mass and to avoid accentuated sink marks, few thinner ribs are better than a single large rib.

Melt flow entering a thin rib can slow down and begin to freeze off while thicker wall sections are still filling.

Deep, unventable blind pockets or tall ribs should be also avoided or a special attention will be granted to the venting holes that will be placed nearby.

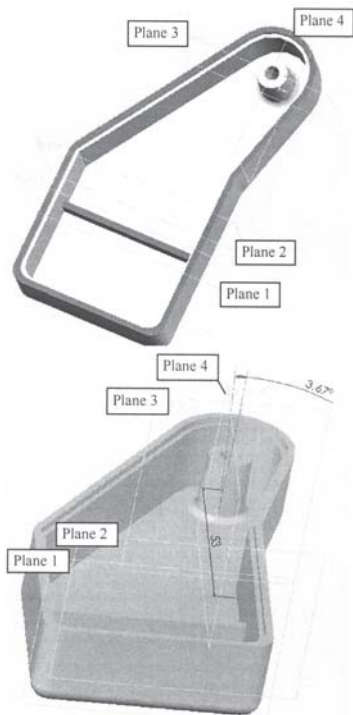
Parameters settings influence

The most part of injection parameters could affect less or more the shrinkage, and through that, the precision and the final dimension of the part.

After injection, while cooling down, the polymer starts to shrink. During the holding stage of the injection moulding cycle, shrinkage is compensated by material pushed into nest at a proper post-filling pressure. On the one hand, for example, a colder mould leads to higher post-mouldings shrinkage and too short post injection parameters (hold pressure, hold time) involve an inconsistent and incomplete shrinkage, deformation of the moulded part and to an alarming variability in part dimensions. The difference of temperature between the 2 plates of daylight section could lead also to important shrinkage and warpage (fig. 2).

Generally, parts moulded under recommended conditions (part and mould design, parameters settings and moulding conditions) are subject to small negligible shot to shot variations in dimensions due to the inherent trifling changes in machine parameters or conditions.

The melting temperature is widely accepted to be the main parameter affecting the rheology, and part dimensions, but the precise one-to-many relationships are



Plane 1 = section through the rib
 Plane 2 = no rib in section
 Planes 3 and 4 = planes through the boss axis, in virtual design position
 Section in Plane 1 = proportional to the virtual shape, but affected by shrinkage
 Section in Plane 2 = dimensions are affected by shrinkage, warpage distortions
 Sections 3 and 4 = unpredictable warpage

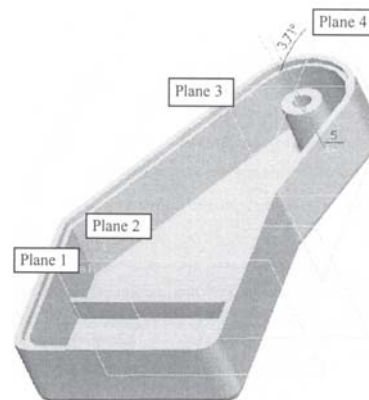


Fig. 6. Warpage in asymmetrical plastic part

generally not available prior to moulding. In symmetrical section, like section 1 or 2 from figure 6, a rib could minimize the internal stresses and the warpage leading to a stable shape similarly to the designed one. Factor scale is the expression of the shrinkage and can be controlled from the earlier stage of tool design and/or later, managing the injection parameters. For asymmetrical geometry of the part, the situation became more intricate and a 3D inspection could reveal the warpage tendencies.

A system view of the classical injection molding process [3, 10] reveal the complexity of this and the possibility to be decomposed into five distinct but coupled and successive interacting stages: plastification, injection, packing, cooling, ejection. The output of each stage not only directly determines the initial conditions of the next stage, but also influences the functionality of one or more others stages, finally affecting the qualities of the moulded part.

A fundamental difficulty in control of injection moulding is that none of the final moulded part properties can be ascertained within the moulding cycle. And because it is not possible to achieve on-line information about material state, structure and aesthetic prior to mould opening and ejection of the part, part quality control is satisfied through a combination of on-line state variable control (through continuous control of the melt state) and off-line cycle-to-cycle adjustment of the machine. But the control of injection moulding process is significantly challenged by the non-linear behaviour of the polymeric materials, dynamic and coupled process physics, and convoluted interactions between the mould geometry and final product quality attributes and there are not precisely known relationships between the machine input variables and final quality attributes. There is a lack of models to define the relationships from inputs to state variables and from state variables to outputs, more than that, the effects of the input variables could interact and the result could be less predictable, [10, 11]. Any change of one parameter has its own influences on the rheological behaviour of the melt, the shrinkage and stress of part material and on at least one other parameters.

For establishing an efficient and applicable steady-state moulding cycle some basic considerations will be taken into account [3, 10, 11, 13]:

- during start-up, the moulding parameters should be set at the mid-point of the recommendations from the raw data sheet;

- shear rate adjustments should be the primary method used to control melt viscosity. Melt temperature adjustments should only be used to fine-tune the process;

- for a minimized shrinkage, special attention should be paid to the temperature of the nest surfaces. An adequate cooling will be able to minimize cycle time;

- to minimize shrinkage issues, adjust second stage pressure as necessary to insure the melt is fully packed into the mould cavity;

- if the lack of quality is due to the gate area, proceed to increase these or to fill the cavity through multiple injection points;

- given the application dependence of precision process control, the methodology for developing precision injection moulding process capabilities is based on a standard decision making process in four steps : measure → analyse → improve → control.

For that, the process variables (at least the most important) could be observable and controllable;

- for process repeatability, the primary criterion used was to see whether part quality could be reasonably controlled from shot to shot. Adjusting the cycle times (injection, part cooling, open close, mould cooling), a steady-state moulding cycle will be particularly determined for each tool. Where needed and also possible, standard practices and suggestions will be formulated and refined for precision injection moulding;

- the professional experience and preliminary experimental injection shots could lead to success in fixing the proper parameters.

The shrinkage and warpage act less or more and thus the real shape of the tool does no longer correspond to its original and basic design. There are two correction ways to follow:

- to accept the injection parameters data sheet (giving proper flow and filling conditions) and to correct the tool. Especially for symmetrically parts, capture first the actual contour of the injected part (digitise the surface of the part) and reshape the tool design to obtain a CAD data set reflecting the proper shape for the nest so that, after shrinkage and warpage, it will correspond to demands then make the correction on tool.

- to modify one or more parameters presented in figure 3, to find a combination giving minimum values for shrinkage and warpage. First it must be established the process capability index, (Cpk), as a measure of how capable a process is of making parts that are within specifications [6, 12], for enabling the establishing of the process tolerances.

Find next the critical factor and experience fine modifications till the product correspond to quality specifications. Once the proper combination revealed, a preventive attitude and a proper strategy will be implemented. Procedures of Statistical Process control (SPC) and other proper quality assurance tools must be applied in mass production, [2, 6, 13].

Additional variance sources

Beside the presented factors, there are more three others important sources of variability : the injection moulding machine design, environmental and human factors [10].

Different injection cylinder and clamp design of the varying moulding machines will induce very different machine dynamics, and provide different levels of moulded part quality for the same process set points. Even identical machines, from the same manufacturer could have smaller differences in their controller due to internal controller variation relating to the shot size, injection velocity, switchover point, injection- and pack-pressure, etc.

The physical environment will also introduce variation through interaction with the process. For instance, outdoor temperature may affect the effectiveness of the coolers that determine the temperature of the plant water. Indoor temperature can likewise have significant effect on the mould temperature as well as the post-moulding behaviour of the moulded parts.

Humidity can affect the dryness of the polymeric material entering the barrel, introducing thus further quality inconsistencies.

An interesting way to reduce, or eliminate, post-injection stress by modifying the interface polymer-metal properties could be the ultrasonic activated nozzle, [14, 15], is still debated but few research teams from EU and Asia confirm his viability in articles from the last decade, [16, 17, 18].

A particular effect of the ultrasonic activation, namely "the thermo-pellicular effect", specific for the particularized activation conditions of the viscous elastically fluids under pressure can change the properties of the contact interface and offers the possibility for increasing the transfer velocity in the proximity of the wall. In extrusion for example, it was estimated an increasing of 80÷200% of the extrusion discharge and also of the productivity and next to the top of the ultrasonic horn we notice a local increase of the melt temperature with 40..80 °C [14].

Human factor is also an important variance source. At least 3 persons are involved in mould processing: mould designer, process engineers and the operator, and many others in the product development project. Sometimes, the lack of quality and higher costs could have the significance of a misunderstanding between these persons having not the same acceptance for "optimal processing conditions".

Conclusions

The best tolerances that can normally be met in injection moulding, with classical equipment is inside of a total composite error between 0.05 and 0.15 mm, as shown in this paper. For high-precise small parts, the tolerance of whole dimensions can be controlled in tolerance ± 0.05 mm and particularly important dimension can be controlled in tolerance not less than ± 0.02 mm. Closer tolerances means higher tooling cost and fine control of the moulding conditions usually will be required.

For the plastic part, "tolerance" may have different interpretations and these different understandings must be aligned. For the part designer it means functional limitations, to the mould designer it means technology and tool fabrication variation allowance and the mould maker looks at it as production tolerance.

Any over estimation in part quality levels requires increased investment, processing time, supplementary control and inspection costs and will lead to the tendency to sacrifice production efficiency for the sake of quality.

Design for moldability is the basis for efficient management and involve close coordination at least between the designer, moulder, and even raw material supplier. Team working capability and professional skills of the involved persons are important elements deciding on the final results.

If complex mouldings are to be produced to close tolerances is highly recommended to involve prototyping techniques in predicting the shrinkage and warpage behaviour in particular injection conditions, acting as suggested in present paper.

If the part needs higher tolerances in (even) a small region, this location will be treated as a critical feature and will be used for the alignment of others less important features.

Bibliography

- 1.HASENAUER J., KUPER D., e.a - Top Ten Design Tips, <http://plastics.dupont.com>
- 2.DAY D., Process Capable Tolerancing, Raines M. and Swift K., CapraTechnology Ltd /UK
- 3.KAZMER D., Precision process control of injection molding, Precision Injection Molding, Hanser Publishers /R.W. Friedl, J. Greener, 2005
- 4.*** http://en.wikipedia.org/wiki/Injection_moulding
- 5.RAUWENDAAL C., SPC: Statistical Process Control in Injection Molding and extrusion, Hanser Publishers, 2000, p. 88
- 6.FETECAU C., Mat. Plast., **42**, nr.4, 2005, p. 290
- 7.GHITA E., GILLICH G.R., BORDEASU I., e.a., Mat. Plast., **44**, nr.2, 2007, p. 158
- 8.DIMONIE D., RADOVICI C., e.a., Mat. Plast., **44**, nr.2/2007, p.148
- 9.*** http://www.dsm.com/en_US/html/dep/dimensionalstability.htm;
- 10.*** <http://www.sdp-si.com>
- 11.KAZMER D., DANAI K., Control of polymer processing, The mechanical Systems design Handbook: Modeling, Measurement and Control, CRC Press Inc, 2001
- 12.CUPSA E., DRAGHICI G., Academic J. of Manufacturing Engineering., **4**, nr.2, 2006, p.23
13. MANN, J.W., *Plastics Engg.*, 1974, 30, nr. 1, p.25
- 14.STAN D.V. - Doctoral thesis, Universite Montpellier 2 /France, 1999;
15. ICLANZAN T., STAN D., *Mat. Plast.*, **42**, nr.2, 2005, p.93
- 16.YURONG C., HUILIN L., Influence of ultrasounds on the processing and structure of polypropylene during extrusion, *Polymer Engg & Science*, **42**, Issue 7/2002, p.1534
- 17.SHAOYUN G., YUNTAO L., e.a., *Polymer International*, **52**, Issue 1/2003, p.68
18. *** <http://www.ultramelt.org>, Ultra-Melt Project

Manuscript received: 11.01.2008