JOURNAL OF MODERN MANUFACTURING SYSTEMS AND TECHNOLOGY
E-ISSN: 2636-9575
VOLUME 8, ISSUE 1, MARCH, 2024, 8 – 17
DOI: https://doi.org/10.15282/jmmst.v8i1.11056

RESEARCH ARTICLE

3D Printed Mold Insert Infill Analysis for Injection Molding Application

Nur Syamimi Mohd Salleh[1], Ahmad Rosli Abdul Manaf[1*], Muhammad Syahir bin Ahmad[1,2,]

[1] Faculty of Manufacturing and Mechatronic Engineering Technology, University Malaysia Pahang Al-Sultan Abdullah, 26600 Pahang, Malaysia.
[2] Faculty of Manufacturing Engineering Technology, University College TATI, Teluk Kalong, 24000 Kemaman, Terengganu, Malaysia

ABSTRACT - Injection molding (IM) normally made from steel, such as STAVAX because of its ability to withstand molding forces such as clamping, injection, and holding force. Beside of this ability, the fabrication of steel insert requires machining as such CNC machining and electro discharge machining (EDM). In contrast, a 3D printed mold insert can overcome of these constraints as it can be printed in less time and cost compared to conventional method of insert fabrication. In this research, a mold insert is fabricated using a 3D printer with a key chain shape cavity. The 3D printed insert are printed using fused deposition modelling (FDM) 3D printer with a three different infill, namely 50%, 75%, and 100% infill percentage. This could determine the performance of the 3D printed insert with the infill percentage. After several test conducted at injection molding, it is found that the infill percentage could improve the insert life span. The 100% infill contributed to longer life of the insert compared to 50% and 75% infill percentage. As the molten polymer is injected into the mold, the polymer tends to fill the void of the insert with the infill percentage of 50% and 75%. As the insert with 100% infill is used, the void is eliminated, thus the cavity can be filled with the molten polymer efficiently. From this research, the capabilities of 3D printed mold insert with different infill percentage is determined.

ARTICLE HISTORY

Received	:	26th Oct. 2023
Revised	:	30th Jan. 2024
Accepted	:	22nd Feb. 2024
Published	:	30th Mar. 2024

KEYWORDS

Injection Molding
Additive manufacturing
3D printed insert

1.0 INTRODUCTION

The injection molding (IM) process is a widely used manufacturing process to form a wide range of frequently used items, as such bottle caps as well as remote control casings, needles, and many more. It is also extensively used in the manufacturing of bigger items like vehicle body panels, i.e. dashboard and car bumper. When producing dozens or millions of identical pieces from a mold, injection molding is typically employed as it can produce the part in less time and less expensive.

During the IM process, the molten polymer is injected into a mold, where it fills the cavity impression and duplicates the shape of the cavity under a controlled parameter such as injection pressure and holding time. The mold is then opened when the polymer cools, allowing the final injected item to be expelled from the mold. Steel mold inserts, similar to STAVAX, are employed in mass production of polymer products because they can withstand the molding force example clamping, injection, and holding force. These materials offer a good balance of machinability, strength and can withstand high temperature environments.

Besides the advantages of the metal inserts, fabricating the insert is costly owing to the tooling and machining cost. As the insert is the most complicated part of the mold components, the metal insert must go through a few machining processes such as CNC machining, grinding, drilling and many other processes according to the design itself. In addition, with the labor cost, its manufacturing and development of metal insert can cost thousands of dollars and take weeks to months. This could also affect the release of the part produced to the market.

On the road to circumvent this problem, additive manufacturing (AM) was introduced, and 3D printing is one of the processes that fall under AM. A computer controlled process of 3D printing compromises Stereolithography (SLA), Selective Laser Sintering (SLS), and Fused Deposition Modeling (FDM), in which the part is printed layer by layer. Technology is given bigger attention nowadays as it can fabricate the part accurately, less expensive and less time. Many polymer materials can be used with the printer including a high temperature material and creates products with a range of sizes, shapes, rigidity, and color utilizing various materials, such as polymers, composites, or biomaterials. This could be utilized to fabricate the mold insert. The use of 3D printed mold components or inserts is a beneficial method for creating new products.

In this research, the keychain shape cavity mold insert will be 3D printed with different infill parameters. The 3D printing infill was set to 50%, 75%, and 100% to balance material usage, print time, and strength, with higher infill percentages providing greater structural integrity at the cost of more material and longer printing times. The insert is

***CORRESPONDING AUTHOR | AHMAD ROSLI** | ✉ arosli@umpsa.edu.my

then will be assembled with the steel mold base. The mold with the 3D printed insert will be tested with a series of experiments. From the experiment, further analysis of the insert condition will be conducted to identify whether it is capable of withstanding the IM process. This study will suggest the infill parameters of the 3D printed mold insert and its capabilities in IM process.

2.0 LITERATURE REVIEW

Injection molding (IM) is a well-known polymer made part fabrication process [1]. It is widely used in many industries i.e., automotive, electrical and electronics, aerospace, and many more. Its popularity is due to the process capable of mass producing polymer-made parts as well as being able to maintain repeatability and accuracy [2]. In the IM process, the polymer resin is heated and melted inside the injection molding machine barrel at a controlled temperature [3]. Next, the molten polymer is injected into the mold until it fills the impression. This is also done under a controlled parameter, such as the injection pressure and injection speed [4]. The curing stage will come next after the injection process in which the molten polymer inside the mold is cooled down with the aid of the mold cooling channel. The final stage is where the mold is open, and the part ejected from the mold.

An injection mold consists of several steel plates and components which are assembled to form a complete mold assembly [5]. The final shape of the part injected using the mold depends on one of the mold components namely the mold insert. The insert required a special material as it received heat and pressure during the IM process. Due to these reasons, the insert is normally made from steel such as P20 and STAVAX. STAVAX for example possesses characteristics of good machinability and hardenability. It also can be polished up to a mirror finish. However, it has a drawback in that the insert fabrication required a good tooling and machining process. This could contribute to the cost [6][7][8]. Plus, the insert fabrication also is time-consuming as it requires a machining process as such milling, turning, drilling as well as CNC machining [9].

A 3D printed prototype mold insert can be exploited to represent a conventional steel insert [10]. It could eliminate the above-mentioned process. Using AM, this insert could be fabricated using 3D printing to replicate the part shape during IM [11][12]. The part shape can be from a simple part up to very complicated structures. This could save the insert fabrication cost using steel. However, lower part volume can be expected using this 3D printed insert. To overcome this, in this method of insert fabrication, a polymer with a higher melting temperature compared to the part material to be injected is used [13]. This could make sure the molten temperature of the material to be injected has minimum effects to the 3D printed insert.

Currently, 3D printing receives excessive attention [14]. This is due to the fact that 3D printing can create a part that is functional, repeatable, and also accurate. In 3D printing, a molten polymer is extruded from the nozzle of rapid prototyping (RP) machine [15]. A thin polymer layer will be created as the nozzle head will travel in x and y directions. This will complete the width and length of the printed part. The height of the part is depending on the printed layer to which each layer was bonded during the movement of the z-axis. The surface roughness of the printed part depends on parameters such as the nozzle diameter, temperature, and traveling speed. Beside the advantages of 3D printed insert, the effects of the infill percentage of the 3D printed insert is less reported. Gohn et.al [10] printed the mold insert using this method and found some voids in the insert causing some defects in the injected part.

3.0 EXPERIMENTAL PROCEDURES

3.1 Insert design & fabrication

In this research, a single cavity mold insert is design using a 3D modelling software, namely Autodesk Inventor. The insert dimension is set to 130 mm length and 70 mm width, while the thickness of the insert is 30 mm. The insert is also included with a feeding system as such the sprue, runner, and gate. The runner is set to a half round runner with diameter of 4 mm, while the branch runner is 3 mm. A rectangular is used to connect the branch runner to the cavity and is set to 1 mm width and 0.5 mm depth. The used of double gating system is to ensure the filling of the impression can be done faster compared to a single gate. The insert is illustrated in Figure 1.

In this initial research, no cooling channels were incorporated into the mold design. This decision was made because the primary goal of the experiment was to investigate the effect of the insert's infill. Additionally, cooling channels would not benefit the 3D-printed insert, as the insert itself is unable to withstand the high temperatures of molten polymer.

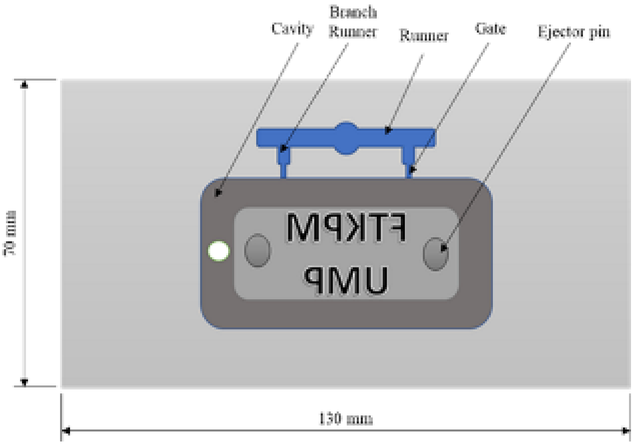


Figure 1. Schematic diagram mold insert.

The insert is fabricated using a 3D printing method. The 3D printer that is used is Ender Creality V1 with maximum build area of 330mm x 240mm x 300mm. The printer is capable of printing a wide range of material as such PLA, Nylon, Polypropylene, Polycarbonate, TPU, Tough PLA, and ABS. The technology used is fused deposition modelling (FDM). A default nozzle of 0.4 mm was used for printing. By using Ultimaker Cura slicer software, the layer height is set to 0.2 mm and 0.4 mm for the line width, while the printing pattern is set to grid pattern. The infill is set to 50%, 75%, and 100% respectively. These infill parameters are essential for reducing material consumption during insert printing and for preventing deformation caused by the weight of a fully solid insert. Consequently, the infill was set to 25%, varying for each insert. However, if the infill is set to less than 50%, it could compromise the insert's strength. Meanwhile, the printing temperature is set to 230°C. Other parameters were set to default as recommended by the software. The fabricated insert is shown in Figure 2.

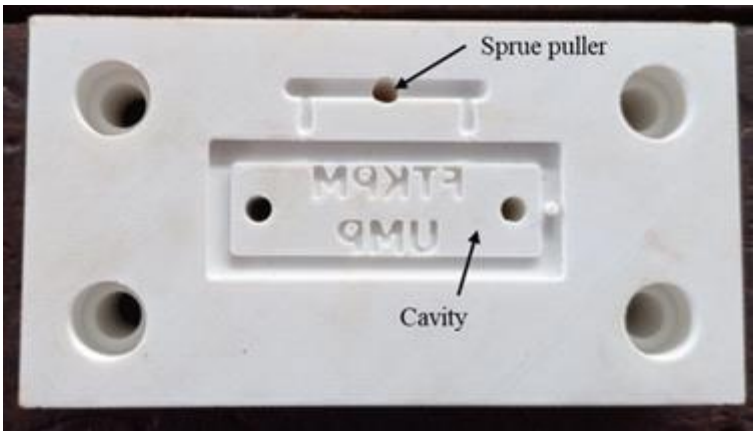


Figure 2. 3D printed mold insert

3.2 Material insert & specimen

As this research focused on filling analysis effect on insert performance, Polylactic Acid (PLA) was used for the insert material. The filament diameter was 1.75 mm. The filament is biodegradable under certain conditions with high heat capacity and high mechanical strength. It can be melted without significant damage and does not emit toxins or fumes. The mechanical properties of PLA are shown in Table 1.

Table 1. Properties of PLA filament

Material properties	Value
Melting temperature	170 – 230 °C
Glass transition temperature	56 – 64 °C
Density	1.24 g/cm^3
Young’s Modulus	3100 MPa
Tensile strength	600 MPa

Polypropylene (PP) was chosen as the specimen. This material was chosen due to its lower melting temperature compared to PLA. Before using this material, it is necessary to become acquainted with its properties. This Table 2 represents the properties of PP. Polypropylene, which is typically available in pellet form, is an excellent plastic for injection molding. Despite its semi-crystalline nature, polypropylene is easy to mold and flows well due to its low melt viscosity.

Table 2. Properties of PP

Material properties	Value
Melting temperature	130 - 170 °C
Thermal expansion	5.8 - 10
Density	0.91 – 0.94 g/cm^3
Tensile strength	3200 – 5000 Psi
Specific volume	30.4 – 30.8

3.3 Injection molding machine & parameters

In this research, a PNX 60 Nissei injection molding machine electric servo drive will be utilized. The parameter setup for the injection molding is recommended by the manufacturer for polypropylene (PP) material as not much focus is given on the injection molding parameters. The main consideration during injection molding process is only 100% cavity filling. The parameters of the injection molding are shown in Table 3

Table 3. Injection molding parameters

Material properties	Value
Front zone temperature	170 °C
Mold temperature	50 °C
Clamping force	380 kN
Injection pressure	5.142 MPa
Shot capacity	50 grams

4.0 RESULTS AND DISCUSSION

4.1 Insert condition

The experiment is conducted with the first insert of 50% infill. Figure 3 shows the result of the inserts after a series of injection molding trials. The insert with infill of 50% was only able to withstand the injection molding shot for 3 times. From the image, it can be seen that the molten polymer is penetrating the sprue puller hole. The penetration might be started by sprue hole wall breaking, in which the molten polymer liquidizes the insert and enters the void of the insert. The further polymer penetrates into the void of the insert, it makes the molten polymer become unable to flow and filling the cavity. This phenomenon is due to the infill of 50% of that insert. The insert is considered in a porous state; thus, the molten polymer enters the porous area.

The temperature of the molten polymer from the injection nozzle is approximately 170°C, which is close to the temperature used to print the 3D-printed insert. While steel can withstand the heat of the molten polymer during injection, the polymer-based insert material is susceptible to melting under these conditions [16]. This issue is exacerbated by the presence of numerous voids in the insert, which allow the molten polymer to penetrate and further damage the structure.

Figure 3. Insert condition after testing (50% infill)

The experiments continued with testing on insert of 75% infill. The first injection shot looks promising, however by the fifth shot, it had started to melt on the cavity area. Figure 4 shows the melted portion of the feeding system area and the "FTKPM" embossing. Due to the initial molten polymer injection to that position, the gate is completely damaged. The embossing is also affected by the wording. Note that this occurs as a result of the molten polymer passing through the cavity during a heated period as shown in Figure 4. Instead of the wording "UMP" underneath, the region wording "FTKPM" melts clearly. This phenomenon is due to the temperature of the molten polymer is higher in that region.

Figure 4. Insert with infill of 75% showing melting area

The third 3D printed insert with 100% infill was tested for a few shots. This insert is capable of withstanding a number of shots. However, after the fifteenth trial, the damage to the insert in Figure 5 is obvious, but it is still evident that it is melting despite the temperature being lowered. This insert is capable of withstanding a number of shots due to the nature of the insert in which it is solid printed without any void. The damage continues as no cooling channel was added to the mold and insert.

Figure 5. Insert fully melted in cavity area (100% infill)

4.2 Surface roughness mold insert

To further understand the effect of the injection molding process on the insert, the microstructures of the surface was also investigated and measured by using Olympus 3D Laser Confocal Microscope. Figure 6 shows the microstructure image of insert of 75% infill before and after the injection molding process. From the image in Figure 6(a), the layer of the insert is clearly visible, and the line shows the printed layer. The surface becomes smooth after in injection molding as shown in Figure 6(b). The surface roughness of the insert before testing is 18.08 µm, while it measured at 6.68 µm after testing. As was already established, this insert's surface roughness value was higher (rougher) before testing than it was after. It can be seen printed layer on surface already disappear. The effect of the melting situation is the cause printed layer been eliminated. The state of the insert surface is evident from Figure 6b itself and is substantially different from previously.

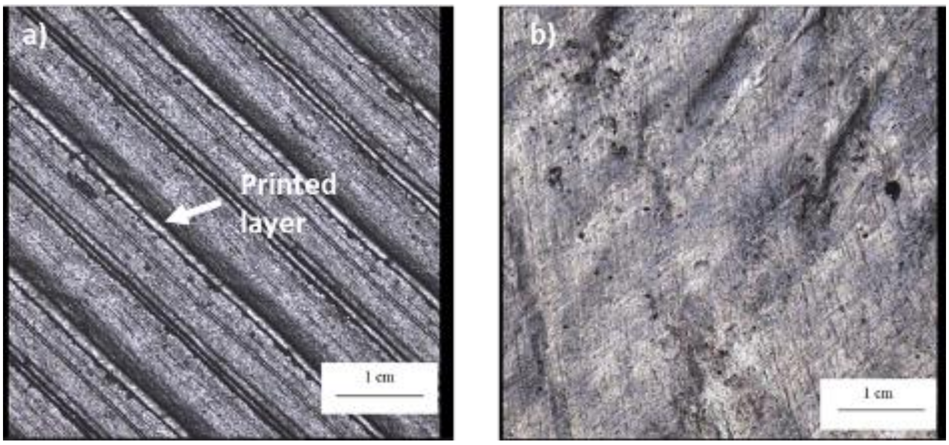
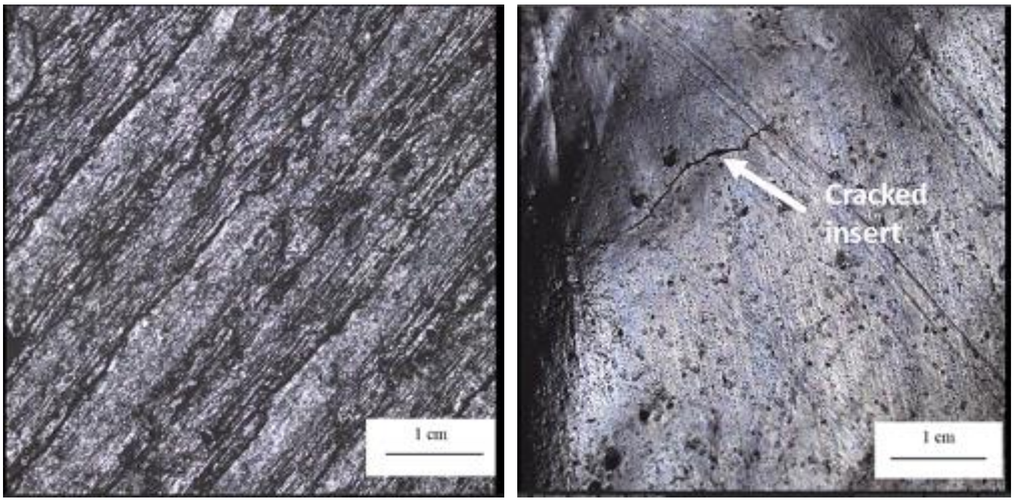


Figure 6. (a) Microstructure image for insert infill 75%, (b) Insert condition under microstructure showing the printed layers after testing.

The same phenomenon observed on 3D printed insert of 100% infill. The surface roughness measured before the testing is 10.06 µm. Figure 7a shows microstructure image of insert. From the image, it shows less layers of the insert compared to the previous image of Figure 6(a). Note that if need a smooth surface, tend to increase the number of outside layers or shells.

As can be seen in Figure 7(b) after the testing, there appears to be a cracking area that may have been caused by stress from the molten polymer get through the cavity. Additionally, seem like a hole as a result of a few failures making it less porous. It became impacted once the insert began to melt. Comparing the surface roughness of the infill surfaces before testing, 100% infill often has smoother surfaces than 75% infill insert. However, the main determinant of smooth surface roughness comes from 3D printer settings like layer height and the number of contours.



7a 7b

Figure 7. (a) Microstructure image of insert, (b) Image at micro level for insert infill 100% after testing

4.3 Injected part evaluation

The injected part using 3D printed insert was also evaluated in this research. However, no injected part from the insert of 50% infill due to the insert failure during the first shot. Four samples from various tests for insert infill 75% are shown in Figure 8 below. From the samples, it shows that the embossing was only clearly seen during the first shot. The FTKPM embossing started to erode when the injection molding continues. For this insert, it is at least reasonable to estimate that 15 keychains can be injection molded before the insert as a whole sustains entirely irreparable damage. The wording (FTKPM) on a keychain becomes less visible from the fifth trial to the fifteenth trial, as well as described above in insert condition insert infill 75%.

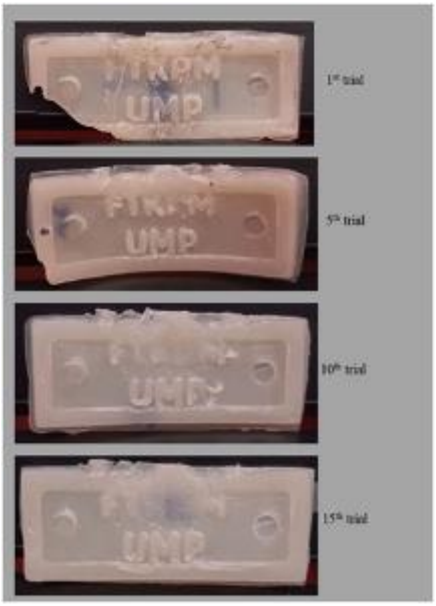


Figure 8. Keychain specimen after injected molten polymer.

Hence The samples from 100% insert infill are shown in Figure 9 below. The samples do not emerge as clearly as the sample in Figure 8 above following injection, as was previously indicated regarding the falling melt temperature for this insert. The wording cannot be read clearly as by means. This is due to compression during injection, which causes the molten polymer to decompress when the temperature drops, and the polymer does not melt as well.

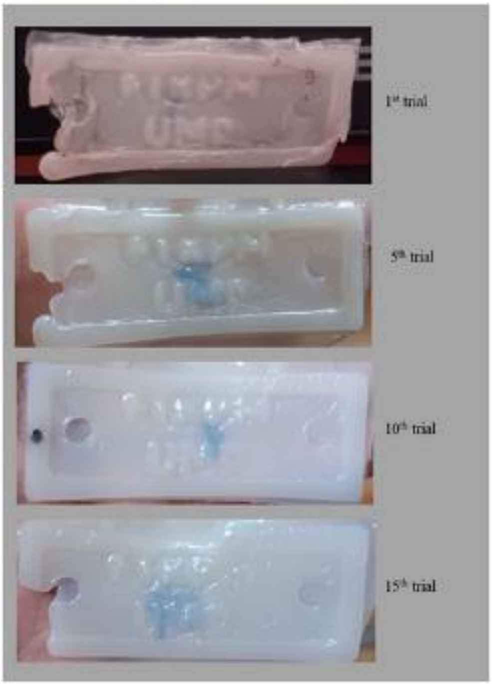


Figure 9. The wording in this keychain specimen is less visible.

4.4 Dimensional accuracy sample part

Throughout the test, each sample was taken dimension for reference on deformation of sample after injection molding process. Figure 10 below shows the actual dimension for every part of the sample. It shows the actual size of the keychain specimen and the real dimension for inert infill 100% and 75% infill. It was shown that the sample dimension decreased throughout the 15 tests performed. One of the reasons is due to the melt temperature and permanent damage on the insert itself. Based on Figure 11 and Figure 12 showing the interpretation data dimension to graph for both dimensions.

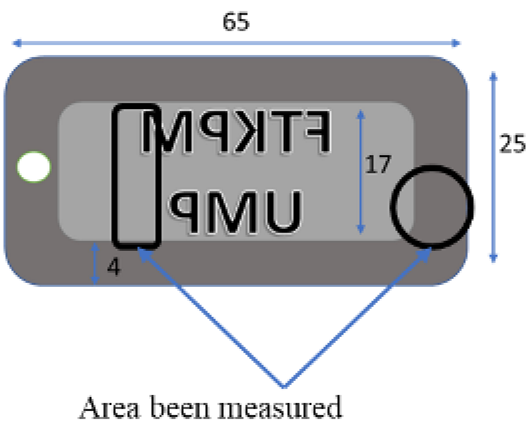


Figure 10. Actual dimension for keychain specimen

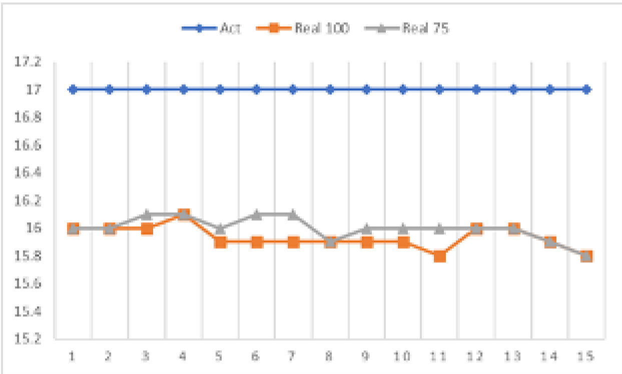


Figure 11. Graph dimension 17

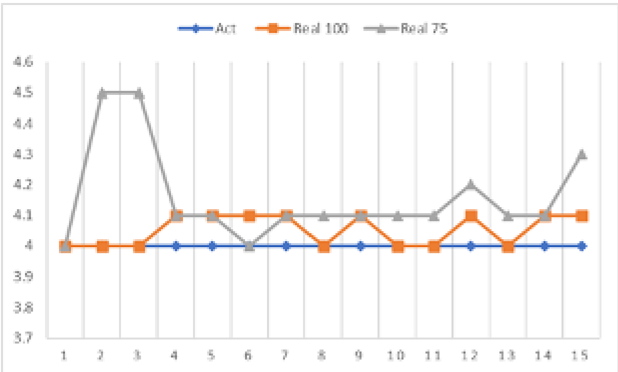


Figure 12. Graph dimension 4

The least imperfection between these two dimensions can be noticed in dimension 4 rather than dimension 17. In this instance, it's most likely because of how it is positioned. The area is not merely affected since it is far from the feeding system, according to the actual dimension of 4 mm in Figure 10. As was already indicated, the feeding system region is particularly susceptible to melting, but the molten polymer still manages to pass through the cavity area. Typically, during the injection molding process, plastics might shrink by 20% to 25%. Thermal contraction, which also affects semi-crystalline and amorphous polymers like polypropylene, is what causes volumetric shrinkage.

5.0 CONCLUSION

This paper examines the impact of 3D-printed infill on the injection molding process. The research shows that the molten polymer from the injection molding machine is damaging the insert. This occurs because the melting temperature of the polypropylene (PP) is nearly the same as that of the 3D-printed insert. The insert is unable to withstand the heat and pressure during the process. The issue is exacerbated when the insert is printed with 50% infill, as the molten polymer finds and enters voids in the insert. While a 75% infill provides better results, the insert should ideally be printed with 100% infill, leaving no voids, to ensure that the molten polymer properly fills the mold impression. Although the insert can endure a few injection cycles, its melting becomes a concern. Even though a fully melted 100% infill insert causes less damage to the mold cavity compared to a 75% infill insert, a material with a higher melting point is recommended for the insert. Additionally, applying a coating could enhance the insert's durability.

6.0 ACKNOWLEDGEMENTS

The authors would like to thank the University Malaysia Pahang Al-Sultan Abdullah for providing financial support under University Internal Grant Scheme No. Reference RDU220360 as well as laboratory facilities to perform this project.

7.0 REFERENCES

[1] R. Injection and I. M. Process, "14 Injection Molding," pp. 241–254, 2021, doi: 10.1016/B978-0-12-818008-2.00019-2.

[2] S. E. M. Selke, J. D. Culter, R. A. Auras, and M. Rabnawaz, "Injection Molding, Closures, Rotational Molding, Compression Molding, and Tubes," Plast.Packag., pp. 293–308, 2021, doi: 10.3139/9781569908235.011.

[3] R. Farooque, M. Asjad, and S. J. A. Rizvi, "A current state of art applied to injection moulding manufacturing process - A review," Mater. Today Proc.,vol. 43, pp. 441–446, 2020, doi: 10.1016/j.matpr.2020.11.967.

[4] S. Kashyap and D. Datta, "Process parameter optimization of plastic injection molding: a review," Int. J. Plast. Technol., vol. 19, no. 1, pp. 1–18,2015, doi: 10.1007/s12588-015-9115-2.

[5] D. G. Baird, "Polymer Processing," Encycl. Phys. Sci. Technol., pp. 611–643, Jan. 2003, doi: 10.1016/B0-12-227410-5/00593-7.

[6] C. Poli, "Injection Molding: Relative Tooling Cost," Des. Manuf., pp. 39–74, 2001, doi: 10.1016/b978-075067341-9.50008-2.

[7] E. R. Larson, Material Selection Based on Cost. 2015. doi: 10.1016/b978-0-323-31299-8.00006-4.

[8] C. Poli, "Injection Molding: Total Relative Part Cost," Des. Manuf., vol. 75, pp. 75–114, 2001, doi: 10.1016/b978-075067341-9.50009-4.

[9] P. Peças, I. Ribeiro, E. Henriques, and A. Raposo, Additive manufacturing in injection molds-life cycle engineering for technology selection. Elsevier Inc., 2018. doi: 10.1016/B978-0-08-102414-0.00004-5.

[10] A. M. Gohn, D. Brown, G. Mendis, S. Forster, N. Rudd, and M. Giles, "Mold inserts for injection molding prototype applications fabricated via material extrusion additive manufacturing," Addit. Manuf., vol. 51, no. January, p. 102595, 2022, doi: 10.1016/j.addma.2022.102595.

[11] D. Kazmer, 28 Three-Dimensional Printing of Plastics, Second Edi. Elsevier Inc. doi: 10.1016/B978-0-323-39040-8/00029-8.

[12] Nagahanumaiah and B. Ravi, "Indirect Rapid Tooling," Compr. Mater. Process., vol. 10, pp. 345–373, Jan. 2014, doi: 10.1016/B978-0-08-096532-1.01014-1.

[13] M. Biron, Plastics Overview. 2020. doi: 10.1016/b978-0-12-821539-5.00002-1.

[14] M. Gurr, "Rapid Prototyping," Ref. Modul. Mater. Sci. Mater. Eng., pp. 1–27, Jan. 2016, doi: 10.1016/B978-0-12-803581-8.01477-6.

[15] F. Biondani, G. Bissacco, P. T. Tang, and H. N. Hansen, "Additive Manufacturing of Mould Inserts with Mirror-like Surfaces," Procedia CIRP, vol. 68, no. April, pp. 369–374, 2018, doi: 10.1016/j.procir.2017.12.097.

[16] Moritz, V.F.; Bezerra, G.S.N.; Hopkins Jnr, M.; Fuenmayor, E.; Günbay, S.; Hayes, C.; Lyons, J.G.; Devine, D.M. Heat Dissipation Plays Critical Role for Longevity of Polymer-Based 3D-Printed Inserts for Plastics Injection Moulding. J. Manuf. Mater. Process. 2022, 6, 117. https://doi.org/10.3390/jmmp6050117