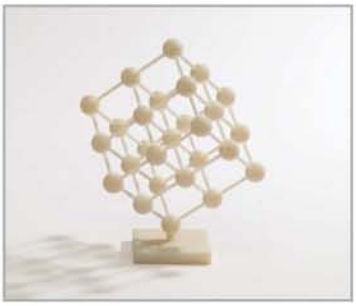
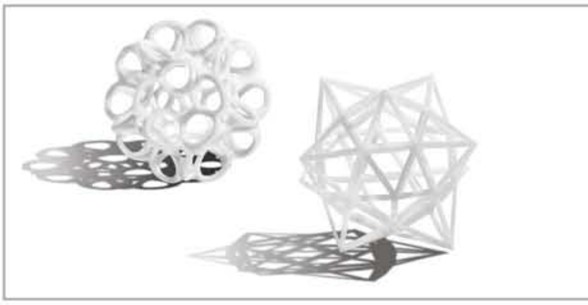
3D Printed Injection Molding Tool ("PIMT") Guide

Objet Ltd.





Injection molding is a high speed, automated and versatile process that can produce high precision complex three dimensional parts from a fraction of a gram to more than a hundred kilograms, from virtually any plastic material. The injection molding process is very fast and therefore very economical. Yet, every part needs its own costly injection molding tool, thus injection molding is economically viable only when mass-production is needed (usually more than 10,000 parts). Injection molding tools take from weeks to months to build. Very often, design mistakes are made and mold changes are needed. In this case time-consuming mold corrections have to be made. Tools may go back and forth a few times from the producer to the tool builder until the final part design and quality are achieved, increasing costs and product time-to-market.

What about when only a small number, say 100 parts are needed?

What about when these parts are needed fast, say in a couple of days, or even in a few hours?

One way would be to produce the parts by rapid prototyping methods. Several technologies can be used for fast prototyping which can produce high precision complex parts. However, fast prototyping techniques use special plastics, usually very different from those that will be used in the real part production. A second way involves the use of lower-cost, simplified and smaller tools (e.g. one cavity instead of multi-cavity) or using tools of softer and cheaper materials (aluminum). In this case some cost is saved but still, tools are expensive and, as with production tools, time is needed to build them and correct them.

There is another and better way: The Objet way.

Object has developed new materials that permit 3D printing of the tool itself, not the parts. The Objet printed injection molding tool (PIMT) is installed on the injection molding machine and the plastic part is injected using the actual production material. PIMTs can be built cheaply and fast, ready to inject in a few hours. If design changes are needed, the corrected PIMT may be ready and installed in the injection machine by the evening shift.

PIMTs are suitable for small production series but since tools cost is low, they are commercially viable also when only a few parts are needed.

Objet PIMTs are not production tools; however they provide a clear advantage, being both cheaper and faster when a limited quantity of parts is needed.

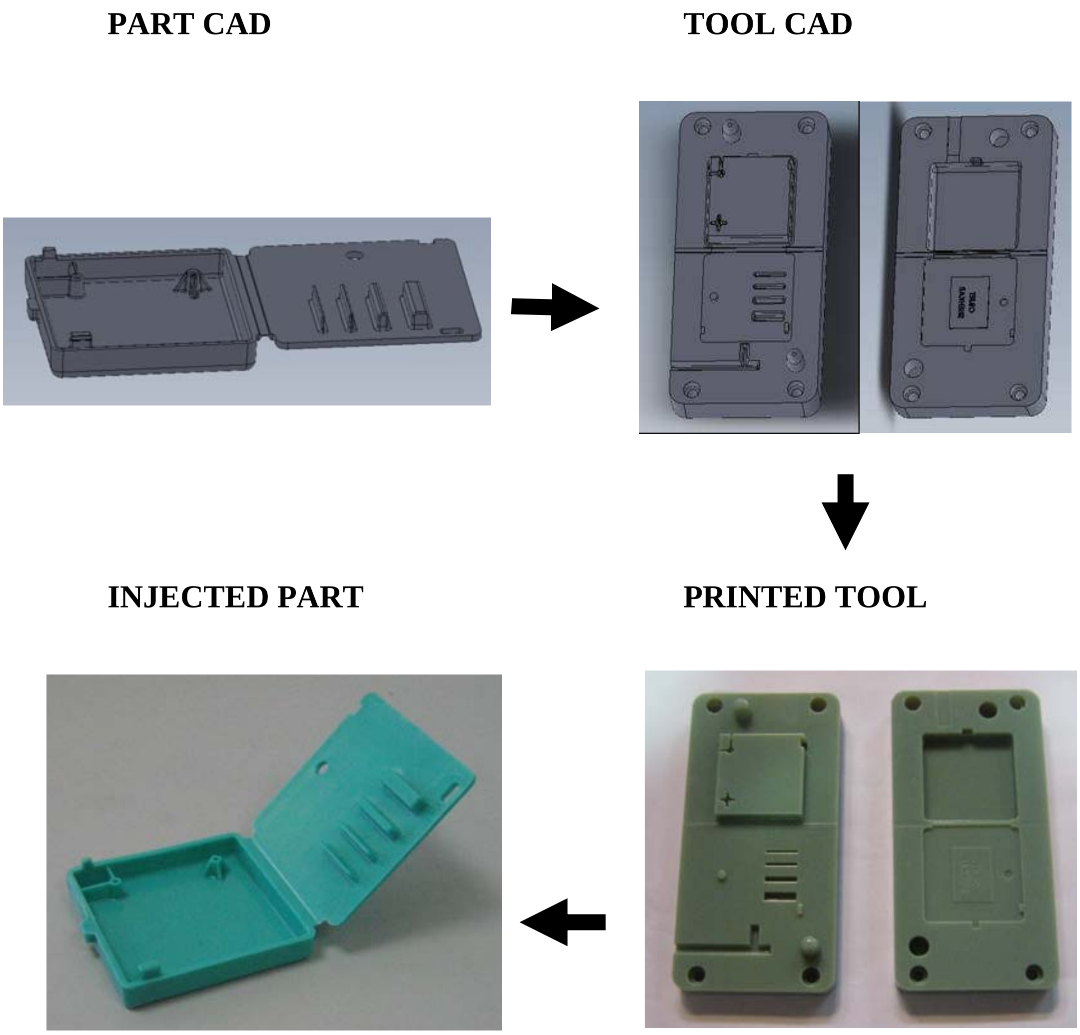


Figure 1: From CAD to part in only a few hours!!!

The following guidelines should be followed when working with Objet PIMTs.

Objet PIMT Capabilities:

1. Polymers:

   Objet PIMTs can be used to inject polyethylene, polypropylene, polystyrene, ABS, and thermoplastic elastomers. However as the polymer's processing temperatures increases, the tool's life will tend to decrease.

2. Part Features:

   Objet PIMTs are able to produce complex parts including features seen in most injection molding parts such as living hinges, holes, thin and thick walls, bosses, gussets, ribs, press fits, snap fits and logos (See figure 1).

3. Part Size:

   Objet PIMTs are especially suitable for small parts. Maximum size is limited according to the relevant machine.

4. Mold Accuracy:

   The overall accuracy for the PIMT, on a well calibrated machine is typically 20-85um for features below 50mm and Up to 200µm for full model size.

   (Depending on geometry, build parameters and model orientation)

5. Part Accuracy

For part accuracy measurements, please refer to the Objet Part Case in the Appendix

6. Tool Life:

   Depending on polymer and part complexity, PIMTs are suitable for the injection molding of up to 100 parts or more. If multi-cavity molds are used then the number of parts can be multiplied.

PIMT Design Guide:

1. Material:

   PIMTs should be printed with high heat resistant, yet strong material. The preferred material for PIMTs is the Objet ABS-like material.

2. Sprue and Runners:

   PIMTs are printed using Objet ABS-like material which is a strong and heat resistant polymer; however it is not as strong as metal. Direct injection on the PIMT surface will cause a reduction of tool life. It is recommended to use the PIMT as an insert installed on a metal mold base including the sprue, sprue puller and the first part of the main runner. To assure a good contact between insert sides, they should protrude from the mold base by about 0.1- 0.2 mm. Leading pins can be printed in the insert to assure good matching

   (figure 1 and 2).

Figure 2: Mold base and PIMT inserts: Left: moving plate (Cavity), Right: stationary plate (Core)

3. Standalone Mold:

   Full stand-alone PIMTs can be also printed. However it is recommended that the sprue bushing is made as a metal insert.

4. Tool Printing Direction:

   PIMTs show some level of surface printing lines. These lines may affect the polymer melt flow during the injection molding process resulting in flow phenomena such as flow unbalance, short shots, hesitation and flow marks. It is highly recommended to print the tools in a manner that the printing lines are oriented in the main flow direction and not transverse to it.

5. Printed surface finish:

   Being a rigid molding tool, PIMT is designed without significant undercuts by itself. This fact makes PIMTs great models to be printed in gloss mode. Better mechanical properties like surface quality and strength are additional reasons for printing PIMTs in gloss mode.

6. Radii and Features:

   Objet 3D printing system allows for the printing of very small radii and features. However, very small, thin-walled or too-tall features may compromise the mechanical integrity of the tool. Too small radii act like notches and can become a starting crack point that will reduce the tool life. It is recommended to use bigger radii as possible and to avoid sharp corners.

7. Pressures and Temperatures:

   High pressures or temperatures will reduce tool life. It is recommended to design the tool in a way that pressures and temperatures are minimized as much as possible, for example by increasing gate size and reducing flow lengths by using multiple gates.

8. Injecting against the wall cavity:

While injecting against the wall cavity may be a desirable practice with metal molds (to prevent jetting for example) it is not recommended in the case of PIMTs. A high pressure developed on the wall cavity when injecting against the wall may reduce tool life. When injecting against the wall cannot be avoided, use generous radii in locations where flow changes direction.

9. Venting:

Design parts with good venting. Decrease pressure and temperature-riser phenomena such as hesitation and dieseling.

10. Draft angles:

Higher draft angles are recommended to reduce stresses on the tool during ejection. Draft angles of at least 1.5^0 are recommended.

Molding with PIMTs:

1. Processing Parameters:

   High temperatures and pressures will reduce tool life so a good balance between them should be achieved. Use materials with good flow ability to reduce both. Materials can be injected in normal processing conditions.

2. Cooling:

   Objet PIMTs are made from plastic and as such, have low heat transmission.

   The PIMT surface can reach high temperatures during injection molding and this heat will not be effectively removed by cooling water flowing in the channels as in a metal tool. Thus, it is much recommended to cool the PIMT surface to about 50^0C by applying air directly to the surface for about 1 minute. This will extend the tool life dramatically. Further cooling of the surface to temperatures lower than 50^0C will extend the tool life even more. Air cooling systems can be integrated and used also when working in fully automatic mode.

   Because of the low heat conduction, cycle time will be longer compared to metal tools. Depending on part size and thickness, cycle times may reach a few minutes; however, since PIMTs are not intended for mass-production, longer cycle times may not be a critical parameter. A small series production of up to 100 parts can be made in one production shift.

   Since cooling times are long, parts will shrink more than when using metal molds.

3. Ejection:

   Parts molded with PIMTs can be ejected using the same ejection systems as for metal tools (Figure4). Pinholes can be printed easily. Metal ejector pins should be used.

Figure 4: Ejector pins through PIMT insert

Appendix:

CASE STUDY: Injection Molding of a Generic Part using Objet PIMT

An injection molding "generic" part including the most common features encountered in most injection molding parts was designed (Figure 1).

The part has the form of a box, 6 cm long by 6 cm wide by 1 cm high with 2 mm wall thickness, and includes a living hinge (0.25 mm thick at its thinnest), ribs of thickness 0.5 mm, 1.0 mm, 3.0 mm and 5.0 mm, a corner boss with ribs, a center boss with gussets, a hole, a snap fit and logo.

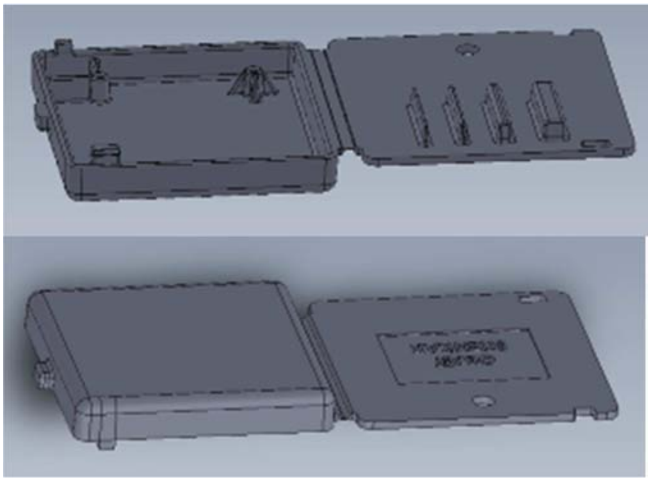


Figure 1 – IM "generic" part, both sides

The following lines (see Objet 3D PIMT Guide) were used when designing the PIMT:

- Inserts were printed from Objet ABS-like Digital Material (Figure 2)**.**
- The PIMT was designed as an insert to be installed in a metal mold base including the sprue, first part of the runner and the sprue puller (Figure 3).
- The PIMT contains four holes for screwing the insert to the metal base, a runner connection to the metal mold base runner, part gate, part cavity and core, leader pins and venting.
- The PIMT was designed to protrude 0.2 mm from the mold base to assure good contact upon mold closing.
- The PIMT was printed in such a way that surface printing lines coincide with polymer flow.
- All corners are rounded to avoid sharp features that may act as notches and reduce tool life.
- The part gate was positioned at the box "cover" side in order to prevent injection against the wall that may reduce tool life
- Venting was provided at end of fill and in confined areas (Figure 2).
- Draft angles are 1.5^0.
- This specific PIMT did not include ejection pinholes or cooling channels. Ejection was done by hand and cooling was done by air surface cooling (see PIMT guide: "cooling").

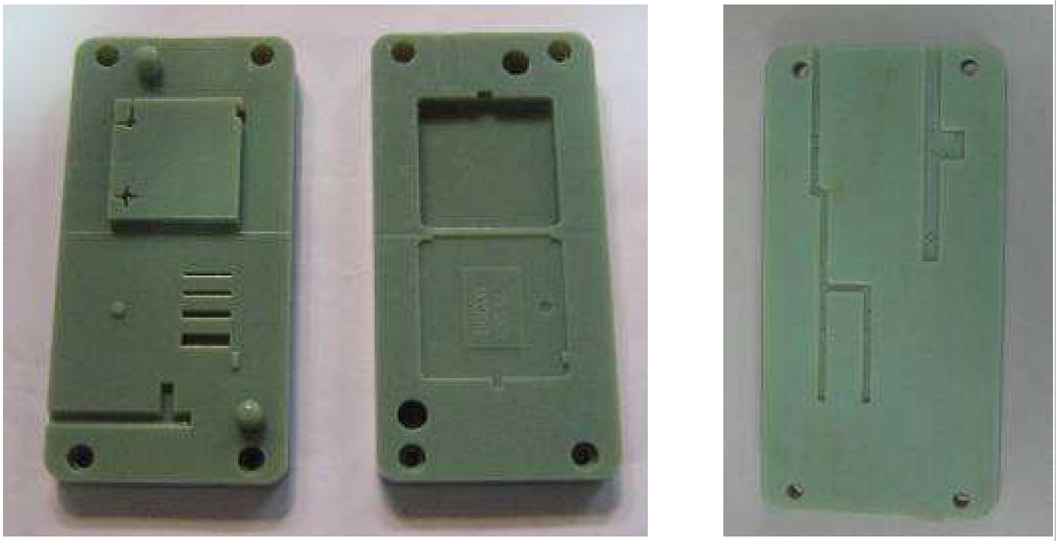


Figure 2 – IM "generic" part PIMT. Left: core and cavity, Right: core opposite side (venting details)

Figure 3: Mold base and PIMT inserts: Left: moving plate (core), Right: stationary plate (cavity).

The mold base containing two sets of PIMTs was installed in an Engel 75 Ton machine. A high flow PP homopolymer grade (Capilene U77, Carmel Olefins) was chosen for this application. The following starting injection parameters were used for the injection molding:

- Cylinder Temperatures (^{0}C):

Zone 1	Zone 2	Zone 3	Zone 5 (Nozzle)
190	200	210	220

- Shot size: 62 cm^3 (50 mm)
- Injection Speed: 55 mm/s
- Switchover: 10 cm^3 (8 mm)
- Injection Pressure: 110 bar
- Clamping Force: 550 kN
- Delay before Plasticizing: 100 s
- Backpressure: 10 bar
- Screw Speed: 45 %
- Holding Pressure: 50 bar
- Holding time: 10 s
- Cooling time: 180 s
- Air cooling time: 1 min.
- The injection unit was retracted after every shot in order to minimize contact time between the hot nozzle and the mold that is not cooled.

Results:

- Good parts with a little flashing were obtained right after the first shots (Figure 4). About 50 parts were injected.

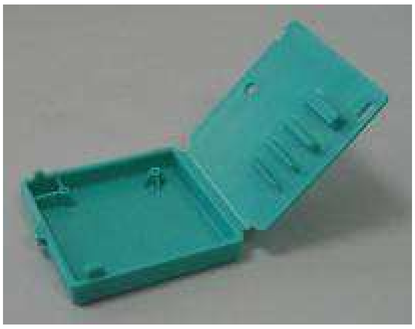


Figure 4: Injected parts

- Insert shot to shot temperature variation:

At the end of each cycle the temperature of the insert's surface was recorded with an infrared thermometer (Figure 5). Following, 1 minute of air cooling was applied to the insert’s surface. After air cooling the insert's temperature was reduced to about 50°C.

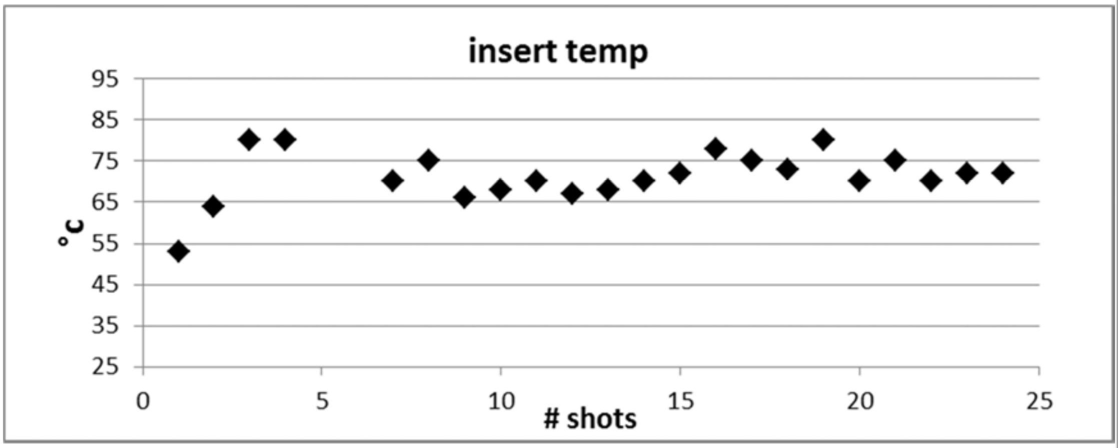


Figure 5: Shot to shot variation of insert’s surface temperature

As can be seen in Figure 5, the temperature of the insert stabilizes after the 5th molding cycle at between 65-75°C. A significant heat buildup would be expected if air cooling was not used, which would lead ultimately to premature failure of the insert.

- Part shot to shot dimension variation:

Specific and representative dimensions of the part were chosen in order to track shot to shot dimensional variation. The representative dimensions are detailed in Figure 6. All dimensions were measured at least 24 hours after the molding session to allow for the material's natural shrinkage.

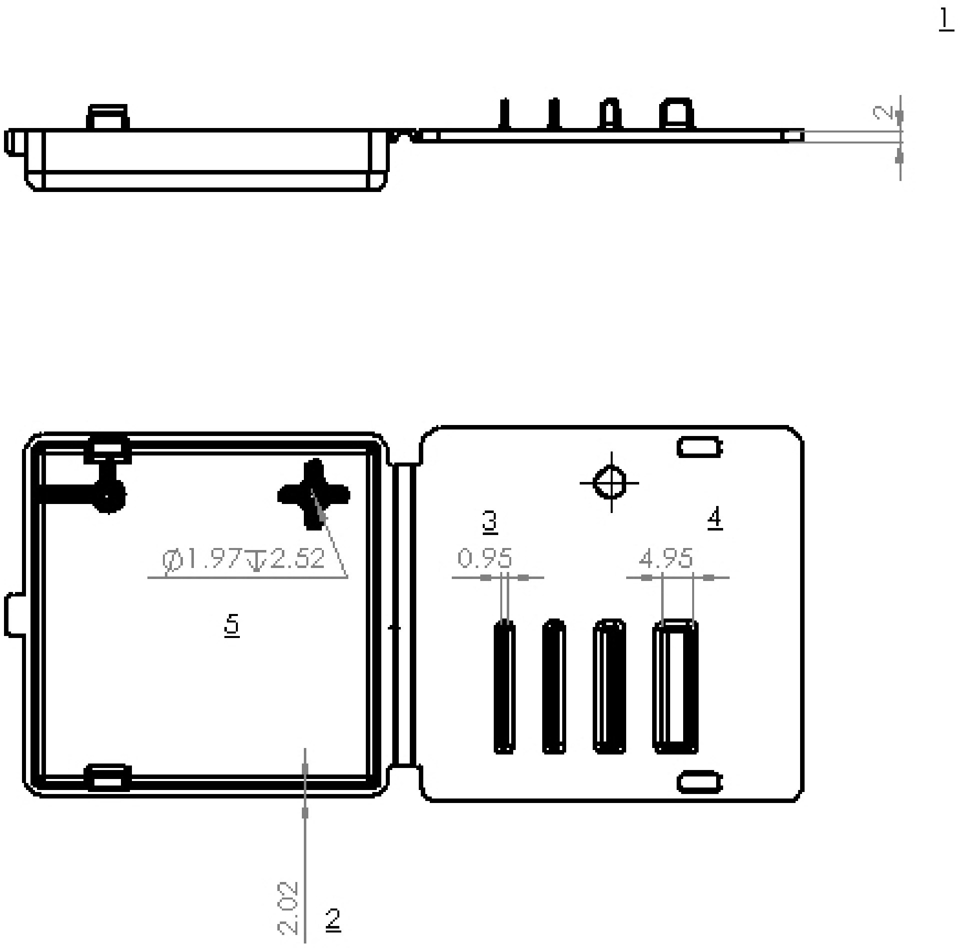


Figure 6: Locations measured for evaluating shot to shot dimension variation

The following figures 7 to 11 demonstrate the shot to shot dimension variation for the tested insert

t nominal - the dimension according to the CAD,

t nominal-2% - the dimension after 2% linear shrinkage (typical for PP) is taken into account.

lsl and usl - the lower and upper spec limit respectively according to commercial tolerances for PP injection molded parts (SPI Plastics Engineering Handbook, 5th Edition, Chapter 28 –Molding Tolerances)

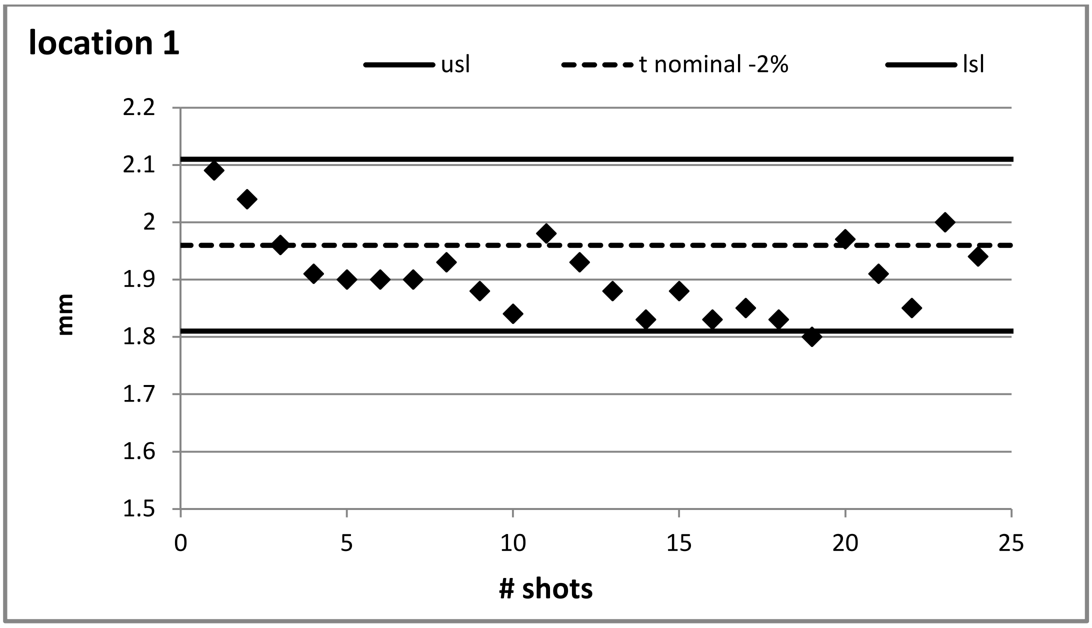


Figure 7: Parts dimension variation in location 1 – lid thickness

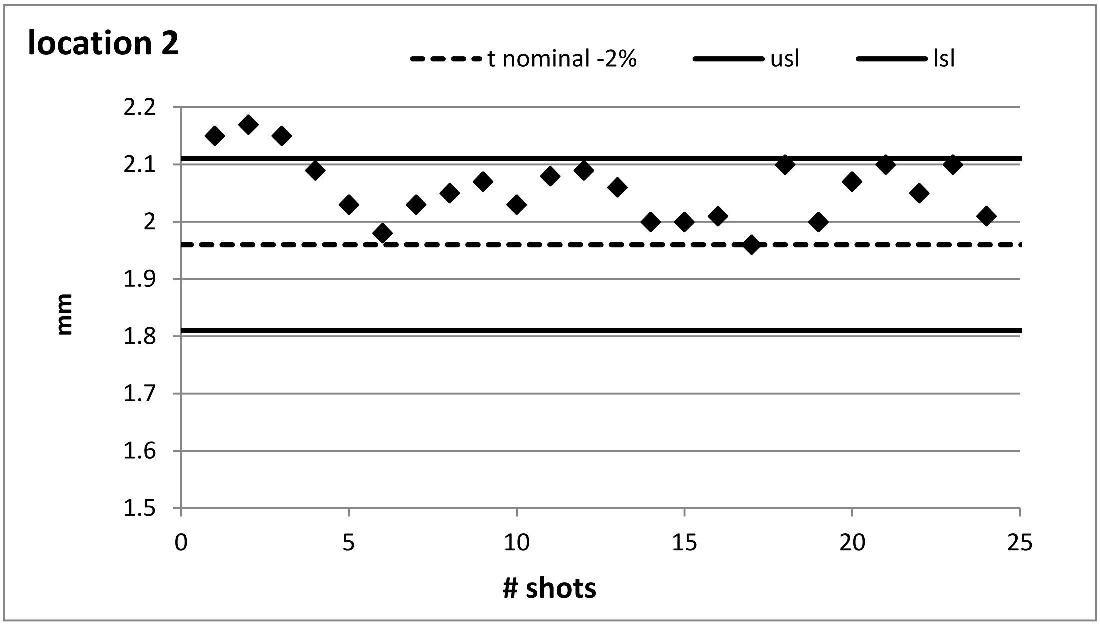


Figure 8: Parts dimension variation in location 2 – box's wall thickness

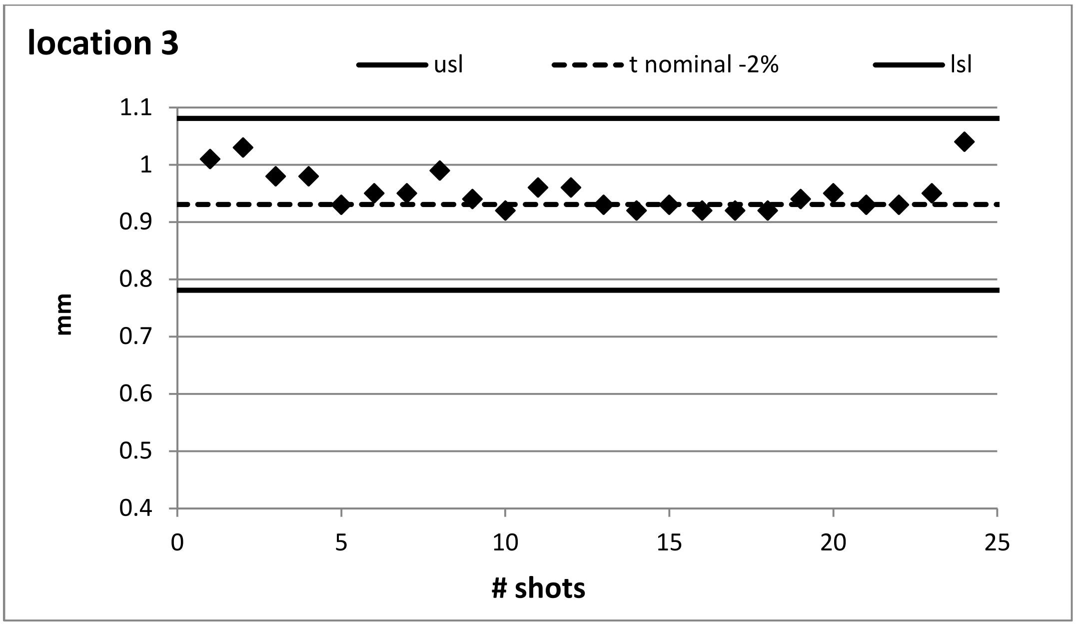


Figure 9: Parts dimension variation in location 3 – thin wall thickness

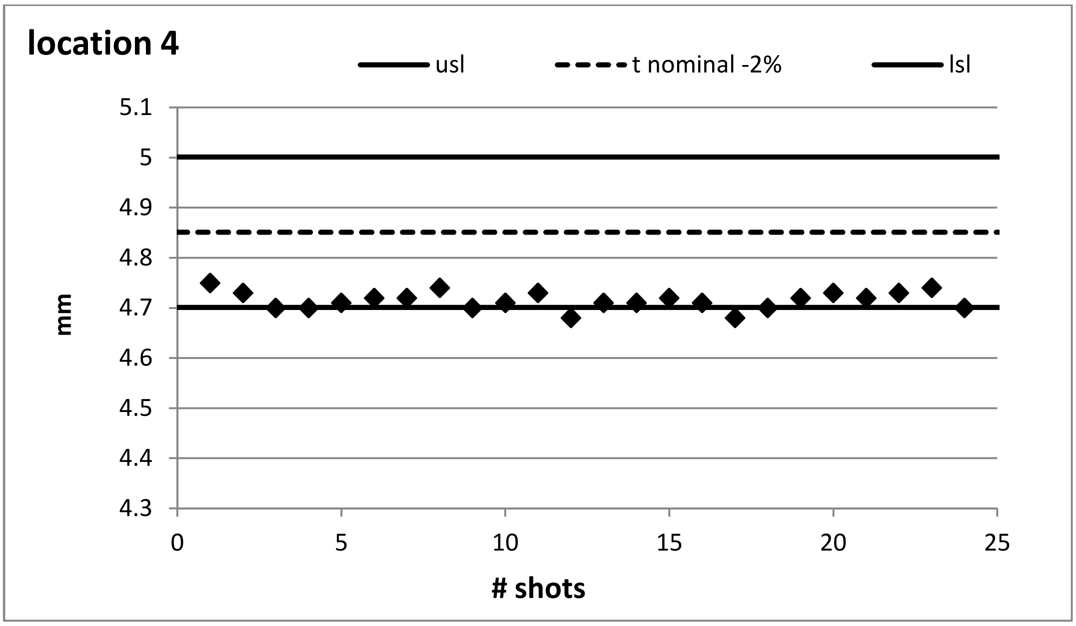


Figure 10: Parts dimension variation in location 4 – thick wall thickness

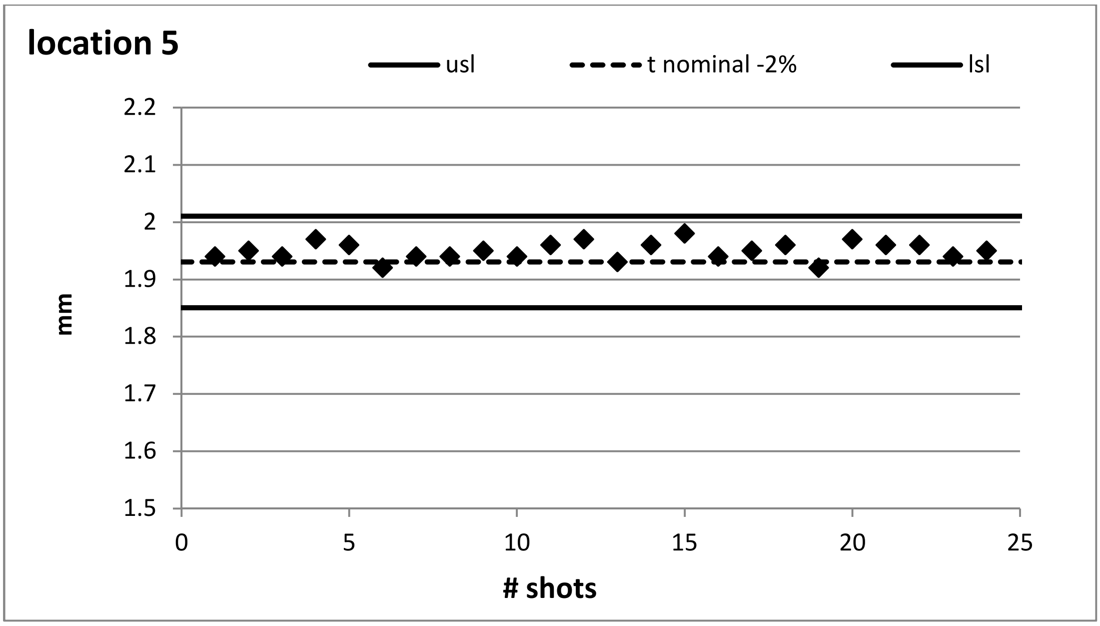


Figure 11: Parts dimension variation in location 5 – boss's diameter

The results show that part dimensions are stable throughout, with a maximum deviation of about 0.1mm. Thus, it can be concluded that the insert dimensions were not significantly distorted with repeating molding cycles. Moreover, it can be concluded that the printed inserts are able to produce molded parts with dimensions that are well within the spec limits according to typical PP commercial tolerances.