KEMENTERIAN PENDIDIKAN MALAYSIA JOJAPS

eISSN 2504-8457

POLITEKNIK MALAYSIA PORT DICKSON

Journal Online Jaringan Pengajian Seni Bina (JOJAPS)

# Base Thin Wall Square Products Shrinkage Due to Cooling Temperature in Injection Molding Process

Dahmir Dahlan[1], Widodo Puji[2] & Arif Riyadi TK[3]

*[1]Professor on Pancasila University, [2]Technical Ass Manager on Sanpak Unggul Industri & [3]Lecturer on Pancasila University*

**Abstract**

Shrinkage on injection molding process is a serius problem that can cause the products damage. Shrinkage fenomena can be minimized by having a good cooling temperature. Base Thin Wall Square Products also have a shrinkage fenomena. It causes many Base Thin wall Square Products out of quality standart. In this study, we try to find the cooling temperature suitable temperature of Base Thin Wall Square Products. As the result of this study, we find that the best temperature of cooling temperature is about $15^0$C – $17^0$C.



*Key-word:* - Shrinkage, Plastic Injection, Injection Molding Process.

## 1.0 INTRODUCTION

Upper Thin Wall Square making process that uses Injection Molding Technique in Sanpak Unggul industry encounters obstacle due to a lot of fail products (out of quality standard products).

**Picture 1.** Upper Thinwall Square Product (Sanpak Unggul Industry)

The production failure occurs due to shrinkage defect in the cooling process. Regarding to the most failure products are caused by this shrinkage, the research on the effect of cooling temperature temperature in the Injection Molding Process is carried out. It is to determine the effect of cooling temperature in the basic product shrinkage. The shrinkage in Injection Moulding Process is functioned to keep the product from the mold that can cause shrinkage defect or out of specification products.

## 2.0 LITERATURE REVIEW

Anggono (2005) in the Mechanical Engineering Journal stated that plastic injection is a manufacture process to make a plastic base product or polypropylene. The process often makes defect products such as shrinkage, cracking, unsuitable dimensions, and damage when the products come out of the mold, also resulting in a lot of material waste. Defect products can be caused by some factors but the most important is shrinkage or material reduction. Firdaus (2002) in the Mechanical Engineering Journal stated that shrinkage defect accurs when the melting temperature is too high. This defect can be reduced by designing the correct and proper parameters. Moerbani (1999) stated that in every mold making, we should always count the occurrence of shrinkage after cooling temperature and after the products get out of the mold cavity. It happens because of the phase changes from the liquid material to solid that will make volume change. The raw material that is used to produce Upper Thin Wall Square is polyprophylene (PP) plastic. This material is included in the thermoplastic group. PP material is one of the derivative types of polymer. Polyprophylene (PP) raw material is obtained by decomposing petroleum (naphthan) by giving hydrogen gas petroleum at oil and natural gas splitting.

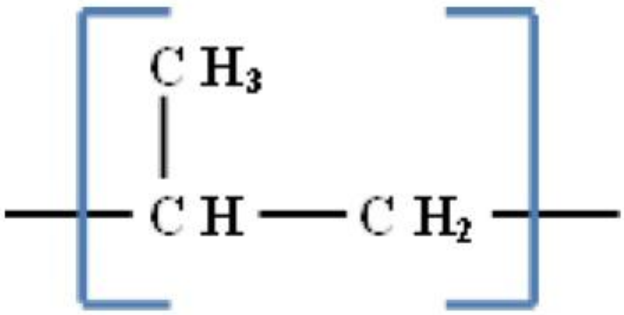

**Figure 2.** Polypropylene molecule (PP)

On the prints, the translucent properties of PP are better than polyethylene (PE) with glossy surfaces, shrinkage in small prints, and it has a better appearance and dimensional accuracy also. The mechanical properties can be improved by mixing glass fiber. Thermal expansion can be provided by adding thermosetting resin.

## 3.0 INJECTION MOLDING PROCESS

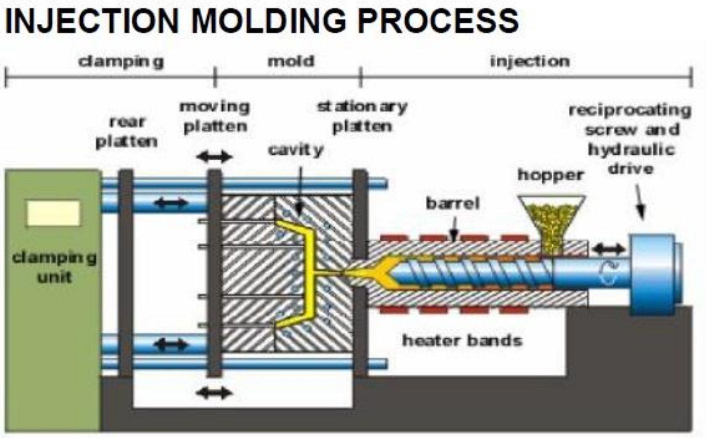


**Figure 3.** Injection Machine (*http://www.oke.or.id)*

There are six important steps on molding process.
1. Clamping
Each injection molding machine consists of three basic equipments. They are molding unit, clamping unit, and injection unit. Clamping unit is used to maintain the mold under pressure during the injection and cooling process. Basicly, the clamping serves to withstand two parts of the mold from injection molding, simultaneously. During the injection process, calmping is used to withstand the force and press and discharge the finished object from the mold.
2. Injection
Prior to injection, the plastic material is still in the form of powder granules. Then, the material in the granular form is inserted into the hopper of the injection unit. The plastic material is further inserted into a heated cylinder to be melted. Then the cylinder works with motorized screw that serves to mix and stir the melted plastic material and press it to the end of the cylinder. Once the material is enough, it will be accumulated at the end of the screw so the injection process works. The plastic material which is already at the end of the screw will be injected into the mold through sprue bushings. The pressure and speed of the injection during the process are controlled by the screw.
3. Dwelling
Dwelling is a temporary halt of the injection process. Plastic material that has been injected into the mold by giving a certain pressure must be ensured to fill in all parts of cavity. This process is to avoid any defects of products due to porous or weld.
4. Cooling
Plastic material that has been filling the mold and molding the objects according to the mold, then cooled with a certain temperature so that the plastic material quickly becomes solid or hardened.
5. Mold Opening
The material that has been hardened after cooled then becomes a finished object. The two molds are opened by clamping plate and setting plate intermeaidries.
6. Ejection
The final step is to remove the finished object from the mold so that the next injection process can be done. In the ejection step, certain molding designs are used to cut runner and sprue from plastic materials. Thus, the molding object does not need to be done in further work (runner cutting and sprue). For certain purposes and designs, sometimes runners and sprue are not cut directly in the ejection process. It requires additional process for cutting runners and sprue. To print the next product together, do the same steps as before until the desired quantity (quantity) of the product is obtained.

## 4.0 MOLD COOLING

PP palstic raw material reaches melting point at 1760C. PP material can be injected into the mold provided if the material is melted. The melting point temperature of the PP material will make a hot propagation to the mold material. If it happens continuously without any cooling process then one time PP plastic material will burn in the mold. So it has become a requirement of cooling process on molding plastic (injection mold) to get the product results in accordance with the quality that has been determined. After the hot plastic material sets into the mold, the mold should be cooled quickly. The cooling process is to maintain the molded part shape as desired when it is removed from the mold. The temperature of the mold is very important, so cooling process plays a role to control and maintain the required mold temperature. The typical intermediate cooling agents are air, water, and a mixture of glycol water/ethylene. The air refers to the heat emission from the mold. The water flows along the canals inside the mold to absorb heat. The water / ethylene glycol is used for extreme cooling and also flows along the canal to absorb heat. The cooling efficiency of a mold is determined by the cooling mechanism / method used, the coolant material / fluid flows rate, the coolant material / fluid temperature, and the type of coolant / fluid used.

There are many parameters in designing an appropriate cooling system for a mold, include:
1. Type of plastic material and melting temperature.
2. Cooling method / mechanism
3. Material / fluid cooling
4. Material flow rate / coolant fluid
5. Material temperature / fluid coolant.
6. Cooling location to be made.
7. Cooling area
8. The size, number, and length of the canal
9. Location of each canal
10. Volume of the cooling canal.

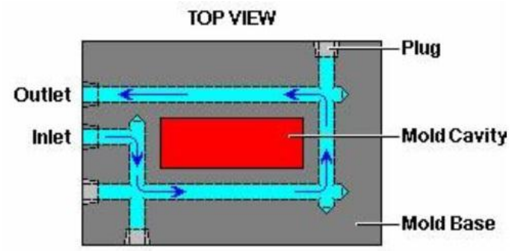


**Figure 4.** Variety of straight cooling channels *(D-M-E plastic_university)*

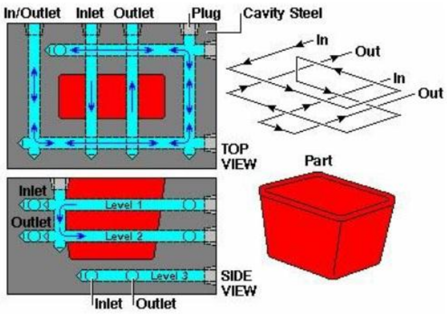


**Figure 5.** Cooling mold of straight cooling channels *(DM-E plastic_university)*

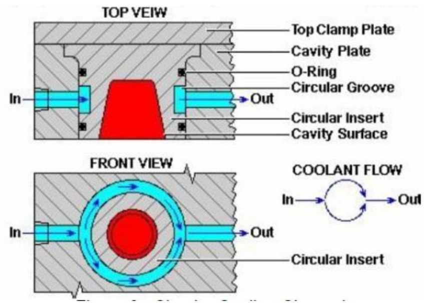


**Figure 6.** Circular Cooling Channels *(D-M-E plastic_university)*

## 5.0 COOLING SYSTEM DESIGN

The temperature control system of a mold must be able to distribute the same temperature in the mold and be able to remove heat from the cavity that filled with molding material.

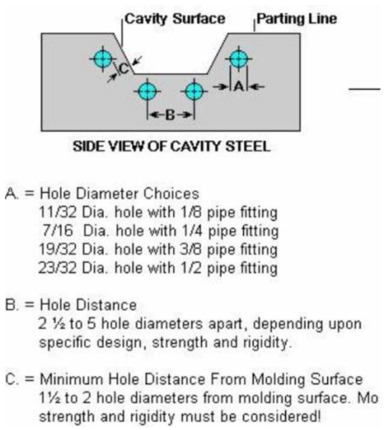


**Figure 7.** Cooling Cannals Size (D-M-E plastic_university)

**6.0 SHRINKAGE**

Shrinkage is a condition of deviation in the formation of plastic due to the cooling process that is not in accordance with the needs. The product making planner must count the existence of material shrinkage after the material or (workpiece) formed into a finished product. In the injection molding process, the heat treatment pressure will also affect the dimension of the product when it is compared to the size of the mold. Actual size of the product will be different, i.e. the size of the workpiece outside will be smaller than the cavity. The direction of material shrinkage leads to a reference point inside the workpiece, means, it can not take the field or line that is inside the workpiece. Figure 8 observes the shrinkage direction clearly. Figure 8. (a) shows the interpretation of the wrong shrinkage direction. The fact is the result of the holes in the workpiece is getting smaller instead of getting bigger. Thus, the observation reference is not a line or area or object inside the workpiece. Figure 8. (b) shows the correct interpretation of the shrinkage direction, i.e. that all points present in the workpiece will shrink down to the P reference point. Thus, the holes result in the workpiece will also shrink smaller towards P point.

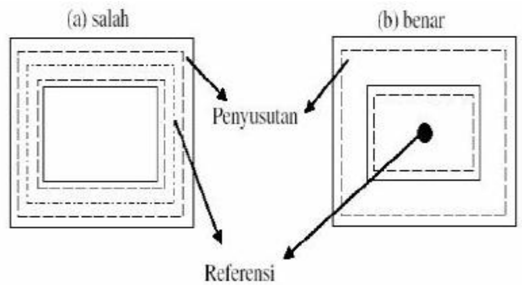

**Figure 8.** Shrink Direction (Moerbani, 1999)

Based on Kwon A. (2006), anisotropic depreciation is formulated:
The shrinkage formula of the product in the direction of the X axis
$S = (1-x)\ \alpha ax\ (Ts - T\infty) + x\alpha cx\ (Tm - T\infty) - \beta xPS - S\gamma\infty$ (1)
The shrinkage formula of the product in the direction of the Y axis
$Sy = (1-x)\ \alpha ay\ (Ts - T\infty) + x\alpha cy\ (Tm - T\infty) - \beta yPS - S\gamma\infty$ (2)
The shrinkage formula of the product in the direction of the Z axis
$Sx \cong Sv - (Sx - Sy)$ .(3)
Where the value of S, (4)

It should also be distinguished between axial shrinkage which occurs in the plastic flow direction and radial shrinkage which the flow is perpendicular to the flow direction. These two-way differences need to be considered as they can cause shrinkage defects. Therefore, in the modling injection design, each size of the workpiece mold must be made larger than the actual object. The actual object will be made by multiplying the actual size of the object by the shrikage factor in order to obtain the desired workpiece as expected. The shrinkage factor value is adjusted to the plastic material to be used, so that, the shrinkage value can be known in the shrinkage material table.

In fact, shrinkage is very difficult to be removed. We can minimize the occurrence possibility of shrinkage only. The effort that can be done is to perform cooling process in a proper cooling temperature during the injection molding production process. Based on these facts, it can be concluded that cooling temperature in the injection molding production process plays a very important role and can not be ignored. Proper cooling temperature will produce injection molding plastic production with minimal shrinkage. Thus, the product defect due to shrinkage can be minimized. It is very necessary to study the proper cooling temperature of the injection molding process continuously to obtain the improvement over time. The cooling temperature is a process to prevent the occurrence of a condition that the mechanism increases in temperature exceeding the maximum allowable temperature limit. If the maximum temperature limit conditions are reached or even exceeded then the mechanism is expressed in over-heating conditions. This condition is greatly avoided because it can cause damage, malfunction, fire, product defect, inficient, and other highly undesirable events. The cooling media used is water fluid which derived from a water chiller conditioning machine. This cooling system is integrated to the injection molding machine, which keeps the machine temperature, mold temperature, and plastic temperature (as raw material and product), as long as the machine is operated and the production process lasts until it is finished and the machine stops working.

In this research, we will conduct experiment to know the influence of cooling temperature to the dimension of plastic injection process. We conduct the direct survey at Sanpak Unggul Industrie. Shrinkage is one of the constraints that causes the high number of product defect, so that, the experiments are performed by varying the cooling temperature of the production process. From each variation of the temperature, we obtain the product 5 results that will be verified the real products dimension (size). Product verification involves measuring the dimensions of the length, width, and height of the thin wall square base product which are compared to the size of the mold.

## 7.0 RESEARCH METHODE

The parameter setting of the production process are as follows:

Product : Base Thinwall Square
Machine : JSW 180 AD-300 H
Color : Natural
Material : PP Titanpro 803
Weight : 32 gram
Injection Time : 0,23 seconds
Cooling temperature : 1,7 seconds
Interval : 0,2 seconds
Cycle time : 7,5 seconds
Clamping Force : 205 Ton
Temperature NH1 : 265ºC
Temperature H1 : 190ºC
Temperature H2 : 225ºC
Temperature H3 : 250ºC
Temperature H4 : 260ºC
Temperaure H5 : 265ºC
Runner Zone 1 : 200ºC
Runner Zone 2 : 192ºC
Runner Zone 3 : 205ºC
Runner Zone 4 : 216ºC
Cooling Mold : Chiller
Room Temperature: 30ºC

Variation of refrigeration used start at temperature 150C up to 320C, with increase of each variation is 0,50C.

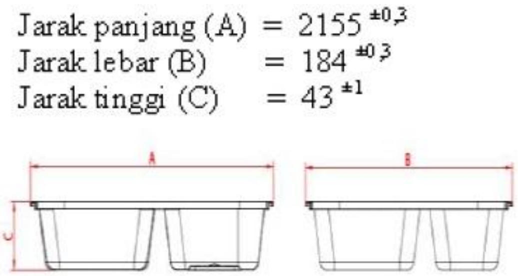


**Figure 9.** Spesifikasi Produk Base Thinwall Square

## 8.0 SHRINKAGE ANALYSIS

The shrinkage analysis of the product is taken from three directions: X axis (product length), Y axis (product width), and Z axis (product height). Measurements are made on the product as well as on the mold as a comparison. Table 1 is the average of histogram shrinkage differentioan in the direction of length, width, and height of product or X, Y, and Z axis. The histogram shows the average shrinkage in the long direction or X axis of the product is found to be different if the cooling temperature of the mold used is different. The lower the cooling temperature the higher of the depreciation presentation. The average of shrinkage is: 1.09% $\leq X \leq$ 1.31%.

In the direction of width or Y axis of the product, there is a difference if the cooling temperature of the mold used is different. The lower of the cooling temperature the higher of the depreciation presentation. The average of shrinkage is: 0.92% ≤ Y ≤ 1.25%. In the high direction or Z axis of the product, there is a difference if the cooling temperature of the mold used is different. The lower the cooling temperature the higher of the depreciation presentation. The average of the shrinkage is: 0.34% ≤ Z ≤ 1.46%.

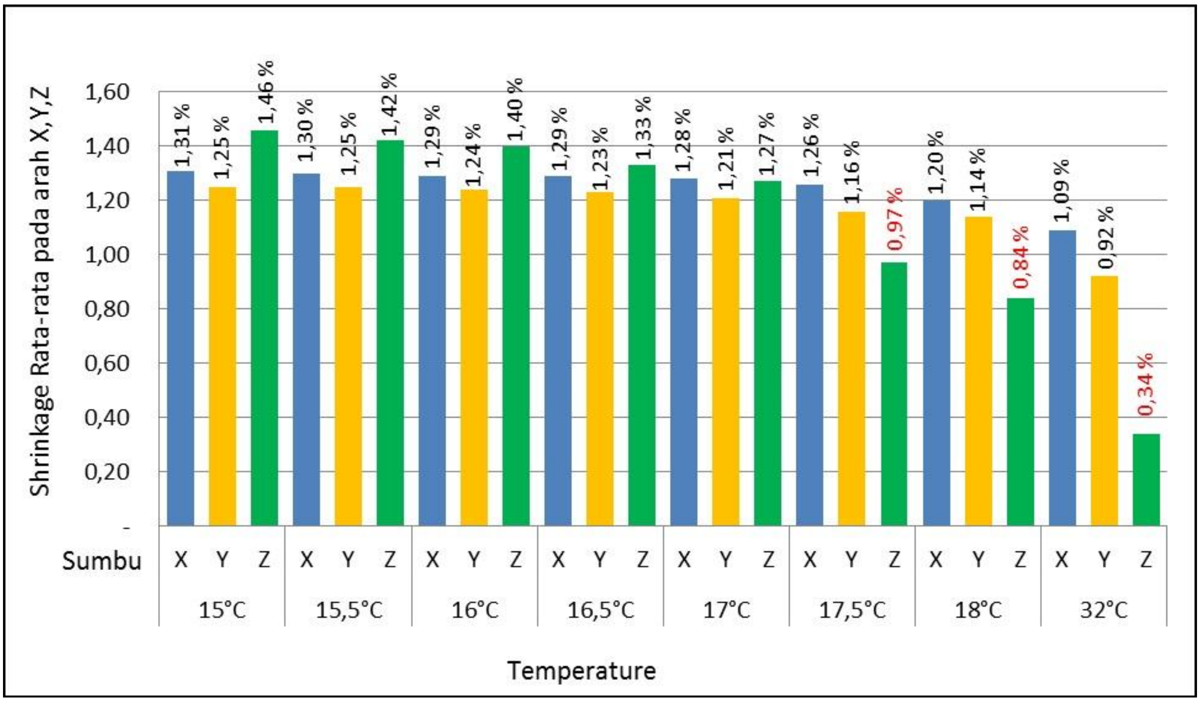


**Table 1** Shrinkage on product length of X, Y, Z axis on test injection molding with coolant chiller with temperature of 15˚C

## CONCLUSION

Based on the result of this study, we conclude that the coolant temperature suitable for the mold base is about 15˚C to 17˚C.

## RECOMMENDATION

The next research should use different plastic materials in order to know the shrinkage difference. This way, it can be concluded the best plastic material for thin wall square base products. Currently, the production of thin wall square mold base is recommended to Sunpak Unggul Industry to use temperatures of 16.5˚C up to 17˚C. With the working temperature, the minimum cooling load is allowed to be used in the production process by producing the product according to the desired quality. So that, the consumption of electrical energy used in the cooling process can be minimized and expected production efficiency can be done.

## REFERENCES

Akay, H. U., 2003, *Prediction of Shrinkagein Plastic Injected Parts Due to Cooling*, Computer-Aided Engineering Analysis.
Anggono. AD. 2005, Prediksi Shrinkage Untuk Menghindari Cacat Produk pada Plastic Injection, media Mesin Vol. 6 No. 2, Teknik Mesin UMS Surakarta.
Dominic V Rosato, P.E. and Donald V Rosato Ph.D, The Complette Molding
Operation, Technology, Perfomance, Economic, New York.
Firdaus, Tjitro, S., 2002, Studi Eksperimental Pengaruh Parameter Proses Pencetakan Bahan Plastik Terhadap Cacat Penyusutan (Shrinkage) pada Benda Cetak Pneumatic Holder, Jurnal Teknik Mesin Vol. 4 No. 2, Universitas Kristen Petra, Surabaya, 75-80.
Haihong Xu, John Wysocki, David Kazmer, 2003, *Shrinkage Study Of* Thermoformed P. Incroperaank P and De Witt, DavidP,1990, introduction to Head Transfer, 2nd Ed, Jhon Wiley and Sons, New York.

Jack G. Zhou and Zongyan He, 1998,*Rapid Pattern Based Powder Intering Technique And Related Shrinkage Control*, Department of Mechanical Engineering and Mechanics Drexel University, Journal of Materials and Design, Vol. 19, pp. 241-248.
Moerbani. J., 1999, Plastic Moulding, Jurnal Akademi Teknik Mesin Industri (ATMI), Surakarta.
Manesmann Demag, 1982, Injection Moulding Pocket Book, 6th Ed. Manesmann Demag Kunstoff Technic, Swiss.
Menges, George, 1986, How to Make Injection Mould 2nd ED, Hanser Publisher, Munic.
Virasantoto, A.T., 2008, Simulasi Mold Alat Circumsisi dengan Software Catia V5 Dan Mold Flow Plastigt Insight 5, Tugas Akhir S1, Universitas Muhammadiyah Surakarta, Surakarta