Clemson University
TigerPrints

All Theses Theses

7-2008

Reduction of sink marks in wire insert molded parts

Gauri Phadke
Clemson University, gphadke@clemson.edu

Follow this and additional works at: https://tigerprints.clemson.edu/all_theses

Part of the Engineering Mechanics Commons

Recommended Citation
Phadke, Gauri, "Reduction of sink marks in wire insert molded parts" (2008). *All Theses*. 427.
https://tigerprints.clemson.edu/all_theses/427

This Thesis is brought to you for free and open access by the Theses at TigerPrints. It has been accepted for inclusion in All Theses by an authorized administrator of TigerPrints. For more information, please contact kokeefe@clemson.edu.

REDUCTION OF SINK MARKS IN WIRE INSERT MOLDED PARTS

A Thesis
Presented to
the Graduate School of
Clemson University

In Partial Fulfillment
of the Requirements for the Degree
Master of Science
Mechanical Engineering

---

by
Gauri Sudhir Phadke
August 2008

Accepted by:
Dr David Angstadt, Committee Chair
Dr Laine Mears
Dr Huang Yong

ABSTRACT

Integrating the huge wiring harness network with the plastic trim components found in the car is an attractive way to reduce the vehicle weight. When any foreign object is inserted in an injection molded part, the change in polymer cross section leads to sink marks which are aesthetic defects and are not acceptable for plastic trim components.

In this thesis, a method to minimize or eliminate sink marks for wire insert molded component using injection molding process is presented. L9 Taguchi DOE experimental framework has been employed to study the effect of process parameters, part rib geometry, and the presence of wire itself on the sink mark formation. Sink depth is defined as the excess deflection in the surface profile which can be felt to hand. A descriptive simulation study is presented where process parameters namely, the mold temperature, the melt temperature, and the pack time are varied on different sets of rib geometry to observe the depth of a sink mark. The simulation results indicate that high mold temperature effectively minimizes the sink depth for all the rib geometries whereas the influence of melt temperature and pack time depends on the particular rib geometry. The results also testify that a proper combination of rib geometry and process parameters eliminates the sink mark.

ACKNOWLEDGMENTS

I would like to thank my advisor, Dr. David C. Angstadt for his guidance, faith and support throughout the course of this project. Dr. Angstadt's constant feedback and high expectations have driven me to keep making progress and complete this work. I extend my sincere thanks to Dr. Mica Grujicic for allowing me access to Moldflow.

A special thanks to my fellow graduate students Peiman Mosaddegh and Celina Renner for their selfless help during the course of this work. I would also like to thank my friends Nitendra Nath, Gayatri Keskar ,Sonia Ramnani, Shyam Panyam, Judhajit Roy and Ajit Kanda for their constant encouragement and help. Finally, I would like to thank my family and friends for all the love and care without which this work would be incomplete.

TABLE OF CONTENTS

Page

TITLE PAGE i

ABSTRACT ii

ACKNOWLEDGMENTS iii

LIST OF TABLES vi

LIST OF FIGURES vii

CHAPTER

I. INTRODUCTION 1

Motivation 1
Injection molding process 2
Injection molding cycle 3
Insert molding process 5

II. INJECTION MOLDING 7

Polymer material 7
Injection molding machine 10
Defects in injection molding 11
Shrinkage and warpage 13
Sink marks 14
Injection molding parameters 15

III. COMPUTER AIDED ENGINEERING 18

Computer aided engineering 18
Finite element analysis meshes 21
Process definition for analysis 24

IV. SIMULATION METHODOLOGY 25

Outline 25
Material selection 27
Geometry selection 29
Design of Experiment 32
Process definition for simulation 35
Sink depth metrics 38

V. RESULTS AND DISCUSSION 41

Analysis of Taguchi DOE 42
Effect of insert molded wire on sink mark depth 44
Effect of parameters on sink mark depth 47
Effect of geometry on sink mark depth 50

VI. CONCLUSION 53

REFERENCES 55

LIST OF TABLES

Table Page

1 Orthogonal array and experimental lay out for L9 model 35

2 Sample calculation table for sink depth for geometry A 44

3 Sink mark depths for effect of insert molding a wire 44

4 Sink mark depths for parameters for geometries A, B, C, D 47

5 Sink mark depths for all geometries 50

LIST OF FIGURES

Figure		Page
1	Injection molding cycle	4
2	Insert molded electro-mechanical connector	5
3a	Ethylene monomer	8
3b	Polyethylene chain a polymer with n ethylene monomers	8
4	Sink mark formation	15
5a	Midplane Mesh	22
5b	Fusion Mesh	22
5c	3D Mesh	22
6	VW Beetle interior parts, Central console cockpit on the Alfa 166olding cycle in Bayblend®	28
7	Configuration A: Rectangular rib; wire being centered in the plane of the flat base plate	31
8	Configuration B: Rectangular rib; wire completely within the rib	31
9	Configuration C: Semicircular rib; wire being in the centre plane of the flat base plate	32
10	Configuration D: Semicircular rib; wire is positioned such that it is enclosed in the rib section	32
11	Configuration E: Semicircular solid rib	32
12	Mesh for Geometry A	37
13	Deflection plot for Geometry C trial 2	39
14	Deflection in Z direction path plot	39

15 Minitab plot for best line fit 41

16 Effect of insert molded wire on sink mark 45

17 Freeze time profile for Geometry E and C 46

18 Effect of parameters on sink marks for all geometries 49

19 Comparison of sink mark depth results for all geometries 51

CHAPTER ONE

INTRODUCTION

Automobile manufacturers are increasingly looking to plastics for light weight solutions. A recent study shows that plastics account for up to 10% of the total weight of a car. Plastics are present throughout the body of the vehicle right from interiors to bumpers and doors. Along with the increase in plastic, there has been tremendous increase in use of electrical equipment in cars. This leads to the need for an extensive network of wires. These wires form the wiring harness of a car. Wiring harness weighs approximately 70 kg and is almost 2 km long in a typical automobile today [1]. Hence, electric wiring harness along with plastics runs from one end of the car to other.

It is interesting to note that the wire sizes selected in the wiring harness of a car are to maintain sturdiness of the wiring instead of the wire's current carrying capacity. This results in the selection of thicker wire than what is needed for a particular application. Considering the presence of both plastic and wiring harness throughout the car body integrating them would be beneficial. If the wire is embedded by some method into the plastic component itself, the gauge selection criterion can be exploited as the sturdiness needed would be lent by the plastic component. This would result in the use of a smaller gauge wire which in turn would translate into weight savings. Further reduction in the weight of the wiring harness is possible by elimination of the use of protective tapes, ducts and clamping devices. Also, if the connectors are incorporated into the plastic components, the electrical connections would be built while the part is assembled; thus, saving time and devising a fool proof method to avoid wrong connections. One of

the easiest and most inexpensive methods for mass production of such components is injection molding with wire as the insert. Although, there are many advantages of using this process, the main factor preventing the use of this method of manufacturing is the accompanying surface defect, typically sink mark.

The area of application of such a method would be in fascia of automobiles. One area of application could be the dash board. A dash board has a variety of wires running along it for applications such as the entertainment system, the lights, air conditioning units, etc. For instance, the car radio requires about 1.5 A to work. A 20 AWG gauge wire with a wire diameter of 0.8 mm can carry a current of 1.5 A for power transmission [2] and thus, meets the requirement of a car radio. Using an insert molded wire in the console to supply power to the radio would mean that the surface of the console would have to be defect free. Thus for the purpose of this study the wire dimension is selected to be 0.8mm.

The goal of this research is to eliminate or minimize sink marks on the plastic component with insert molded 0.8mm diameter wire.

Injection Molding Process

Injection molding is the most common manufacturing technique used for cyclic mass production of plastic components. Injection molding, as the name suggests, consists of injecting molten polymer into a mold under pressure to get a finished part. It is a high speed, low running cost process requiring very little or in most cases no finishing process

with high tolerance control and repeatability. Both thermoplastics and thermosets can be injection molded to create complex parts.

Injection molding cycle

The injection molding cycle begins with the closing of the two mold halves to create the mold cavity. This is followed by the filling phase, packing phase and the cooling phase [3].

Filling Phase: The barrel moves forward when the cycle begins. The start of the molding cycle is triggered by either manual control or by closing of the mold for automatic and semi-automatic molding cycle. The heating coils melt the polymer material in the screw. The screw moves forward and creates pressure and forces the molten plastic through the nozzle and into the mold cavity via the gate and runners. The phase continues until the mold cavity is completely filled. Figure 1 shows the various stages in injection molding cycle.

Packing Phase: At the end of the filling phase when the mold cavity is almost filled completely the packing cycle begins. During this phase, the pressure inside the cavity is maintained until the gate freezes off. By maintaining pressure inside the mold, a small amount of molten polymer flows into the cavity to compensate for the volumetric shrinkage occurring inside due to cooling.

Cooling Phase: At the end of the packing phase the material is allowed to cool further in the mold itself until it is rigid enough to be removed. The cooling cycle is the longest of all three phases followed by the packing time. During this phase, heat is removed from the mold by the cooler system until the specified cooling time is completed.

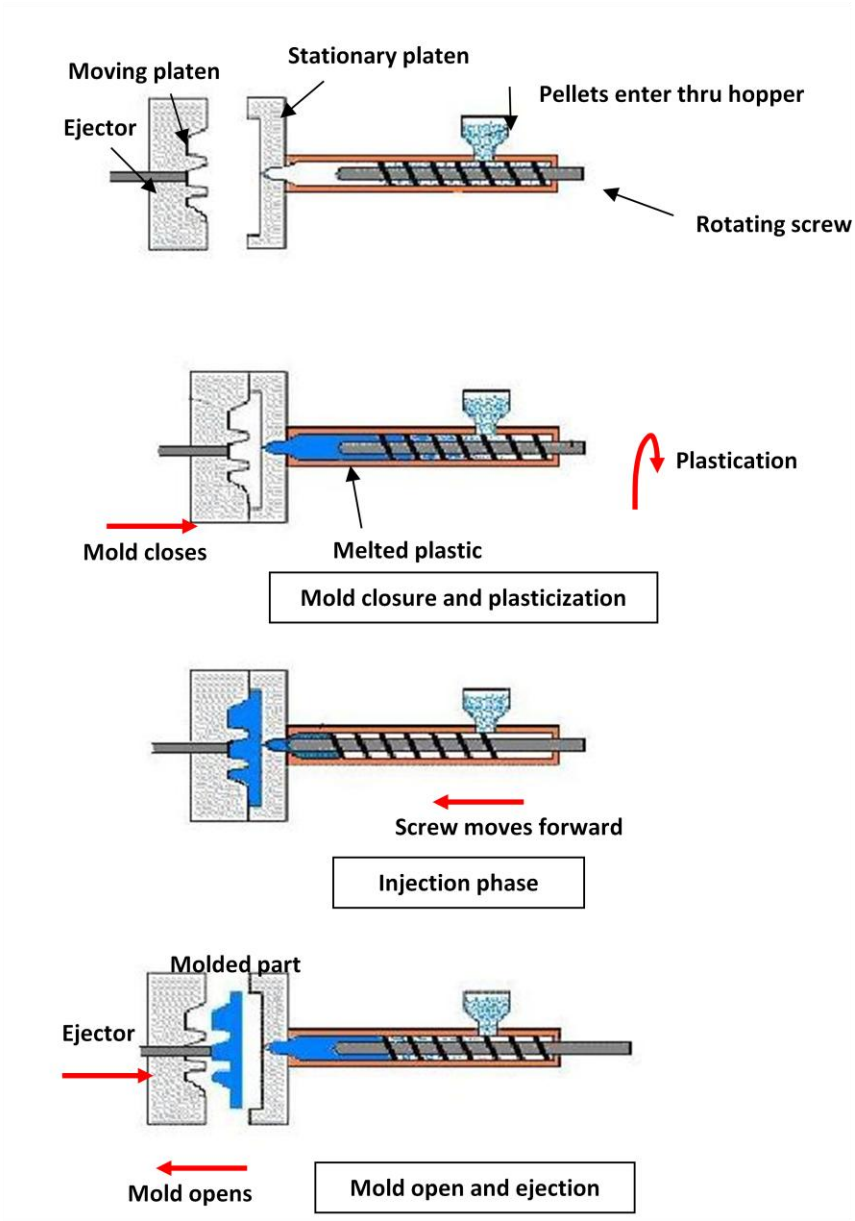

Figure 1 : Injection molding cycle

Insert Molding Process

Insert molding consists of placing an insert in the mold cavity and injecting polymer around it. This process forms a single component consisting of the insert embedded in the plastic. The insert itself could be a metallic object as in this case a wire or another polymer. Some of the common applications of insert molding are electrical plugs, threaded fasteners, tail lamp components in cars and bolts in bumpers among others. The main advantage of insert molding a wire into the plastic component is that it leads to decreased assembly time and labor costs since the previously independent wiring harness and trim are now one single component. Having one part instead of two also makes the structure more sturdy and reliable. This combination can allow for the use of thinner wires leading to reduction in weight.

Figure 2 Insert molded electro-mechanical connector [4]

Maintainability and reparability are to be considered if such a method is to be used to integrate electric wiring harness and plastic components. If, at any time, the

wiring inside the component undergoes even a small failure, the whole part would have to be replaced. This is not an economic option from the customer's point of view.

CHAPTER TWO

INJECTION MOLDING

One of the main advantages of injection molding is the ability of the process to make repeatable identical parts on a mass scale. This requires making defect free parts consistently. Important aspects of injection molding that go into making a good injection molded part are:

1. Part design
2. Mold design
3. Polymer material
4. Processing parameters
5. Molding machine

A proper combination of the aforementioned aspects makes a good defect free plastic part. The next sections will discussion injection molding briefly with respect to these aspects.

Polymer Material

Polymer as the name suggests is made of chains of monomers held together by covalent bonds. Monomers are the periodically repeated structural units in the polymer chain. Figure 3 shows the ethylene monomer, n such monomers together make the Polyethylene polymer. Polymers are broadly classified as thermoset, thermoplastic, and elastomers.

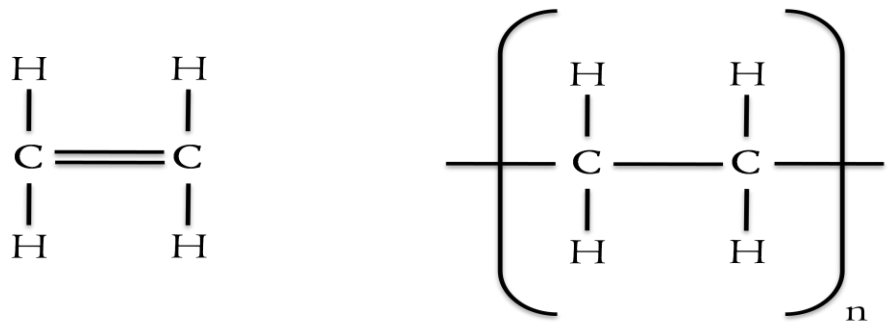

Figure 3 a) Ethylene monomer b) Polyethylene chain a polymer with n ethylene monomers

Thermoset

Thermoset polymers are made of covalent bonds [5]. The themoset plastic when heated, solidifies by chemical curing. This curing forms a network of crosslinked covalent bonds. These bonds are present within the polymer chain and between multiple polymer chains. The covalent bond between the polymer chains prevents thermoset from melting again once they are fomed. Thermoset once formed cannot be melted again to give a new shape which means they cannot be recycled. These polymers are much stiffer and stronger compared to thermoplastics. Some of the commonly found thermosets are, vulcanized rubber, Bakelite, epoxy resins melamine resins, etc.

Thermoplastic

Thermoplastic polymers have covalent bonds within a chain but the chains themselves are held together by van der Waal's forces which are weaker than covalent bonds. Due to the presence of these weaker bonds connecting the chains, these polymers can be remelted and reformed unlike the Thermoset polymers. They are further divided into two categories depending on their polymer structures as amorphous and semi-crystalline polymers.

Amorphous polymers are those which have a random structure of polymer chains. They do not have a distinct melting point at which they go from being solid to liquid instead they have a glass transition temperature (Tg)at which the polymer bonds weaken and the polymer 'flows'. Amorphous polymers shrink uniformly in both flow direction and transverse direction [3]. Some common amorphous polymers are Polycarbonate (PC), Polystyrene (PS), Acrylonitrile butadiene styrene (ABS).

Semi-crystalline polymers are those which have a fairly ordered molecular structure forming crystals with amorphous regions surrounding these patches of crystal structures. These polymers have a very well defined melting temperature (Tm) at which their semi-crystalline structure disappears. They usually show less shrinkage in the direction of flow as compared to transverse flow. Some common semi-crystalline polymers are High density polyethylene (HDPE), Polypropylene (PP).

Elastomers

Thermoplastics above their glass transition temperature are highly elastic in nature. They have a mixture of van der Waal forces and covalent bonds which give the elastomers their high elasticity. Due to their structure, they can be straightened out and show large elastic deformations. Some of the common elastomers are Natural rubber, Butyl rubber, Poly butadiene.

Injection molding machine

A typical injection molding machine consists of the following components:

- Injection system: It consists of hopper, barrel with a reciprocating screw and a nozzle. The function of this unit is to take the polymer material, melt it, create pressure and inject it into the mold. The hopper holds the material in the form of pellets or granules and provides it to the barrel. The barrel houses a reciprocating screw and heating bands which plasticizes the polymer material. The screw compresses the material and conveys it to nozzle which is the link between the barrel and the mold.
- Mold system: The mold is the shaping unit of the process. It has the inverse geometry of the desired part needed. It consists of stationary and moving platens, base plates which when joined form the mold cavity, the sprue and runner system which route the molten polymer to the mold cavity, and cooling channels for cooling the mold to keep the mold surface temperature at a desired value. The mold basically acts as the heat removal system as the melt freezes to form the molded part.
- Hydraulic system: It consists of pumps, valves, hydraulic motors and the related tubing and reservoir. It provides the power to operate mold closing and opening, clamping force to keep the mold closed during the cycle, move the ejector pins and rotate and translate the screw. The clamping unit holds the two mold halves together to create a mold cavity. If the clamping pressure is too low, it results in the mold not closing properly leading to polymer leakage through the parting line

and creating flash whereas too high a clamping pressure results in to mold wear and damage.

- Control system: All the electronics used to control the processing parameters like temperatures, pressures, screw speed and position, mold closing opening, times collectively form the control system of the machine. It is essential for the running the automatic operations for the machine.

Defects in injection molding

In situations where the aforementioned parameters are not controlled properly, a variety of commonly observed defects are caused. Some of them are mentioned below [12]:

1. Dimensional variation is the variation in the part dimensions from batch to batch or shot to shot production while the machine settings remain the same. Unreliable machine control, small molding window or unstable material usually lead to dimensional variations.
2. Sink marks are local depressions in a molded part surface. Voids are vacuum bubbles in the core. Combination of processing parameters leads to sink marks and voids.
3. Air traps are air trapped inside a molded part. Improper venting in the mold leads to these.
4. Weld and Meld lines are lines seen on the surface of a molded part. Weld lines are formed when separate melt flow fronts flowing in opposite direction meet. Meld

lines are formed when flow fronts parallel to each other meet. Holes, inserts multiple gates, wall thickness variation lead to multiple flow fronts leading to weld and meld lines.

5. Flash is the excessive material leaking out from locations where the mold separates such as the parting line. Low clamp pressure, gaps in mold closing, high melt temperatures and improper venting leads to excessive flashing.
6. Jetting is when polymer is injected into the mold at high velocity which leads to part being filled from extreme end of cavity instead of progressively starting at the gate. Jetting leads to weak parts with surface and internal defects. Improper mold design and injection profile along with melt temperature selection may lead to jetting.
7. Black specks or streaks, brittleness, burn marks and discoloration are dark spots that are formed due to material degradation or entrapped air.
8. Delamination is the peeling off of surface layer of a molded part. Incompatible blend of polymers, low melt temperatures, excessive moisture or mold release agents may lead to delamination.
9. Fish eyes are unmolten materials getting injected into mold cavity and appearing on the surface of the molded part. Low barrel temperatures, material contamination or low screw rotation speed and back pressure lead to fish eyes.

Shrinkage and Warpage

All polymers undergo large volumetric changes as they go from their molten stage to a rigid solid state. In absence of any external pressure, the solid state volume is usually about 75 % of the liquid volume. In case of injection molding since the filling is done under pressure, the volumetric change is about 15%. This drastic change in the polymer volume results in changed dimensions as a part freezes leading to shrinkage and warpage.

Shrinkage and warpage are the two most commonly seen defects which are almost unavoidable but with proper care can be controlled within acceptable limits. Part shrinkage is the reduction in the dimension of the part. Shrinkage occurs due to contraction of the molten polymer as it undergoes cooling and thermal energy is lost. As the thermal energy is given out from the polymer mass, the polymer chains begin to relax from their stretched states during the molten phase which leads to shrinking of the part.

If the shrinkage is uniform, the part does not deform, just the end dimensions of the part are smaller than the mold dimensions. In such cases, where the percentage change of polymer is know, the problem of changed dimension can easily be resolved by creating a larger mold than the desired part dimension. Uniform shrinkage is acceptable can be acceptable and can be controlled by cooling rate and packing.

When the shrinkage is not uniform, the part tends to deform and the result is warpage of the part. As the part shrinks unequally as it cools, stresses develop within a part and depending on the stiffness of the part, it undergoes deformation. Some causes of warpage are dissimilar wall sections, gating in thin sections, cores that cause weld lines,

non uniform characteristics of material like anisotropic shrinkage of semi crystalline thermoplastics as mentioned earlier, etc.

Shrinkage occurs in 3 stages [6]:

1. In-Mold shrinkage which takes place during the filling packing and cooling phase.
2. As – Mold shrinkage which happens just after mold opening.
3. Post shrinkage is the shrinkage occurring at the end of 48 hours after the mold has been ejected. The change in dimension is mainly due to aging, creeping, crystallization, etc.

Sink Marks

Sink marks are local depressions formed in an injection molded part due to differential cooling. Sink marks are aesthetic defects alone and do not affect the strength of the part. For components forming a fascia of a product, sink marks lead to rejections and are not acceptable.

As the melt is injected into a mold, the molten polymer comes in contact with the cooler mold. The polymer in contact with the mold wall starts to freeze. As the polymer cools it undergoes volumetric shrinkage. Hence, as the polymer in that cross section starts to shrink it requires more polymer material at the core section to compensate for the shrinkage. If molten polymer is not provided to this section it tends to pull the frozen skin layer towards the core leading to a sink mark as seen in Figure 4.

Due to the nature of the sink mark, light is reflected in different angles which makes them visible to eye. This visual defect makes a part aesthetically unacceptable for

all fascia applications. Since the objective of this study is to try and incorporate the wire into trims, which are visible and run throughout the automobile, sink mark reduction and elimination is important.

If there is an insert present within the mold, it acts as another heat sink and the polymer around the insert also cools and solidifies. Now at that cross section in a relatively thin section we have 2 frozen layers of plastic.

In this study, a wire is inserted in the rib cross section. In this structure, the wire now acts as a heat sink as the plastic melts and inhibits shrinkage around the pin as observed by S Fathi and A. H. Behravesh [7].

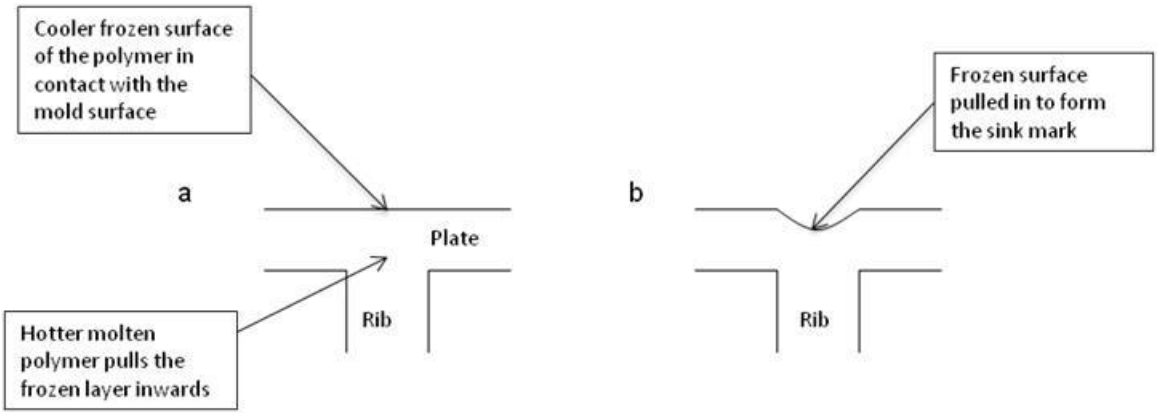

Figure 4 Sink mark formation

Injection Molding Parameters

The quality of the part depends on the processing conditions used to make the part. To control the sink mark the parameters which are easy to control and have a major impact are as follows:

1. Polymer material: Shrinkage characteristic of the polymer needs to be taken into consideration while selecting a material for an application if the part dimension is critical. Depending on the structure of the thermoplastic material, (amorphous or semi-crystalline) some insight can be had in predicting the behavior of the polymer as it is processed. Chang and Faison [8] observed that High density polyethylene (HDPE) a semi-crystalline plastic shrinks more than General purpose polystyrene (GPPS) and Acrylonitrile butadiene polystyrene (ABS) which are amorphous. They also observed that across the flow shrinkage for HDPE than along the flow direction. The converse is true for GPS and ABS.
2. Mold Temperature: It is the surface temperature of the mold walls. The mold temperature along with the melt temperature determines the rate of cooling of the polymer being molded. Higher mold temperature leads to longer cooling times but at the same time post mold shrinkage decreases and the molded part is more stable and durable.
3. Melt Temperature: It is the temperature of the melt. The higher the melt temperature, the lower is the viscosity of the melt which makes it easier for the melt to flow while filling the mold cavity. It also means that the part needs longer cooling before it is ejected.
4. Injection rate and pressure: It is the speed and pressure at which the material is filled in the cavity. If the speed on injection is too slow, the material begins to cool before the cavity is filled and the pressure required goes up. Also the gate freezes off too soon which prevents packing phase from completing. If the speed

is too high, it leads to rise in the melt temperature which causes more shrinkage as the part cools. High injection pressures can improve the surface finish of a part but at the same time, too high could lead to mold opening and flashing.

5. Packing pressure and time: Increasing packing pressure and time decreases shrinkage. The packing pressure should be high enough else it leads to reverse flow of the polymer leading to excessive shrinkage near the gate region. Similarly too short packing time also leads to reverse flow if the gate does not freeze of.
6. Cooling temperature and time: if the part is ejected at higher temperatures where the center of thicker sections is still in transition stage, the ejector pins may leave their marks or deform the part. If allowed to cool for too long, molded in stresses develop inside the part leading to post mold warpage. Also large cooling time leads to increased cycle time which may be uneconomical for mass production.

CHAPTER THREE

COMPUTER AIDED ENGINEERING

Computer Aided Engineering

Today use of computers in decision making stages is unavoidable. Using computer simulations for engineering applications in known as CAE or computer aided engineering.

A physical process can be represented as a mathematical model [9]. A mathematical model is generally a set of equations usually differential in nature. Computer can be used to solve these equations using numerical methods if certain conditions at particular locations are known. This can be achieved by using finite element analysis (FEA) which involves discretizing the geometrical space into set of points which are nodes. Nodes are further connected to form elements. These elements define the geometry of the part. The elements form a mesh. If the part is 2D, the mesh is made up of triangular or quadrilateral elements where as the elements are tetrahedral in case of 3D. These models now represent the geometry and the mathematical models are applied and solutions for the equations are used as a way to simulate the process. Computer codes using numerical methods are generally required for solving the complex mathematical models [10].

To use CAE to fluid flow process, the principles of conservation of mass, momentum and energy are used. These equations are solved for each element. The

solution to these equations is found at each node for each time step to simulate the fluid flow.

The principle of conservation of mass is given by [12],

$$\nabla \bullet \bar{v} = 0 \tag{1}$$

where,

$\bar{v}$ is the velocity vector

The principle of conservation of momentum is,

$$\rho \frac{\partial \bar{v}}{\partial t} = \rho \bar{g} - \nabla p + [\nabla \bullet \eta \dot{\gamma}] - \rho [\bar{v} \bullet \nabla \bar{v}] \tag{2}$$

where,

ρ is the fluid density

$\bar{g}$ is the gravitational effect

p is the fluid pressure

∇ is the del operator $\nabla = i\frac{\partial}{\partial x} + j\frac{\partial}{\partial y} + k\frac{\partial}{\partial z}$, i,j,k are standard basis in R3 and x,y,z are the coordinate axis

η is the viscosity of the fluid

$\dot{\gamma}$ is the shear rate

The principle of conservation of energy is,

$$\rho C_p \left(\frac{\partial T}{\partial t} + \bar{v} \cdot \nabla T \right) = \beta T \left(\frac{\partial p}{\partial t} + \bar{v} \cdot \nabla p \right) + \eta \dot{\gamma}^2 + k \nabla^2 T \tag{3}$$

where,

C_p is the specific heat of the fluid

T is the fluid temperature

β is the fluid expansivity

k is the thermal conductivity of the fluid

Among all the polymer processing methods, CAE is most profitable in terms of investment of time and software charges to the savings made on actual making of the part on the machine, when used for injection molding process. Compared to other plastic manufacturing processes, the percentage of components made using injection molding process is far greater than any other like extrusion, blow molding, thermoforming etc. Also, experimenting for best results after a mold is made is too expensive. Compared to the cost of experimentation, investing in simulations to study the process before actually making any mold is far more profitable. Due to the low cost of simulation and advances in computational field more and more people turn to CAE for initial decision making. This has led to numerous companies having their own codes with their advantages and disadvantages. Moldex3d, Moldflow Plastic Insight are some of the better known software used for the analysis. This study is conducted using Moldflow Plastic Insight.

Moldflow Plastic Insight has modules like MPI/Flow, Cool, Warp to analyze different phases of injection molding, MPI/Fiber, Stress, Shrink to deal with specialized results and MPI/Gas, Mucell, Co-injection and Reactive Molding to simulate plastic molding processes other than injection molding. To study the insert molding process, MPI/flow and warp modules are used. The Flow module is used to study the filling and

packing phase and Warp module to calculate the warpage of the part at end of the cooling cycle.

Moldflow plastic insight (MPI) is a product suite designed to simulate the plastic injection molding process and its variants such as gas-assisted injection molding, injection compression molding and thermosets processing [11].

Working with Moldflow allows simulating the entire process including modeling, meshing, mesh editing, model validation, job setup, job control, results visualization, and report generation. It allows the freedom to choose the molding machine, material, mold cavity design, mold material, process parameters and ejection criteria before any decision is made. A major advantage of using Moldflow is its library over of 290 machine descriptions and more than 7,500 thermoplastic materials which gives the flexibility to choose and experiment different combinations without actually investing in creating lot of prototypes.

Finite Element Analysis Meshes

Moldflow allows for three kinds of meshes Midplane mesh, Fusion mesh and the 3D mesh as shown in Figure 5. Midplane mesh is used where the thickness of the part is ignored. The mesh is defined on the Midplane or the plane containing the centerline of the part by using triangular elements. Thickness properties are assigned to this plane and hence the name 2.5D model. Fusion mesh which is also called the Dual Domain™ is another 2.5D mesh. The part is modeled as outside surfaces or skin mesh with triangular

™ Dual Domain is a registered trademark of Moldflow Corporation

elements. The elements on the faces are aligned and matched. The distance between the surface mesh determines the thickness of the part. Flow is simulated on the top and bottom surface of the mold cavity. Using this mesh for simulation purposes gives the fastest results amongst all the three meshes. 3D mesh descritizes the whole part into tetrahedral elements and is a true representation of the part geometry. It is used when the part under consideration is thick and chunky with the ratio of thickness to width is 4:1.

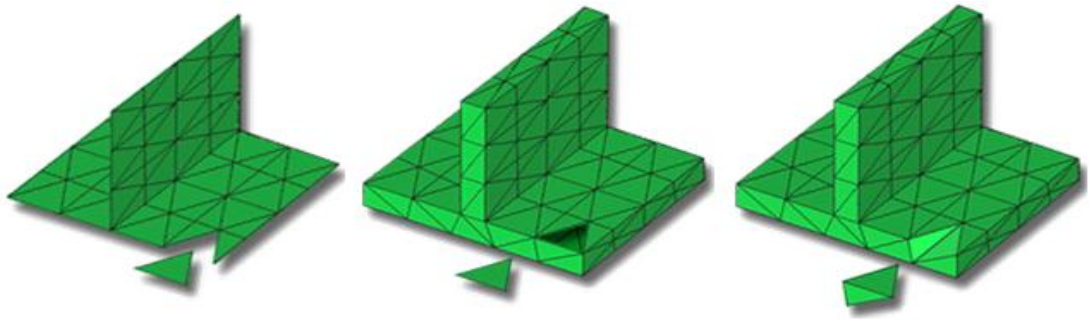

Figure 5 (a) Midplane Mesh (b) Fusion Mesh (c) 3D Mesh

For flow analysis using Midplane and Fusion mesh, Hele-Shaw flow model is used which makes the following assumptions:

1. Laminar flow of generalized Newtonian fluid
2. Inertia and gravity effects are ignored
3. In-plane heat conduction and convection is negligible compared to conduction in the thickness direction
4. Heat loss from edges can be ignored for triangular elements.

Making these assumptions especially to thick chunky parts would give misleading results and hence use of these meshes should be avoided. Instead a 3D mesh is used.

3D mesh make fewer assumptions are made compared to Midplane and Fusion meshs. 3D meshes use [12]:

1. Use full 3D Navier Stokes equation solver
2. Solves for mass, momentum, and energy conservation.
3. Solves for pressure, temperature and the three directional velocities at each node are calculated which lead to a more accurate solution to flow.
4. Considers heat conduction in all directions
5. Inertia and gravity effects are included in calculations
6. Extensional viscosity effects, pressure dependency of viscosity effects are considered.

3D mesh analysis uses the Navier-Stokes equation to study the molten plastic behavior. The Navier – Stokes equation describes the fluid motion. It is based on the principles of conservation of mass, momentum and energy [13].

The generalized form of this equation for compressible fluid flow is given as,

$$\rho\left(\frac{\partial \bar{v}}{\partial t}+\bar{v}.\nabla\bar{v}\right)=-\nabla p+\mu\nabla^2\bar{v}+f+(4\mu/3+\mu^{\nu})\nabla(\nabla.\bar{v}) \tag{4}$$

where,

ρ is the fluid density

$\bar{v}$ is the flow velocity vector

∇ is the del operator

p is the pressure

f is the body force acting on the fluid

μ^{ν} is second viscosity coefficient

For packing analysis, Moldflow uses the conjugate – gradient pressure solver for the partial differential equations. Conjugate gradient method is an iterative method for numerical solution of system of linear equations whose matrix is symmetric and positive definite [14].

Process definition for analysis

To simulate the process and obtain results comparable to the actual, defining the process in terms useful for simulation is critical. This involves modeling the material, the machine and the setting parameters for the machine. The Moldflow materials library carries the mathematical model defining the material properties of all major plastic manufacturers. The machine is defined in terms of its stroke length, screw diameter, clamping force and the likes. The machine library has well defined machines by major manufacturers along with some based on their tonnage.

The filling phase is defined using the injection velocity profile, the switchover point from filling to packing phase, mold and melt temperature. The injection velocity profile can be defined in terms of absolute or percentage of maximum ram speed or flow rate over either stroke length or time. The switchover is defined in terms of ram position, percentage of volume filled, injection or hydraulic pressure and injection time.

The packing phase is defined using the packing pressure profile. The packing phase is described by hydraulic, injection or percentage of maximum machine pressure over time in seconds. Definition of the process ends with the cooling time or the ejection criteria.

CHAPTER FOUR

SIMULATION METHODOLOGY

Outline

As seen earlier, to control the sink depth we have multiple factors to deal with such as,

1. Material for molding
2. Part geometry
3. Process parameters namely
    a. Mold temperature
    b. Mold temperature
    c. Packing pressure and time
    d. Cooling time
    e. Injection pressure and time
    f. Switchover point from filling to packing time

From previous research in this field, geometry and material selection play a major role in the shrinkage behavior of the molded part. Chang and Ernest [7] found that material plays a major role in shrinkage values. Alam and Kalam [15] used runner balancing of a part of the mold geometry to optimize shrinkage. Smith and Nam [16] found that changing the rib entrance shape and inducing unequal flow rates from the two sides of the rib reduced the sink depths by 33%. Therefore, the geometry and material are major factors whereas process parameters are used in fine tuning the process. The strategy adopted here is first

to compare the effect of insert molding a wire on the sink by simulating the injection molding process for a part with and without the wire in it. Next the effect of process parameters on sink depth is analyzed for different geometries followed by effect of geometry on sink marks. From these trials we hope to conclude the best geometry and parameter combination for best sink mark characteristics.

Since we have multiple parameters to study, the Design of Experiments (DOE) technique is implemented to do the same systematically. As seen in the previous sections, sink marks can be reduced or eliminated by a correct combination of various parameters such as the part design (i.e. geometry of the part) process parameters, polymer material used for the application. A systematic way to study the effect of these parameters is by using the technique of DOE. DOE refers to experimental framework used to quantify indeterminate measurements of factors and interactions between factors statistically through observance of forced changes made methodically as directed by mathematically systematic tables [17]. DOE allows for systematic study of effect of parameters on a process. DOE is applicable in various scenarios such as achieving a desired target for the process which is similar to fine tuning a process, maximize or minimize the response or to reduce variation or to make the given process robust. In this study the aim is to achieve minimal or no sink marks for an insert molded plastic part.

Some of the common terms used when dealing with DOE are:

- Factors are the parameters that can be controlled and influence the performance of the final result.

- Levels of a parameter are the values or settings made of the factors to study in the DOE,

- Response of the DOE is the performance or the output achieved for the particular combination of factors set at particular level.

DOE technique is based on making deliberate changes to one or more factors in order to observe the effect these changes have on the response.

Material Selection

To improve the properties of a part, instead of using a single polymer a combination of two or more polymers is used. When the mixed polymers have different glass transition temperatures and the properties of the resultant material are an average of the individual finished polymer, then the resultant mixture is a polymer blend. ABS/PC is such a kind of blend. The properties of ABS/PC blend falls in between those of PC and ABS and is becoming increasingly popular due to its low cost high performance properties such as high processability of ABS and impact and heat resistance along with good mechanical properties of PC. Some of the major manufacturers are Bayer (Bayblend), Dow Chemicals (Pulse) GE Plastics (Cycoloy) amongst others.

Advantages of PC/ABS blends [18]:

1. Show good dimensional stability,
2. Low distortion and shrinkage,
3. Low moisture absorption,

4. High softening point (Vicat B = 112-134 ºC, increases with increasing PC content),
5. Stiffness and hardness comparable to that of PC,
6. Good notched impact strength down to —50 ºC,
7. Good electrical properties in the low voltage/low power range.

Bayblend FR110 [19] is a PC/ ABS blend by Bayer Polymers. It has desirable properties such as ultra-high melt-flow, impact strength and rigidity. Also it can be easily painted, metalized or in-mold decorated which makes more variations in shape, texture and color of the final product, an important consideration if the product is going to be a visible component with aesthetics having a major role. Bayblend FR 110 has good color stability, and maintains high impact strength even at low temperatures which makes it a good choice for automotive interior and exterior parts. Some of the examples of Bayblend being used in cars are as shown in Figure 6

Figure 6 VW Beetle interior parts, Central console cockpit on the Alfa 166 in Bayblend®

Geometry Selection

Since the motivation for the study is to try and implement the technique for automobile industry the geometry selected for study has to be similar to some commonly observed geometric configuration for plastic part. To improve the structural strength of a plastic part usually ribs are provided to the part. As mentioned earlier, the plastic could act as a structural support for thinner copper wires, we use a flat plate with a rib to house the wire as our test geometry.

The thickness of the plate is taken to be 3mm and the geometry of the rib would be changed to study its effect on the sink characteristics.

Guidelines for a rib suggest that the rib thickness should be less than 50% of the part thickness to prevent sink marks. Michael H. Brannan and Jerry R Nichols [20] used a geometric feature which reduced the nominal part thickness around both the sides of the rib so that the smaller cross section of the polymer will lead to faster cooling leading to lesser shrinkage and thereby no sink marks. They implemented this method to prevent sink mark while using a rib which was 72% of the wall thickness.

Four types of geometries are selected for study. A square plate (100 X 100 X 3 mm) is the base plate with a rib. Four different combinations of rib geometry and wire positioning are studied as:

A. Rectangular rib (5 X 100 X 1 mm) wire being centered in the plane of the flat base plate (Figure 7).
B. Rectangular rib (5 X 100 X 2 mm) wire is positioned such that it is enclosed in the rib section at all times (Figure 8).

C. Semicircular rib (100 X 3 mm) wire being in the centre plane of the flat base plate (Figure 9).

D. Semicircular rib (100 X 5 mm) is positioned such that it is enclosed in the rib section at all times (Figure 10).

In this study, the rib thickness is 166% of the part thickness which clearly violates the 50 % rule and hence the process parameters would play a major role in fine tuning the sink depths observed.

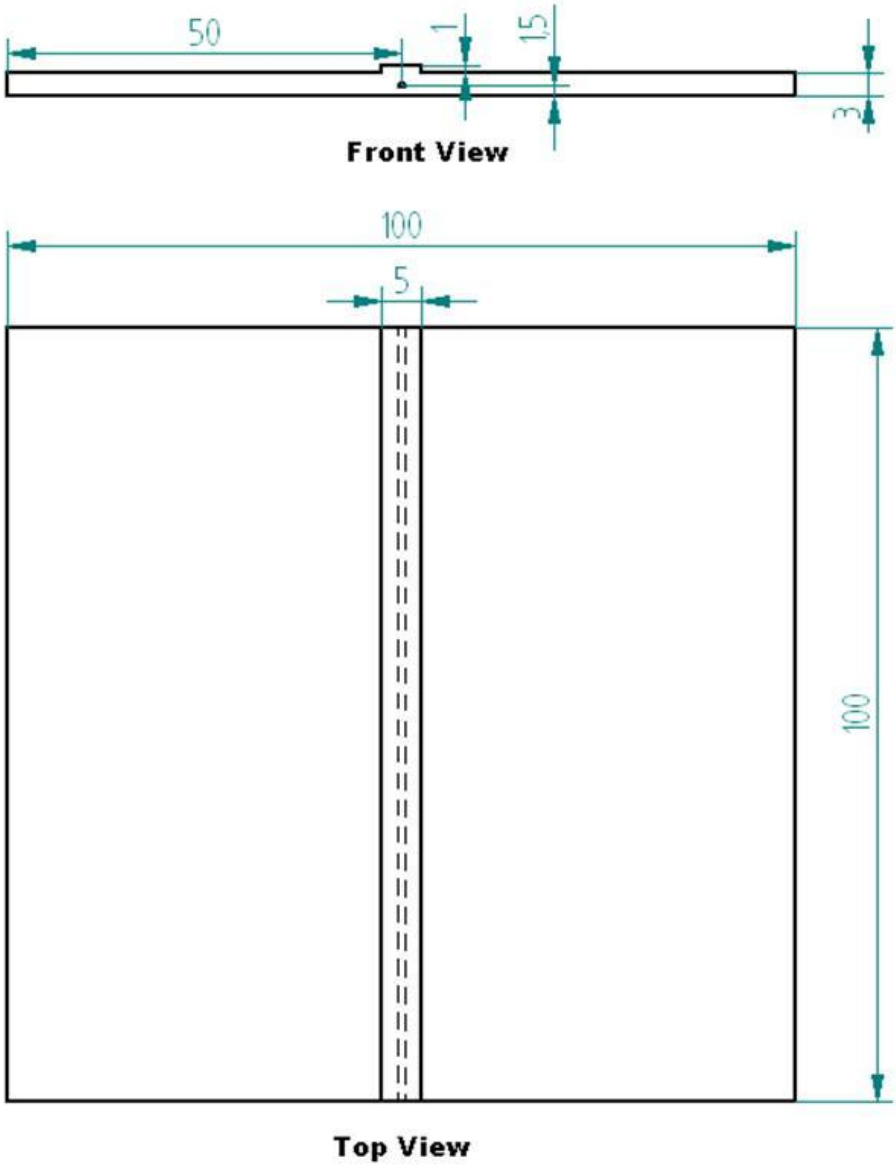

Figure 7 Configuration A: Rectangular rib; wire being centered in the plane of the flat base plate.

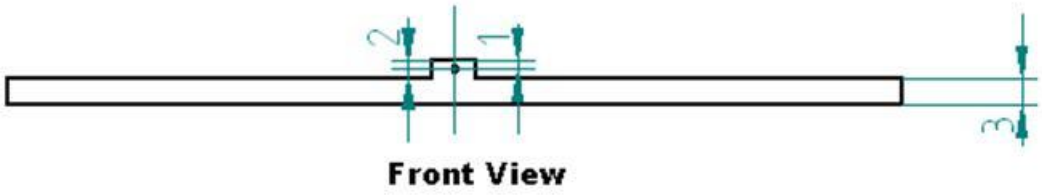

Figure 8 Configuration B: Rectangular rib; with wire completely within the Rib

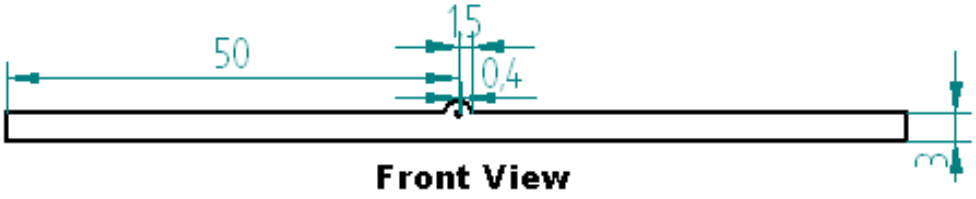

Figure 9 Configuration C: Semicircular rib; wire being in the centre plane of the flat base plate

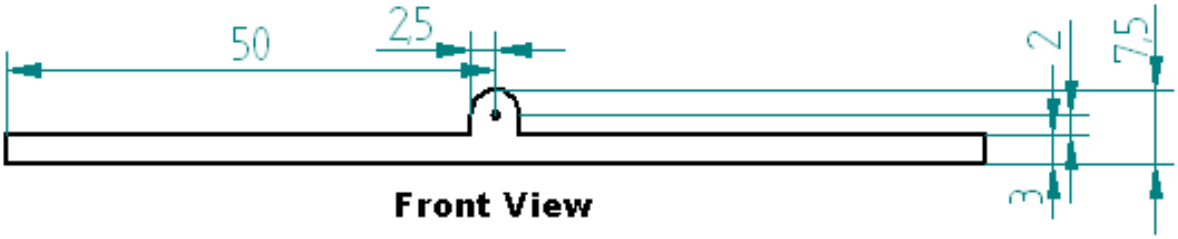

Figure 10 Configuration D: Semicircular rib; wire is positioned such that it is enclosed in the rib section

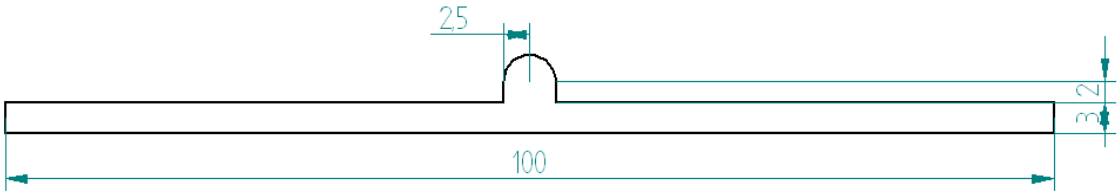

Figure 11 Configuration E: Semicircular solid rib

Design of Experiment

The Taguchi DOE technique is one standard and well accepted DOE technique proposed by a Japanese engineer Genichi Taguchi [21]. A major advantage of using this DOE technique is that it allows us to study a large number of parameters without having to conduct trials for all possible combinations. An important advantage of using this method is that the trial matrix is orthogonal in nature [22]. The Orthogonal array ensures

that no one factor is given more weight in the experiment compared to the other factors. Orthogonality ensures that the estimate of any particular factor on the response will not be distorted by the effects of other factors which means that the effect of each factor can be mathematically calculated independent of the other factors.

Employing Taguchi DOE to improve the injection molding process has been used successfully by various authors. This technique has been implemented by Tao C. Chang and Ernest Faison III [7] studied shrinkage behavior and optimization of injection molded parts, S. J. Liao et al. [23] used this method to find optimal process conditions for minimizing shrinkage and warpage or thin walled parts, K. M. B. Jansen, D. J. Van Dijk, M. H. Husselman [5] studied the effect of processing parameters on the shrinkage of injection molding process. Gaitonde V. N et al. [24] used this method for achieving better machinability in unreinforced and reinforced polyamides.

DOE starts by defining the objective and scope of the experiment followed by selecting the process parameters to be studied. Once the number of parameters under consideration is fixed, a suitable experimental design to study those effects in an economical fashion is made. The Taguchi designs range from L4 design for studying 3 factors at 2 levels in 4 trials to L81 design which is used to study 40 factors at 3 levels. It is obvious that these designs allow us to study large number of factors set at multiple levels in comparatively small number of trials. Once the experimental design is finalized the actual experimentation, which in our case being simulation, begins. The data collected is analyzed and the results obtained are used to improve the process.

From previous trials conducted the following factors are observed to affect the response (sink depth) the most and are hence selected for further study using the DOE:

a. Mold Temperature (°C)

b. Melt Temperature (°C)

c. Pack Time (sec)

The other factors such as injection profile, the switchover point, packing pressure the gate location all are kept constant. The cooling time affects the final warpage the most. Since the flow rate during the packing phase is very low, the packing time can also count in the total cooling time the material has to cool within the mold. To equalize the effect of cooling time for all the parts, instead of defining the ejection criterion as the cooling time in seconds, the temperature of the mold is used. Once the mold reaches a specified temperature, the part is ejected.

We have selected 3 factors and using 3 levels for each factor, we have 2 possible designs L9 and L27. We select L9 which needs just 9 trials to study the effect of these parameters. The levels for Melt temperature factor are selected from the manufacturer's recommendation range. The level settings for Mold temperature and Pack time factors are selected using the initial simulation results. A standard L9 Taguchi DOE matrix along with the actual values for the process parameters is as shown in Table 1.

The trials are run in a random fashion when on a machine so that there is no lingering effect of the previous trials on the new ones. As simulation is used to study the results instead of a machine, the results remain the same irrespective of the order in

which they are carried out. Performing the analysis in a random order has been ignored in this study.

Trial Number	**Levels of Process Parameters**			**Actual Values of Process Parameters**		
	Factor A	Factor B	Factor C	Mold Temp (°C)	Melt Temp (°C)	Pack Time (sec)
1	1	1	1	70	260	8
2	1	2	2	70	270	10
3	1	3	3	70	280	12
4	2	1	2	90	260	10
5	2	2	3	90	270	12
6	2	3	1	90	280	8
7	3	1	3	110	260	12
8	3	2	1	110	270	8
9	3	3	2	110	280	10

Table 1 Orthogonal array and experimental layout plan for L9 model

Process definition for simulation

To define the process for a CAE analysis, we need to define the material, the machine, the geometry and the processing parameters for the machine in a fashion so that mathematical analysis is possible. Steps followed in conducting the DOE trials in Moldflow are:

1. The geometry created in CAD software in imported and meshed. Care is taken to ensure that the mesh is fine enough for the desired geometry. The geometry of the

part is defined using a 3D mesh with an element size of 1.5mm with a cord length control of 0.1 mm. The runner system is as shown in the Figure 12. The runner is 10 mm in Y direction and 10 mm in Z direction. Beam elements are used to define the runner system. The gate is located at end of the runner and the mold is a single cavity mold. The gate is such so that the flow of the melt is always parallel to the wire. The mesh is then checked for any errors as a fault in the basic mesh will lead to faulty and misleading results.

2. Mold material is set to be Tool steel T20.
3. The wire is treated as a cavity with the properties same as those of the mold material.
4. The plastic is selected from the pre-defined Moldflow material library.
5. The injection molding machine to be used to simulate the process is chosen from the pre-defined Moldflow library.
6. The injection molding process is defined using the process parameters like:
    a. mold and melt temperatures,
    b. injection control using injection time in our case,
    c. switchover point from injection to pressure phase,
    d. packing phase pressure profile,
    e. cooling criterion which is the mold temperature in these simulations,

Figure 12 Mesh for Geometry A

The parameters set at constant level for the simulation are:

Injection pressure	= 1.8000E+02 MPa
Clamp force	= 7.0002E+03 MPa
Injection rate	= 5.0000E+03 cm^3/s
Atmospheric temperature	= 25.00 °C
Velocity/pressure switch-over	= 99.00 % volume filled
Packing Pressure	= 8 Mpa
Ejection temperature for cool time	= 110.00 °C

Sink depth metrics

As mentioned earlier, sink mark is an aesthetic defect. Here the metric of interest is that change in deflection of the molded part which makes the depression visible. This means that we are not interested in the deflections due to warpage of the part but rather that deflection which can be felt to touch or seen by the eye. A warped surface is smooth to touch but a sink mark is felt to touch.

At the end of the injection molding simulation using Moldflow, deflection of the part along Z axis is obtained as a contour plot as shown in Figure 13. This plot shows the deflection of each node with respect to its original position. The maximum deflection node for the bottom face is identified using the query plot in Moldflow. Once the node is identified, a deflection path plot for the selected XZ plane which contains that node is generated. A path plot thus generated give the deflection (mm) Vs x coordinate (mm) plot for the selected plane. The data obtained thus is used to further analyze the sink mark. Figure 14 shows the deflection path plot results for the XZ plane.

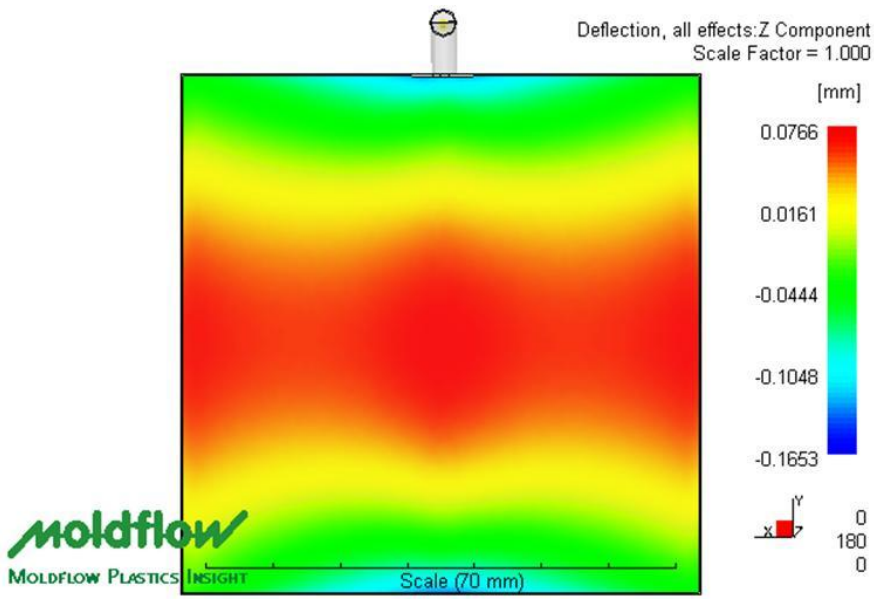

Figure 13 Deflection plot for Geometry C trial 2

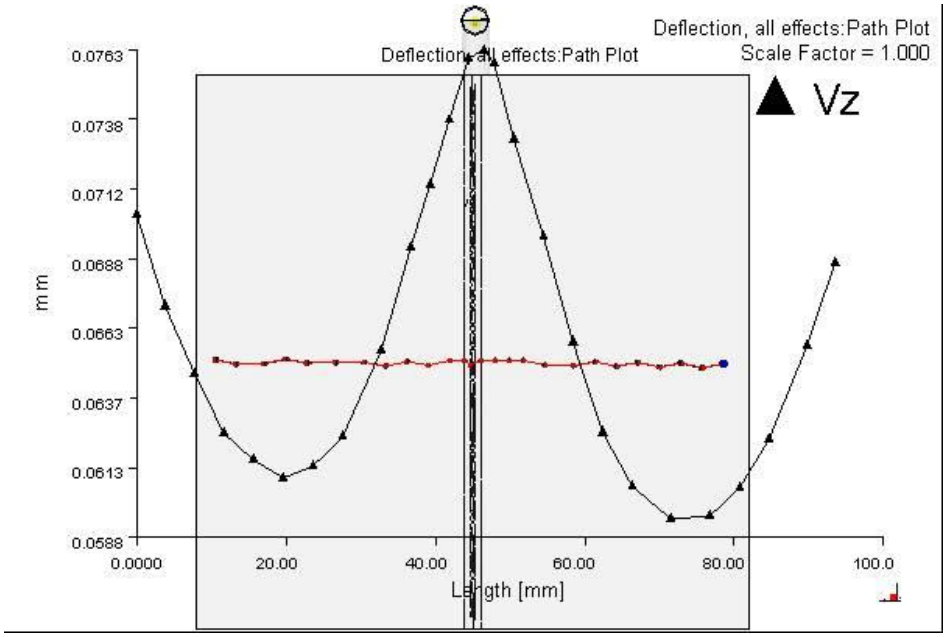

Figure 14 Deflection in Z direction path plot

Now, using Minitab, regression analysis is conducted and a best fit line is obtained for the data points. It is seen that cubic regression model gives better results than

linear of quadratic models; hence, used here. Sink marks influence the warpage of the part, hence, if there is a sink mark present it will lead to deflection values greater than those observed had there been no sink. Hence we need to differentiate between the deflections due to sink and warpage.

If the part is big enough compared to the region of sink, then we can assume the deflection of the part in regions away from the center (where the sink is seen) is due to warpage alone. To use this assumption to calculate the sink depth, those data points with deflection values that do not fall in the general trend (usually 15 mm from center) are deleted until we get a line which has R^2 value, the coefficient of fit, greater that 95%. This gives an equation which defines the surface to the part had there been no sink mark on it. Using the x-coordinate at the point of maximum deflection and using the equation obtained from the above assumption the deflection value of the part at that point having warpage alone is calculated. The difference between the observed value in simulations and the calculated value is the sink depth.

Sink depth calculated in this fashion is used for further analysis.

CHAPTER FIVE

RESULTS AND DISCUSSION

Sink depth is calculated as described in Chapter 4. A sample calculation following the mentioned steps is as follows:

Neglecting the points in the vicinity of the sink and only using the points located at the edges of the part alone, a line is fitted through them to get the an equation of line which mathematically describes the warpage. Here as seen from Figure 15, the R^2 value is 97.2% which is a good fit and hence, the equation for warpage thus obtained is acceptable for further calculations.

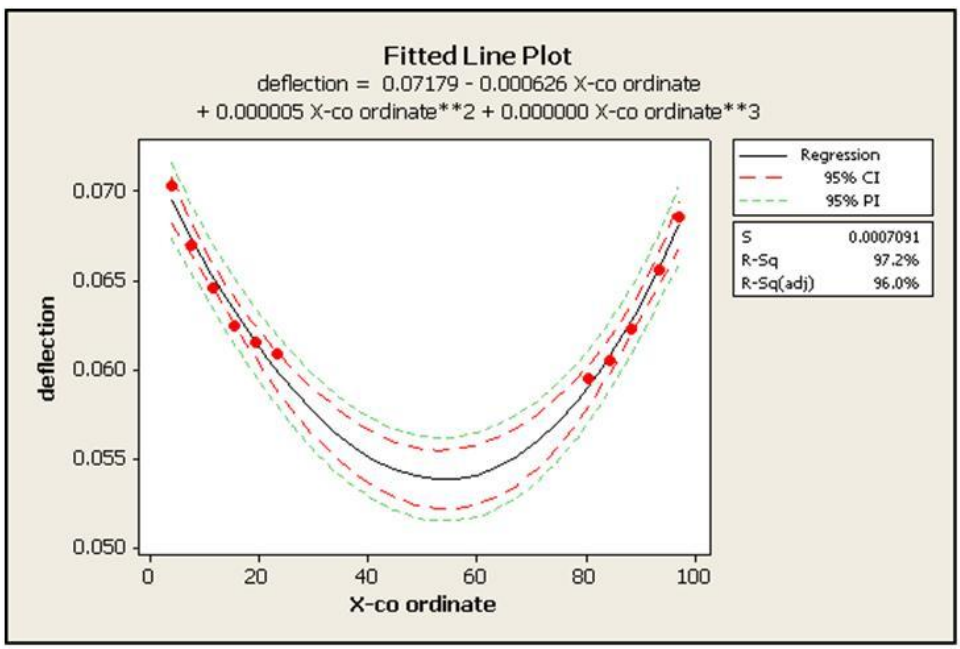

Figure 15 Minitab plot for best line fit

From the Figure 15, we get the equation of the line as:

$$def = 5.198*10^{-9}*x^3 + 6.478*10^{-6}*x^2 - 7.22*10^{-4}x + 0.07243 \quad (5)$$

where,

def = deflection (mm)

x = x coordinate (mm)

The equation of the line obtained for the data now represents the deflection due to warpage alone. Next the x coordinate of the node with maximum deflection is identified and using the equation (5) obtained above the deflection of the node due to warpage alone is calculated. The difference is the values for the calculated deflection above and that seen from the simulation gives the sink depth.

Here, the maximum value was observed for the node with x = 50.227 mm. Substituting in the above equation the result for sink depth can be obtained as,

Deflection due to warpage	0.053173 mm
Deflection observed from simulation	0.076226 mm
Sink Depth	0.023053 mm

Hence the sink depth for this particular trial is 0.023053 mm.

The sink depth calculated thus is used in further analysis.

Analysis of Taguchi DOE

To analyze the Taguchi DOE, the method of Analysis of Means (ANOM) is followed. The analysis of means (ANOM) is a graphical procedure for comparing a collection of means, rates, or proportions to see if any of them differ significantly from the overall mean, rate, or proportion [25]. The ANOM is a type of multiple comparison procedure. The calculation steps are as follows:

1. The averaged response values calculated as above are filled in the response observed column.
2. The rest of the columns are filled such that the response of the trial for that setting of the parameter is entered. These places to be filled correspond to white cells in the table.
3. The totals of each column are entered.
4. Average of each column is calculated.
5. Response of the experiment for each parameter is calculated by subtracting the average corresponding to the low parameter setting from the medium parameter setting (i.e. col(2)-col(1) for each case), medium parameter setting from the high parameter setting (i.e. col(3)-col(2) for each case), low parameter setting from the high parameter setting(i.e. col(3)-col(1) for each case),
6. This response can also be seen graphically by plotting the response values for each parameter setting.

A sample calculation following the above steps is shown in Table 2. The response values i.e. the sink depth values calculated are in mm.

Trial No	Response	Mold Temp			Melt Temp			Pack time		
		70	90	110	270	290	310	8	10	12
		1	2	3	1	2	3	1	2	3
1	0.027323	0.027323			0.027323			0.027323		
2	0.029927	0.029927				0.029927			0.029927	
3	0.030924	0.030924					0.030924			0.030924
4	0.020984		0.020984		0.020984				0.020984	
5	0.048974		0.048974			0.048974				0.048974
6	0.021213		0.021213				0.021213	0.021213		
7	0.013446			0.013446	0.013446					0.013446
8	0.01276			0.01276		0.01276		0.01276		
9	0.010441			0.010441			0.010441		0.010441	
Total	0.215992	0.088175	0.091171	0.036646	0.061754	0.09166	0.062577	0.061295	0.061352	0.093344
Average	0.023999	0.029392	0.03039	0.012215	0.020585	0.030553	0.020859	0.020432	0.020451	0.031115

Table 2 Sample calculation table for sink mark depth for geometry A

Effect of insert molded wire on sink mark depth

We compare the response values for 2 experiments one conducted with Geometry C with and without the wire inserted. The calculations as in Table 2 are repeated for these two trials and the results are tabulated as follows:

Trial No	Sink depth in mm	
	Geometry C	Geometry E
Trial 1	0.0008	0.0166
Trial 2	0.0028	0.0166
Trial 3	0.0028	0.0121
Trial 4	0.0109	0.0127
Trial 5	0.0093	0.0152
Trial 6	0.0005	0.0152
Trial 7	0.0003	0.0099
Trial 8	0.0005	0.0116
Trial 9	0.0022	0.0134

Table 3 Sink mark depth calculations for geometries C and E

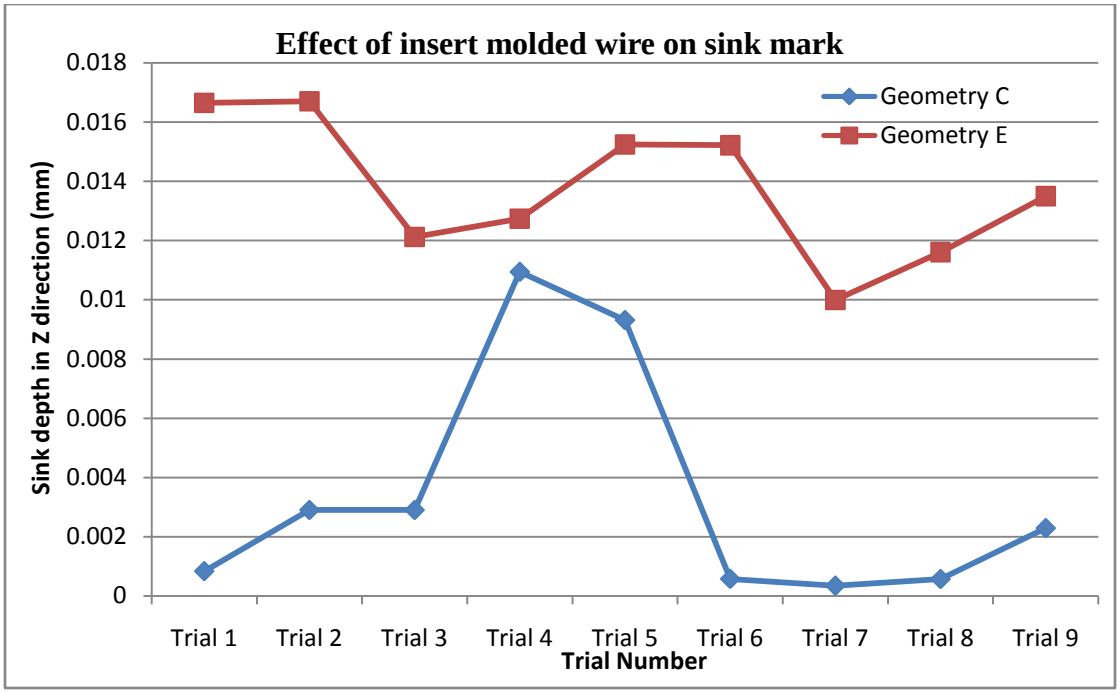

Figure 16 Effect of insert molded wire on sink mark

From Table 3 and Figure 16 it can be clearly seen that the response (the response here being the sink mark depth) of the process for geometry with the wire insert molded is better than that without a wire in it.

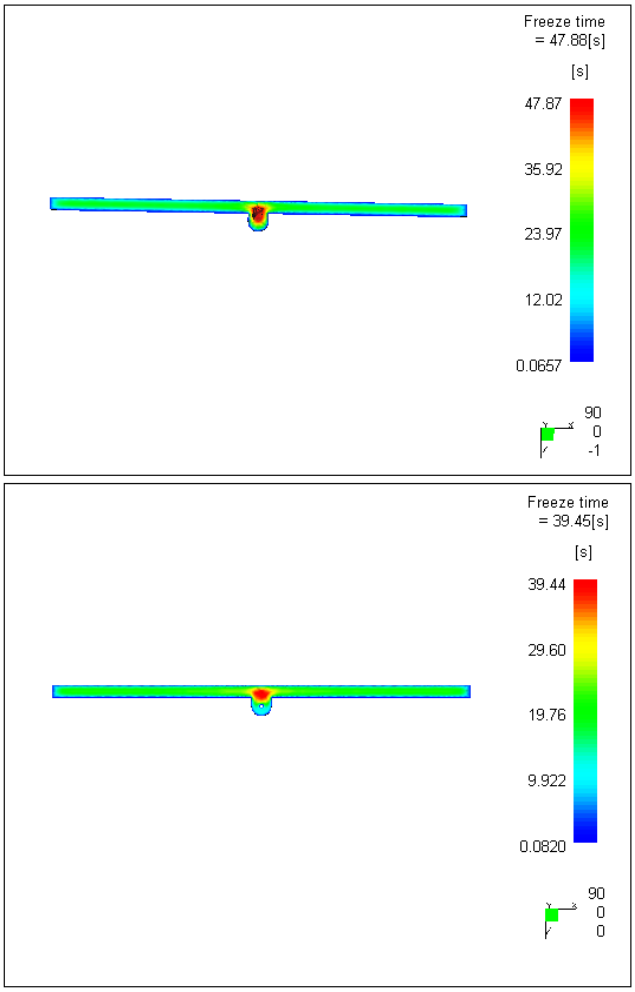

Figure 17 Freeze time profile for Geometry E and C

Figure 17 shows the time taken for the cross section to freeze. Top part shows the time taken to freeze by geometry E which has a solid rib and the bottom half shows for geometry C. As mentioned earlier, the sink mark is created when the core of the part is molten and the surface freezes off. By inserting a wire in the mold, an accumulation of large molten polymer mass at the base of the rib is avoided. Since the wire itself acts as a heat sink and removes heat from the polymer melt, the creation of a region or core with molten polymer between 2 frozen layer decreases leading to minimization of sink mark

formation. The Figure 17 shows that the polymer just below the surface at the location of the rib takes 47 seconds to freeze as against 39 seconds when a wire is present. It is observed that the amount of molten polymer between the two frozen layers at the rib location is lesser when a wire is present.

Effect of parameters on sink mark depth

The averages at the bottom of the Table 2 are the response values of the process for that parameter setting. Repeating these calculations for all the other geometries, we summarize the results as in Table **4**.

Factors		**Geometry A**	**Geometry B**	**Geometry C**	**Geometry D**
Mold Temperature (°C)	70	0.0293	0.0138	0.0228	0.0022
	90	0.0303	0.0137	0.0141	0.0069
	110	0.0122	0.0141	0.0104	0.0010
Melt Temperature (°C)	260	0.0205	0.0139	0.0138	0.0040
	270	0.0305	0.0137	0.0172	0.0042
	280	0.0208	0.0139	0.0162	0.0019
Pack Time (sec)	8	0.0204	0.0136	0.0170	0.0006
	10	0.0204	0.0143	0.0134	0.0053
	12	0.0311	0.0137	0.0169	0.0041

Table 4 Sink mark depths for parameters for geometries A, B, C, D

The above table gives averaged response of the process to that parameter for its high medium and low settings. To compare these results better, the graphical representation of these results would be helpful. Here we are interested in reducing or eliminating the sink marks, thus the criterion for judging the results would be smaller the better. The response values for each factor are plotted and the responses for the high, medium and low setting for the factor are connected with lines. Lines with slopes indicate that the factor level affects the response. The steeper the slope, the more pronounced the effect of the factor on the response. For ease of comparing the results all the geometries are plotted alongside each other as in Figure 18

From the graphs as in Figure 18 the following observations are made for the geometries:

1. The geometry of the rib plays a role in the setting needed for the parameters.
2. Mold temperature affects the variation in the response values the most. Higher the mold temperature the better the response.
3. Melt temperature setting needs to be in comparison with the mold temperature. High mold and melt temperatures give better results, but the exact setting varies for each geometry.
4. Low packing time gives better results for all the geometries. This can be due to the fact that lesser material packed leads to lesser shrinkage.

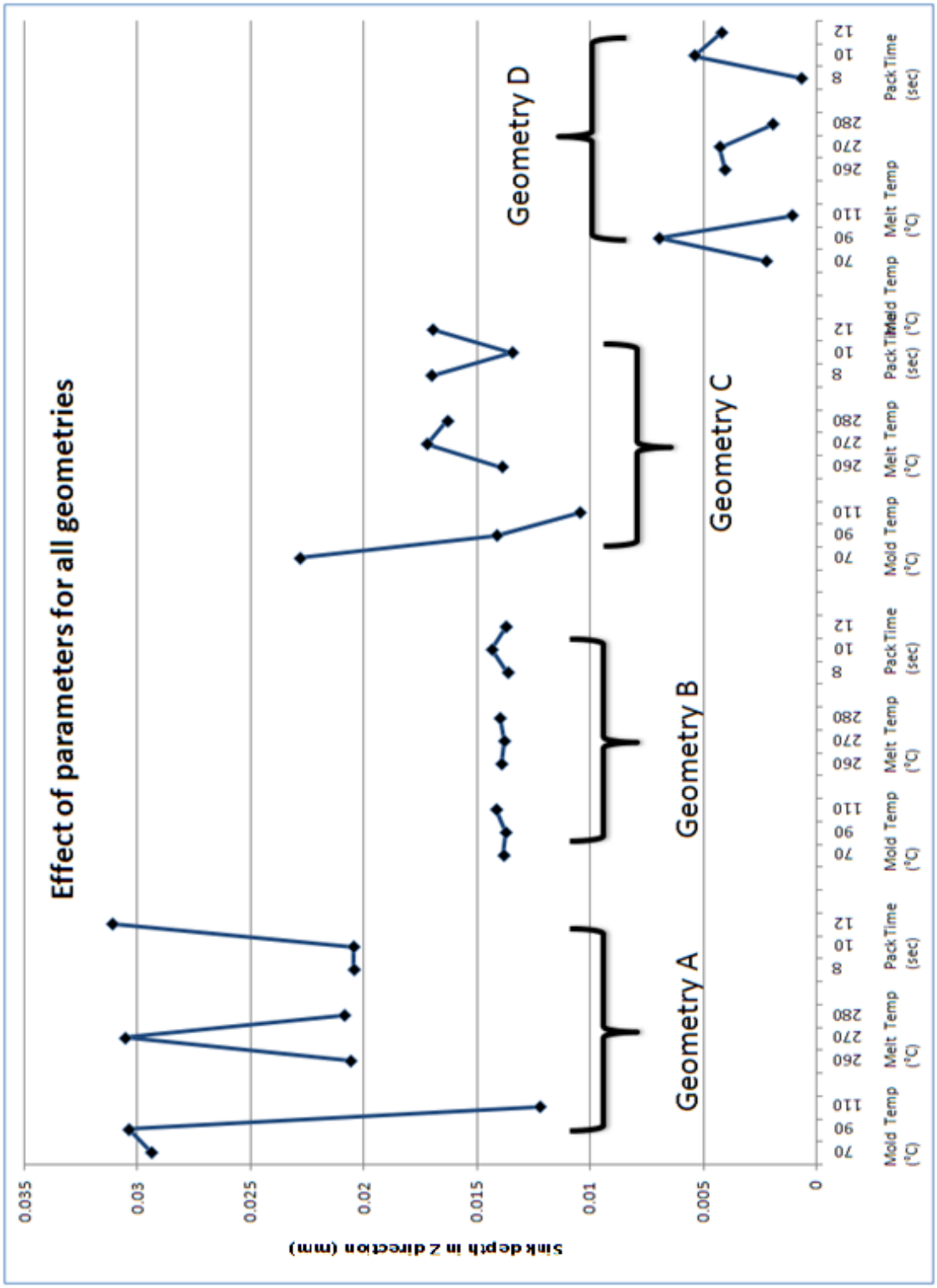

Figure 18 Effect of parameters on sink marks for all geometries

Effect of geometry on sink mark depth

In the same fashion as shown in Table 2, calculations are done for all the other geometries. A convenient table for all the geometries is as follows in Table 5:

Trial No	Sink depth in mm			
	Geometry A	Geometry B	Geometry C	Geometry D
Trial 1	0.0273	0.0130	0.0222	0.0008
Trial 2	0.0299	0.0143	0.0230	0.0028
Trial 3	0.0309	0.0139	0.0231	0.0028
Trial 4	0.0209	0.0144	0.0085	0.0109
Trial 5	0.0489	0.0129	0.0168	0.0093
Trial 6	0.0212	0.0137	0.0170	0.0005
Trial 7	0.0134	0.0142	0.0108	0.0003
Trial 8	0.0127	0.0139	0.0117	0.0005
Trial 9	0.0104	0.0141	0.0087	0.0022

Table 5 Sink mark depths for all geometries

To compare these results between geometries, we graphically plot the deflections Vs Trial number for all geometry as in Figure 19

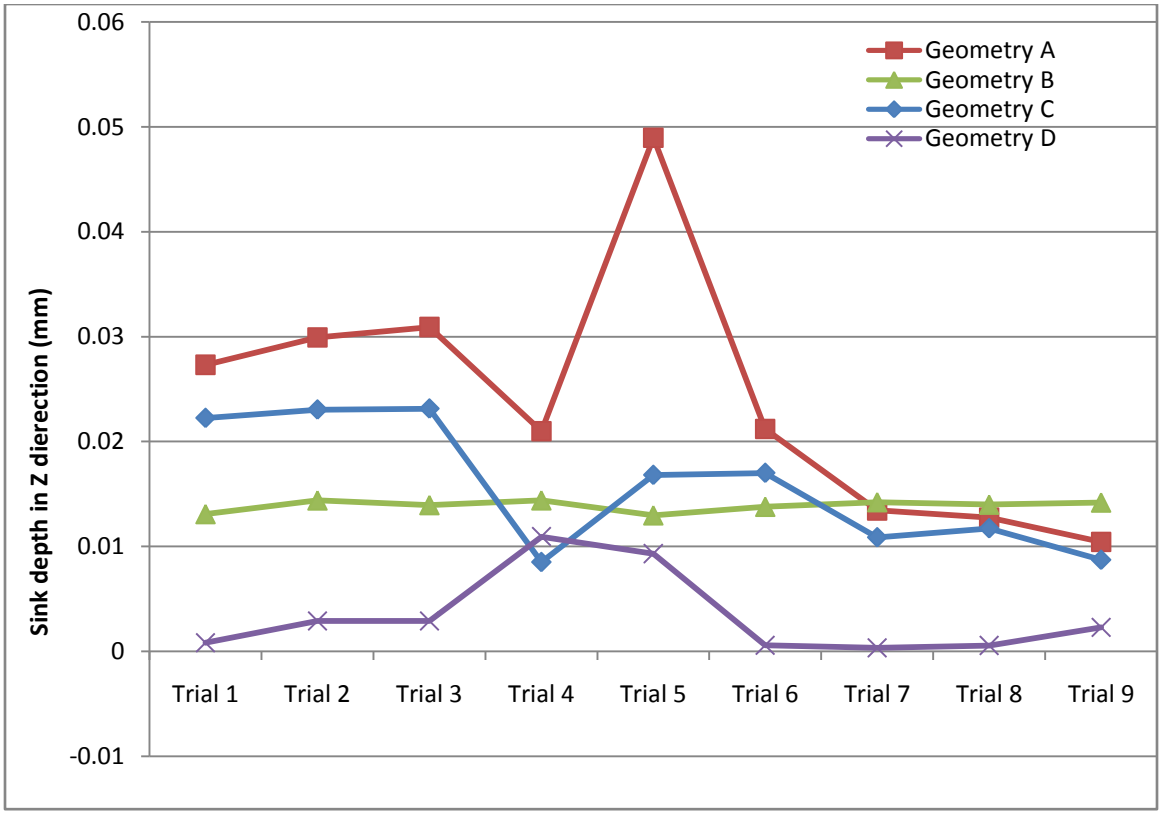

Figure 19 Comparison of sink depth results for various geometries

As seen from the Figure 19 the following observations are made:

1. Geometry A has the worst response where as Geometry D has the best response.
2. Geometries B and D wherein the inserted wire is completely enclosed in the rib section show the best response. With the wire completely in the rib the filling pattern allows for the melt to flow unobstructed in the cavity making the packing phase work better than in the other two cases.
3. Geometry A has a very slight rib but, the location of the wire makes it act as an obstacle to the fluid flow due to which the sink mark observed is the deepest for it.

4. A circular rib shows comparatively better result than a rectangular rib due to the absence of corners in the rib.
5. Geometry B with a wide rib with wire insert completely located completely within the rib shows almost no appreciable variation across the trials. The geometry is thus robust and fairly insensitive to the process parameters.
6. For geometries A and C the sink can be controlled by controlling the process parameters. Trials 6, 7, 8 and 9 show better sink than the rest. These are those trials where the mold temperature was set to high. Trial 8 which has the best results has high mold temperature high melt temperature and low pack time. This observation is consistent with those made earlier.

CHAPTER SIX

CONCLUSION

This study was focused at reducing or eliminating sink marks from wire insert molded components. The measure of sink mark was taken as their depths as felt to the hand run over the surface of the part. The effect of wire, geometry of the rib and processing parameters on the sink mark depth was investigated.

The results showed that the presence of wire in a broad rib helped reduce the sink marks by acting as another heat sink. It inhibits excessive shrinkage around itself. In geometries with wire acting as an obstacle to the flow led to deeper sink marks compared to those cases where the geometry allowed the flow in the base plate to be unobstructed. I was noted that a proper location of the wire in the rib i.e., its location from the desired surface plays a major role in dictating how the sink mark will be affected by it. Keeping the wire in that location by use of clamping devices and the ability of the device to keep the wire from moving around in the mold during the injection and cooling phase are critical from the manufacturing point of view.

It was concluded that by controlling the process parameters, the depth of a sink mark can be controlled. The major factor that controls the response depends on the rib geometry in use. Further, it was seen that higher mold temperatures lead to better response with increase in molding cycle time.

Future work in this direction will include investigating further by varying the part size, gate locations, wire dimensions, material for molding and other process parameters (such as packing pressure, a changed pressure profile, injection profile, a different

cooling profile with the use of cooling channel etc). A design to address the routing of multiple wires in a part would be another future research.

REFERENCES

1 BMW, Seamless Wiring Project Summary, 2005, pp.3-4

2 Handbook of electronic tables and formulas for American wire gauge

3 Fischer, J.M. (2003), Handbook of Molded Part Shrinkage and Warpage, William Andrew Publishing/Plastics Design Library

4 www.methode.com

5 Oswald, Tim A. and Menges Georg, "Materials science of polymers for engineers", Hanser Publishers

6 Jansen, K.M.B, Van Dijik, D.J., and Husselman, M. H. "Effect of Processing Conditions on Shrinkage in Injection Molding" Polymer Engineering and Science, Vol. 37, Issue 5, pp. 838-846, May 1997

7 Fathi, S., Behravesh, A.H., "Visualization of in-mold shrinkage in injection molding process", Polymer engineering and science, Vol. 47, Issue 5, pp.750-756, May 2007

8 T.C. Chang and E. Faison, 'Shrinkage behavior and optimization of injection molded parts studied by the Taguchi method' Polymer engineering and science, Vol. 41, Issue No. 5, pp.703-710, May 2001

9 Modern Plastics; Harper, Charles A. Modern Plastics Handbook. McGraw-Hill. Online version available at: http://knovel.com/web/portal/browse/display?_EXT_KNOVEL_DISPLAY_bookid=1008&VerticalID=0

10 Rosato, Dominick V.; Rosato, Donald V.; Rosato, Marlene G.," Injection Molding Handbook (3rd Edition)" Springer - Verlag.
Online version available at:
http://knovel.com/web/portal/browse/display?_EXT_KNOVEL_DISPLAY_bookid=1009&VerticalID=0

11 http://www.moldflow.com

12 Jay Shoemaker(Ed.) Moldflow Design Guide: A Resource for Plastics Engineers, Hanser

13 P. K. Kennedy, Flow Analysis of Injection Molds, Hanser, New York, 1995.

14 Hestenes Magnus R. and Stiefel Eduard, "Methods of Conjugate Gradients for Solving Linear Systems", Journal of Research of the National Bureau of Standards, Vol. 49, Issue No. 6, pp. 409-436, December 1952

15 Kevin Alam and musa R. Kalam,"Runner balancing by genetic optimization of shrinkage", Polymer Engineering and science, Vol 44, Issue No. 10

16 Kenneth L. Smith, Nam P. Suh, "Kenneth L. Smith, Nam P. Suh", Polymer Engineering and science, Vol 19, Issue No. 12, pp 829-834

17 http://www.isixsigma.com/dictionary/Design_of_Experiments_-_DOE-41.htm

18 Utracki, L.A. Commercial Polymer Blends. Springer - Verlag.

19 http://amco.ws/plastics/bayer/bayblend.asp

20 Michael H. Brannan and Jerry R. Nichols ‘Sink mark prevention feature for injection molded plastic parts’ Technical report [online]
Online version available at :
http://www.priorartdatabase.com/IPCOM/000008465/

21 *NIST/SEMATECH e-Handbook of Statistical Methods*,
http://www.itl.nist.gov/div898/handbook

22 Robert H. Lochner, Josheph E. Matar: Designing for Quality – An introduction to the best of Taguchi and Western methods of statistical experimental design

23 S. J. Liao, D. Y. Chang, H. J. Chen, L. S. Tsou, J. R. Ho, H. T. Yau, W. H. Hsieh, James T. Wang, Y. C. Su, “Optimal process conditions of shrinkage and warpage of thin-wall parts”, Polymer Engineering and science, Vol 44, Issue No. 5, pp 917-928

24 Gaitonde V. N., J. Paulo Davim, Karnik S. R., Francisco Mata, “Taguchi Approach for Achieving Better Machinability in Unreinforced and Reinforced Polyamides”, Journal of Reinforced Plastics and Composites, Jan 2008

25 http://www.analysisofmeans.com/