# SINK MARKS DEFECT ON INJECTION MOLDING USING DIFFERENT RAW MATERIALS

MOHD NAZRI BIN NOORDIN

Thesis submitted in fulfilment
of the requirements for the award
of the degree of
Bachelor of Mechanical Engineering with Manufacturing Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

**SUPERVISOR'S DECLARATION**

I hereby declare that I have checked this project report and in my opinion this project is satisfactory in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Manufacturing Engineering.

Signature:
Name of Supervisor: EN. ASNUL HADI BIN AHMAD
Position: LECTURER
Date:

## STUDENT'S DECLARATION

I hereby declare that the work in this report is my own except for quotations and summaries which have been duly acknowledged. The report has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature:
Name: MOHD NAZRI BIN NOORDIN
ID Number: ME06054
Date:

To my beloved parents and my siblings

## ACKNOWLEDGEMENTS

**In the name of Allah, the Most Gracious and Most Compassionate**

The author would like to thank Allah Almighty for blessing and giving him strength to accomplish this thesis. Specials thanks to his supervisor Encik Asnul Hadi bin Ahmad whose help, stimulating suggestions and encouragement him helped in all the time of research and writing of this thesis. The author grateful to him for his patience constructive comments that enriched this research project. His time and efforts has been a great contribution during the preparation of this thesis that cannot be forgotten forever.

Many thank to all of the technicians in Mechanical Laboratory, Faculty of Mechanical Engineering, for their cooperation and assisting him in the various laboratory tasks. The author like to express his sincere appreciation to all of his friends and colleagues in Universiti Malaysia Pahang for coloring his daily live and helped him in one-way or another.

Deepest gratitude to his parents, who give him a real love, pray, support, and all they have. Finally, the author grateful to Universiti Malaysia Pahang for all of support during the period of this research work.Thank you.

## ABSTRACT

The quality of an injection molded part is affected by material properties, mold geometry, process conditions and etc. Obtaining optimum parameters is the key problem to improve the part quality. Sink marks is one of several important flaws of injection molded parts. In this project, numerical simulation is combined with Taguchi design-of-experiment (DOE) technique to investigate the influence of parameters on sink marks of the injection molded part and optimization of parameter settings in injection molding process. The Acrylonitrile Butadiene Styrene (ABS) and polyethylene (PE) materials were used to analyze the sink marks. An orthogonal array based on the Taguchi's method and Analysis of Variance (ANOVA) was conducted to observe the sink marks of injection molded parts, and to allocate the significant of each factor that contribute to sink marks. Four factors were consisting of packing pressure, mold temperature, melt temperature and packing time. These factors were found to be the principal factors affecting the sink marks of the injection molded parts. It was found that optimum parameters for ABS material are packing pressure at 375 MPa, mold temperature at $40^0$c, melt temperature at $200^0$c and packing time and packing time 1s. While for PE material, the optimum parameter was found are packing pressure at 75 MPa ,mold temperature at $70^0$c, Melt temperature at $190^0$c and packing time at 1.5s.

## ABSTRAK

Kualiti pengacuan suntikan dipengaruhi oleh ciri-ciri bahan, acuan geometri, keadaan proses dan sebagainya. Mendapatkan parameter optimum adalah sukar untuk meningkatkan kualiti bahagian. Tanda sink adalah satu daripada beberapa kecacatan pada bahagian pengacuan suntikan. Dalam projek ini, simulasi digabungkan dengan kaedah Taguchi untuk menyiasat parameter yang mempengaruhi tanda sink pada bahagian dibentuk dan untuk mendapatkan parameter optimum dalam proses pengacuan suntikan. Susunan ortogon berdasarkan kaedah Taguchi telah dikendalikan untuk mengkaji tanda sink pada bahagian terbentuk, dan untuk mengkaji kepentingan setiap faktor yang menyumbang kepada tanda sink. Empat faktor tersebut ialah tekanan padatan, suhu acuan, suhu lebur dan masa padatan. Faktor-faktor ini telah didapati mempengaruhi tanda sink pada bahagian-bahagian pengacuan suntikan. Didapati parameter optimum untuk bahan ABS adalah tekanan padatan pada 375 MPa, suhu acuan pada $40^0$c, suhu lebur pada $200^0$c dan masa padatan 1s. Manakala untuk bahan PE, parameter optimum telah didapati adalah tekanan padatan pada 75 MPa, suhu acuan pada $70^0$c, suhu lebur pada $190^0$c dan masa padatan pada 1.5s.

# TABLE OF CONTENTS


| | Page |
|---|---|
| **SUPERVISOR'S DECLARATION** | ii |
| **STUDENT'S DECLARATION** | iii |
| **DEDICATION** | iv |
| **ACKNOWLEDGEMENTS** | v |
| **ABSTRACT** | vi |
| **ABSTRAK** | vii |
| **TABLE OF CONTENTS** | viii |
| **LIST OF TABLES** | xi |
| **LIST OF FIGURES** | xi |
| **LIST OF ABBREVIATIONS** | xiii |

**CHAPTER 1 INTRODUCTION**

| | | |
|---|---|---|
| 1.1 | Project Background | 1 |
| 1.3 | Problem Statement | 2 |
| 1.3 | Project Objectives | 2 |
| 1.4 | Project Scopes | 2 |

**CHAPTER 2 LITERATURE REVIEW**

| | | |
|---|---|---|
| 2.1 | Introduction to Injection Molding | 3 |
| | 2.1.1 History of Injection Molding | 4 |
| 2.2 | Materials | 5 |
| | 2.2.1 Acrylonitrile Butadiene Styrene (ABS) | 5 |
| | 2.2.2 Polyethylene (PE) | 6 |
| 2.3 | Process cycle | 8 |
| | 2.3.1 Clamping | 7 |
| | 2.3.2 Injection | 7 |
| | 2.3.3 Cooling | 8 |

| | | |
|---|---|---|
| | 2.3.4 Ejection | 8 |
| 2.4 | Injection Molding Parameters | 9 |
| 2.5 | Molding Defects | 10 |
| | 2.5.1 Sink marks defect | 11 |
| | 2.5.1.1 Causes of Sink Marks Defect | 13 |
| | 2.5.2 Weld Lines | 15 |
| 2.6 | Computer-Aided Design Model | 17 |
| 2.7 | Moldflow Simulation Software | 18 |
| 2.8 | Design of Experiment (DOE) | 19 |
| 2.9 | Taguchi Method | 20 |
| 2.10 | Orthogonal Arrays | 20 |
| 2.11 | Signal to Noise Ratio (S/N) | 21 |
| 2.12 | Analysis of Variance (ANOVA) | 22 |

**CHAPTER 3 METHODLOGY**

| | | |
|---|---|---|
| 3.1 | Introduction | 23 |
| 3.2 | Selection of Parameters and Orthogonal Arrays (OAs) | 25 |
| 3.3 | CAD Data Development | 27 |
| 3.4 | Moldflow Analysis using MPI 5.0 | 28 |
| 3.5 | Analyze Data | 29 |

**CHAPTER 4 RESULTS AND DISCUSSION**

| | | |
|---|---|---|
| 4.1 | Introduction | 30 |
| 4.2 | Moldflow Analysis Result for ABS and Polyethylene (PE) | 31 |
| 4.3 | Signal To Noise Ratio (S/N) | 33 |
| 4.4 | S/N Response Diagram for ABS and Polyethylene (PE) | 35 |
| 4.5 | Analysis of Variance (ANOVA) | 37 |
| 4.6 | Verification Test | 39 |

**CHAPTER 5 CONCLUSIONS AND RECOMMENDATIONS**

5.1 Conclusions 40
5.2 Recommendations 41

**REFERENCES** 42
**APPENDICES** 43
A1 Gantt Chart FYP 1 44
A2 Gantt Chart FYP 2 45
B1 Moldflow Analysis Results for ABS Material 46
B2 Moldflow Analysis Results for PE Material 55

## LIST OF TABLES


| Table No. | Title | Page |
|---|---|---|
| 2.1 | Material Properties for ABS and PE | 7 |
| 2.2 | Common Molding Defects | 10 |
| 2.3 | Causes of Sink Marks | 14 |
| 2.4 | Causes of Weld Lines | 16 |
| 3.1 | The parameters for 3 levels of selected factors | 25 |
| 3.2 | Experimental plan using $L_{18}$ Orthogonal Array | 26 |
| 4.1 | The Sink Index and S/N ratio for ABS | 33 |
| 4.2 | The Sink Index and S/N ratio for Polyethylene | 34 |
| 4.3 | Analysis of Variance for ABS | 38 |
| 4.4 | Analysis of Variance for PE | 39 |


## LIST OF FIGURES


| Figure No. | Title | Page |
|---|---|---|
| 2.1 | Injection Molding Machine | 4 |
| 2.2 | Sink Marks Defect | 11 |
| 2.3 | Variation in Wall Thickness causes Sink Marks. | 12 |
| 2.4 | Sink Marks can be eliminated by creating a design, rib, serrations | 14 |
| 2.5 | Formation of Weld Lines | 15 |
| 2.6 | CAD Model of a Card Holder | 17 |

| | | |
|---|---|---|
| 2.7 | Moldflow Simulation | 18 |
| 3.1 | Project Methodology Flow | 24 |
| 3.2 | (a) Matric Card Holder 3D view, (b) Front View (c) Side View | 27 |
| 3.3 | Moldflow Analysis using MPI 5.0 | 28 |
| 4.1 | Moldflow Analysis for ABS (test no. 1) | 31 |
| 4.2 | Moldflow Analysis for Polyethylene (test no. 1) | 32 |
| 4.3 | S/N response diagram for ABS | 35 |
| 4.4 | S/N response diagram for Polyethylene (PE) | 36 |

## LIST OF ABBREVIATIONS

| | |
|---|---|
| CAD | Computer Aided Design |
| DOE | Design of experiment |
| DOF | Degree-of-freedom |
| MPa | Mega Pascal |
| ABS | Acrylonitrile Butadiene Styrene |
| PE | Polyethylene |
| OAs | Orthogonal arrays |
| MPI | Moldflow Plastics Insight |
| ANOVA | Analysis of variance |

# CHAPTER 1

# INTRODUCTION

## 1.1 PROJECT BACKGROUND

The injection molding process was first designed in the 1930s and was originally based on metal die casting designs (Douglas, 1996). This process is most typically used for thermoplastic materials which may be successively melted, reshaped and cooled.

Sink marks are depressions on the surface of injection molded plastic parts caused during the plastic cooling process. Thicker sections of plastic will cool at a slower rate than others, and will yield a higher percentage of shrink in that local area. The extra shrinkage in that local area is what causes the depressions. After the material on the outside has cooled and solidified the core material start to cool. Its shrinkage pulls the surface of the main wall inward, causing a sink mark (Michael et. al, 1997).

The purpose of this project is to investigate and analyze sink mark defect on injection molding process and reduce the defect using optimum parameters. The Taguchi Method based on orthogonal arrays used in this study to determine and analyze the optimal injection molding parameters. The parameters was investigated are packing pressure, packing time, melt temperature and mold temperature. ABS and PE are material was tested in this study.

## 1.2 PROBLEM STATEMENT

A sink mark is a local surface depression that typically occurs in moldings with thicker sections, or at locations above ribs, bosses, and internal fillets. Sink marks and voids are caused by localized shrinkage of the material at the thick sections without sufficient compensation when the part is cooling. However, other researchers used different parameters settings and different raw materials and it cause different results compare to each other. In order to understand the sink mark defect on certain parameter settings and certain raw materials used, this project need to be conducted by using optimum parameters, it can be reduce the sink mark defect.

## 1.3 PROJECT OBJECTIVES

The main objectives for this project are:

(i) To investigate sink mark defect on Injection Molding Process.

(ii) To minimize sink mark defect on Injection Molding.

(iii) To determine optimum parameters to reduce sink mark defect.

## 1.4 PROJECT SCOPES

The scopes for this project are focusing on simulation of sink mark defect on different raw materials. The existing part such as matric card holder was used. The selection of orthogonal arrays (OAs) depends on the level and parameter involved thus the 3 levels and 4 parameters were chosen. In this project the parameters settings was tasted are packing pressure, mold temperature, melt temperature and packing time. Materials that were used are Acrylonitrile Butadiene Styrene (ABS) and polyethylene (PE). The Solidworks 2006 and Moldflow Plastics Insight 5.0 software are used to design and show the simulation of the molten plastics injected into cavity.

# CHAPTER 2

# LITERATURE REVIEW

## 2.1 INTRODUCTION TO INJECTION MOLDING

Injection molding is a manufacturing technique for making parts by injected molten plastic at high pressure into a mold, to produce the product's shape (Douglas, 1996). In manufacturing field, injection molding is widely used to produce a variety of parts, from the smallest component to biggest component.

Injection molding is the most common method of production, with some commonly made items including bottle caps and outdoor furniture. Injection molding can also be used to manufacture parts from aluminum or brass. The melting points of these metals are much higher than those of plastics, this makes for substantially shorter mold lifetimes despite the use of specialized steels.

Injection molding machines consist of a material hopper, an injection ram or screw-type plunger, and a heating unit (Douglas, 1996). They are also known as presses, they hold the molds in which the components are shaped. Presses are rated by tonnage, which expresses the amount of clamping force that the machine can exert. This force keeps the mold closed during the injection process. Tonnage can vary from less than 5 tons to 6000 tons, with the higher figures used in comparatively few manufacturing operations (Tim et. al., 2007). Figure 2.1 shows the injection Molding Machine.

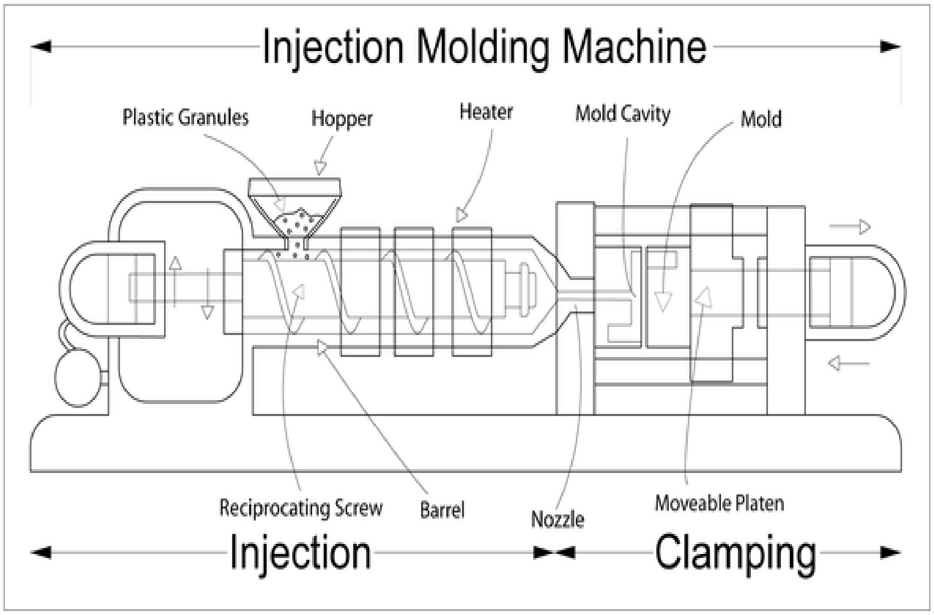


**Figure 2.1** Injection Molding Machine

### 2.1.1 History of Injection Molding

In 1868 John Wesley Hyatt developed a plastic material he named Celluloid which had been invented in 1851 by Alexander Parks. Hyatt improved it so that it could be processed into finished form. In 1872 John, with his brother Isaiah, patented the first injection molding machine (Patent, 1872). This machine was relatively simple compared to the existing machines today. It basically worked like a large hypodermic needle injecting plastic through a heated cylinder into a mold. The industry progressed slowly over the years producing products such as collar stays, buttons, and hair combs until it exploded in the 1940s because World War 2 created a huge demand for inexpensive, mass-produced products. In 1946 James Hendry built the first screw injection machine. This machine allowed material to be mixed before injection, which meant colored plastic or recycled plastic could be added to the virgin material and mixed thoroughly before being injected. Today screw injection machines account for 95% of all injection machines. The industry has evolved over the years from producing combs and buttons to producing a vast array of products for many industries including automotive, medical, aerospace, consumer, toys, plumbing, packaging, and construction.

## 2.2 MATERIALS

### 2.2.1 Acrylonitrile Butadiene Styrene (ABS)

Acrylonitrile Butadiene Styrene (ABS) is the polymerization of Acrylonitrile, Butadiene, and Styrene monomers. Chemically, this thermoplastic family of plastics is called "terpolymers", in that they involve the combination of three different monomers to form a single material that draws from the properties of all three.

ABS possesses outstanding impact strength and high mechanical strength, which makes it so suitable for tough consumer products. Additionally, ABS has good dimensional stability and electrical insulating properties. Dynalab Corp's plastic fabrication shop fabricates thousands of catalog and custom ABS products (Tony Whelan, 1994).

Since plastic injection molding of ABS readily shows evidence of sink marks, maintaining uniform wall thickness throughout the part is essential in producing an aesthetically pleasing molding. Ribs and gussets should be used to core out thick sections. The rib thickness should not exceed 50% of the intersecting wall thickness. Bosses incorporated in the design can also result in sink marks if not properly designed. Sharp corners result in stress concentrations and should be avoided (Tony Whelan, 1994).

Reprocessing waste material from sprues and runners is a common practice in plastic industry. This reprocessing involves changes of properties of the material, which have to be evaluated. Consecutive injection process has been used to simulate the reprocessing in a laboratory environment.

Thus, all processing history data are available and consequences of degradation are quantified easily. In the present study, the effects of reprocessing ABS polymer have been studied. The objectives of this investigation are to determine and quantify the effects of the thermal and shear rate history on the viscosity at high shear rates.

### 2.2.2 Polyethylene (PE)

Polyethylene (IUPAC name polyethene) is a thermoplastic commodity heavily usedin consumer products. Over 60 million tons of the materials are produced worldwide every year. Polyethylene is a polymer consisting of long chains of the monomer ethylene (IUPAC name ethene) (Tony Whelan,1994).

The recommended scientific name 'polyethene' is systematically derived from thescientific name of the monomer. In certain circumstances it is useful to use a structure–based nomenclature. In such cases IUPAC recommends poly (methylene) (Tony Whelan, 1994).

The difference is due to the 'opening up' of the monomer's double bond upon polymerisation. In the polymer industry the name is sometimes shortened to PE, in a manner similar to that by which other polymers like polypropylene and polystyrene are shortened to PP and PS, respectively.

In the United Kingdom the polymer is commonly called polythene, although this is not recognized scientifically. Polyethylene is created through polymerization of ethene. It can be produced through radical polymerization, anionic addition polymerization, ion coordination polymerization or cationic addition polymerization.

This is because ethene does not have any substituent groups that influence the stability of the propagation head of the polymer. Each of these methods results in a different type of polyethylene.

Polyethylene is classified into several different categories based mostly on its density and branching. The mechanical properties of PE depend significantly on variables such as the extent and type of branching, the crystal structure, and the molecular weight (Tony Whelan, 1994). Table 2.1 shows the material properties for ABS and PE.

**Table 2.1:** Material Properties for ABS and PE

| Material properties | ABS | PE |
|---|---|---|
| Material structure | Amorphous melt | Crystalline |
| Temperature ($^oC$) | 200-240 | 190-250 |
| Mold Temperature ($^oC$) | 40-80 | 30-70 |
| Max shear stress (MPa) | 0.3 | 0.26 |
| Max Shear Rate (1/s) | 50000 | 24000 |
| Melt density ($g/cm^3$) | 0.94752 | 0.73817 |
| Solid density ($g/cm^3$) | 1.0432 | 0.95163 |
| Elastic Modulus (MPa) | 2240 | 690 |

## 2.3 PROCESS CYCLE

The process cycle for injection molding is very short, typically between 2 seconds to 2 minutes and consists of the following four stages:

### 2.3.1 Clamping

Prior to the injection of the material into the mold, the two halves of the mold will first be securely closed by the clamping unit. Each half of the mold is attached to the injection molding machine and one half is allowed to slide. The hydraulically powered clamping unit pushes the mold halves together and exerts sufficient force to keep the mould securely closed while the material is injected.

The time required to close and clamp the mould is dependent upon the machine - larger machines (those with greater clamping forces) will require more time. This time can be estimated from the dry cycle time of the machine·

### 2.3.2 Injection

The raw plastic material, in the form of pellets, is fed into the injection molding machine, and advanced towards the mold by the injection unit. During this process, the material is melted by heat and pressure. The molten plastic is then injected into the mold very quickly and the buildup of pressure packs and holds the material.

The amount of material that is injected is referred to as the shot. The injection time is difficult to calculate accurately due to the complex and changing flow of the molten plastic into the mould. However, the injection time can be estimated by the shot volume, injection pressure, and injection power.

### 2.3.3 Cooling

The molten plastic that is inside the mold begins to cool as soon as it makes contact with the interior mold surfaces. As the plastic cools, it will solidify into the shape of the desired part. However, during cooling some shrinkage of the part may occur.

The packing of material in the injection stage allows additional material to flow into the mold and reduce the amount of visible shrinkage. The mold cannot be opened until the required cooling time has elapsed. The cooling time can be estimated from several thermodynamic properties of the plastic and the maximum wall thickness of the part.

### 2.3.4 Ejection

After sufficient time has passed, the cooled part may be ejected from the mold by the ejection system, which is attached to the rear half of the mold. When the mould is opened, a mechanism is used to push the part out of the mold. Force must be applied to eject the part because during cooling the part shrinks and adheres to the mold.

In order to facilitate the ejection of the part, a mold release agent will be sprayed onto the surfaces of the mold cavity prior to injection of the material. The time that is required to open the mold and eject the part can be estimated from the dry cycle time of the machine and should include time for the part to fall free of the mold. Once the part is ejected, the mould will be clamped shut for the next shot to be injected.

## 2.4 INJECTION MOLDING PARAMETERS

There are a few parameters that involved in injection molding process. Below here explain the details of each parameter:

**a)** **Melt Temperature** is the temperature of the cylinder of the machine which determines the temperature of the material that will be injected into the mold.

**b)** **Mold temperature** is the temperature of the steel mold.

**c)** **Packing pressure** is used for packing out a part and is often related to the fill pressure. Packing pressures are commonly between 20% and 100% of the fill pressure, and can be higher and lower. An important aspect of the packing pressure is that it cannot be high enough to exceed the clamp limit of the machine.

**d)** **Packing time** should be long enough so the gate has a chance to freeze off. This time can be estimated from the time to freeze plot from a filling analysis, however, it will generally be low due to shear heat during packing.

**e)** **Injection speed** is the speed of advance of the screw which is driven by a motor coupled with it.

**f)** **Cooling time** can be defined as the time needed for the circulated water around the mold to cool and solidify the plastic part.

**g)** **Holding pressure** is the pressure used for regulating and closing the mold.

**h)** **Injection pressure** is the pressure that is applied to the ram during the injection phase, causing the material to flow, and can be measured approximately by a transducer located in the nozzle. There is a direct relationship between the injection pressure and the hydraulic line pressure called the machine intensification ratio.

**i)** **Injection time** is the time it takes to fill the cavity

## 2.5 MOLDING DEFECTS

Injection molding is a complex technology with possible production problems. They can either be caused by defects in the molds or more often by part processing (molding). Table 2.2 shows the common molding defects.

**Table 2.2:** Common Molding Defects

| Molding Defects | Alternative name | Descriptions | Causes |
|---|---|---|---|
| Sink marks | | Localized depression (In thicker zones) | Holding time/pressure too low, cooling time too short, with sprueless hot runners this can also be caused by the gate temperature being set too high |
| Voids | | Empty space within part (Air pocket) | Lack of holding pressure (holding pressure is used to pack out the part during the holding time). Also mold may be out of registration (when the two halves don't center properly and part walls are not the same thickness). |
| Weld line | Knit line / Meld line | Discolored line where two flow fronts meet | Mold/material temperatures set too low (the material is cold when they meet, so they don't bond) |
| Warping | Twisting | Distorted part | Cooling is too short, material is too hot, lack of cooling around the tool, incorrectwater temperatures (the parts bow inwards towards the hot side of the tool) |

### 2.5.1 Sink Marks Defect

Sink marks or shrink marks are hollows or indentations that occur on the outer surfaces of molded components. Whether or not sink marks are treated as a problem depends on the required quality of appearance. For example, this would not be acceptable for external molding components which must be highly attractive in nature. Sink mark behavior depends on the volumetric shrinkage of the plastic such as the isothermal PVT characteristic and the chronological history of all locations within the injection molding process are important. In specific terms, this phenomenon occurs during the transition from the molten condition upon injection to the solid condition upon dwelling and cooling. (Brydson, J, 1999). Figure 2.2 shows Sink Mark Defect

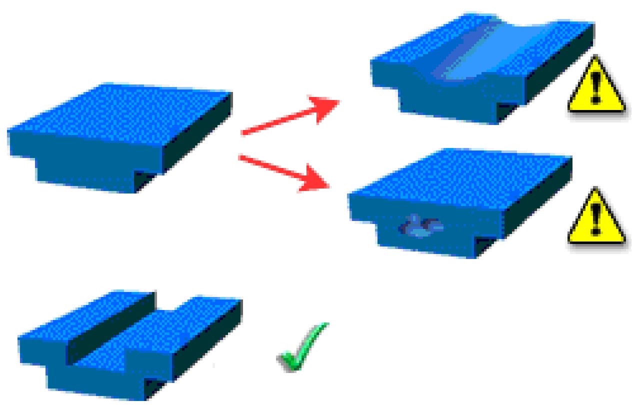

**Figure 2.2**: Sink Marks Defect

Molten plastic that has been injected into the die begins to cool and solidify from the die surface. As plastics continues to cool and harden from the outside (i.e., during well and cooling), certain injection settings such as the dwell pressure and time make it possible to compensate for changes in the volume of plastics (i.e., volume shrinkage) resulting from the PVT characteristics. In these cases, the plastic at the surface of the die can be drawn towards the inside of the molding when volumetric shrinkage occurs in the molten plastic still present at the interior, and this results in the cosmetic defect referred to as sink marks