SINK MARKS DEFECT ON INJECTION MOLDING USING DIFFERENT RAW MATERIALS

MOHD NAZRI BIN NOORDIN

BACHELOR OF ENGINEERING

UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG
FACULTY OF MECHANICAL ENGINEERING

I certify that the project entitled "*Sink Marks Defect on Injection Molding using Different Raw Materials*" is written by *Mohd Nazri bin Noordin*. I have examined the final copy of this project and in my opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. I herewith recommend that it be accepted in partial fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering with Manufacturing Engineering.

Pn Shahida Bt Mohd Shalahim	....
Examiner	Signature

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 40^0c, melt temperature at 200^0c 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 70^0c, Melt temperature at 190^0c 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 40^0c, suhu lebur pada 200^0c dan masa padatan 1s. Manakala untuk bahan PE, parameter optimum telah didapati adalah tekanan padatan pada 75 MPa, suhu acuan pada 70^0c, suhu lebur pada 190^0c dan masa padatan pada 1.5s.

TABLE OF CONTENTS

CHAPTER 1 INTRODUCTION

CHAPTER 2 LITERATURE REVIEW

CHAPTER 3 METHODLOGY

CHAPTER 4 RESULTS AND DISCUSSION

CHAPTER 5 CONCLUSIONS AND RECOMMENDATIONS

LIST OF TABLES

LIST OF FIGURES

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.

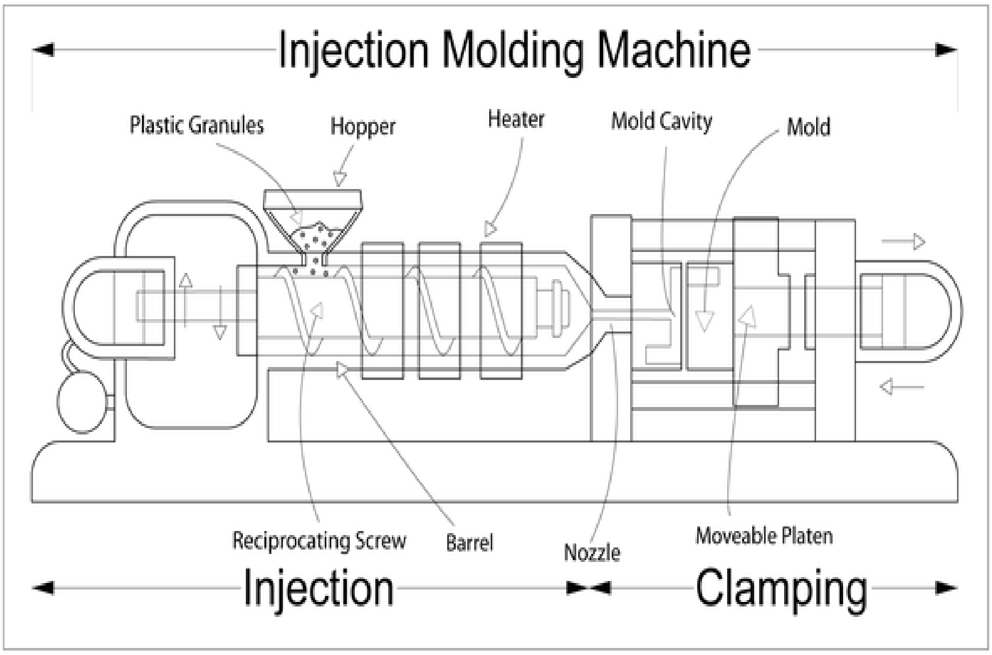


Figure 2.1 Injection Molding Machine

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 (^{o}C)	200-240	190-250
Mold Temperature (^{o}C)	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

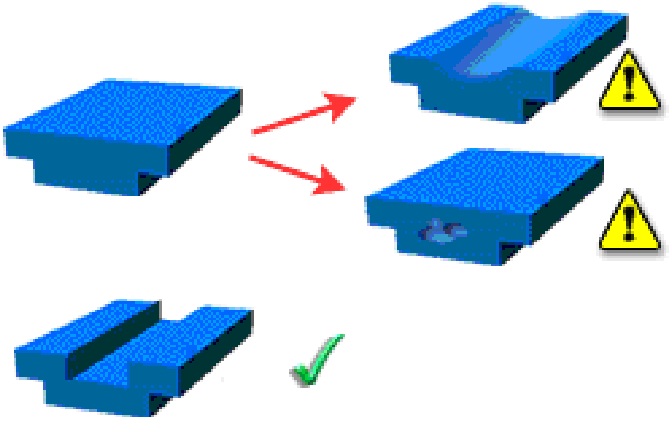


Figure 2.2: Sink Marks Defect

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

Alternatively, when the outer layer of the molded component has sufficient strength to resist the pull of volumetric shrinkage, voids will be generated at the interior, and in certain cases, this will not be manifested as an appearance-related problem. Furthermore, if latent causes exist in the shape of the product such as bosses, ribs, thick sections, uneven thickness or in the construction of the die, it will be impossible to eliminate sink mark problems unless advance countermeasures are implemented at the product design and die design stages.

On falling temperature of melt in the mould decrease in volume is more than the increase in volume on relaxation of pressure. Therefore void can not be perfectly filled in. Hence sink mark is inevitable. Sink marks happen when cooling resin loses some of its volume, and this shrinkage is not replenished. This volume reduction can also cause voids inside the plastic. Thermoplastic melt is highly compressible. It can be compressed up to 15% under pressure. Hence under such condition sink marks are unavoidable. It can be made acceptable by designing the part with out much variation in wall thickness and with out large mass of melt at any region in the part (Brydson, J, 1999). Figure 2.3 shows the variation in Wall Thickness causes sink mark.

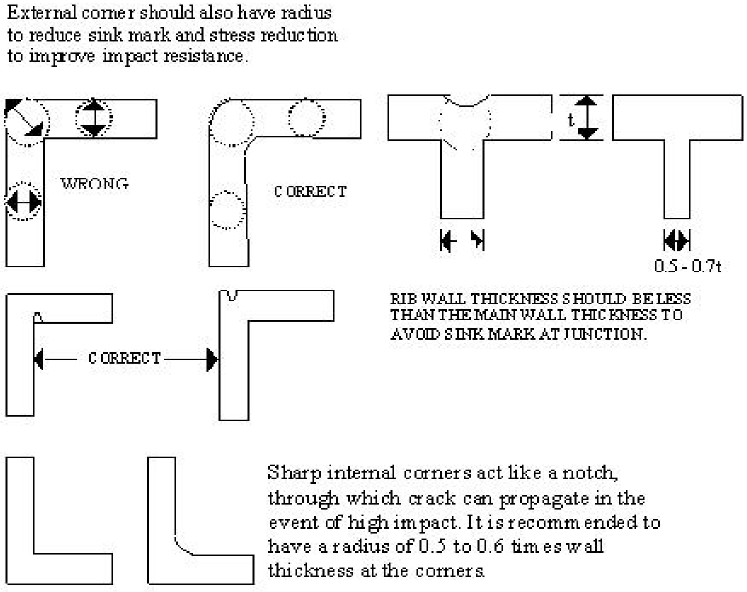


Figure 2.3: Variation in Wall Thickness causes Sink Marks

2.5.1.1 Causes of Sink Marks Defect

There are many factors cause of sink marks defect (Michael et. al, 1997). First is injection molding pressure. When Injection molding pressure is low, insufficient material is sent to cavities to compensate compressive behavior of resin so that sink marks occur by the amount of volume shrinkage. The solution is to increase injection molding pressure to sufficient level.

Besides that, mold temperature is too high while cooling rage is too low. In this case, wrinkles can occur, showing sink marks in such areas. The solution is to maintain appropriate mold temperature. Next factor is holding-pressure time. If holding-pressure time is short, it is hard to compensate volume shrinkage by solidification, causing sink marks. The solution is to increase the holding time. Non-uniform thickness of product is the fundamental cause of sink marks. Product thickness shall be designed uniformly (Brydson, J, 1999).

Next factor is rib thickness. When thickness of rib exceeds 60 percent of product thickness, shrinkage can occur. Rib thickness shall be designed within 60 percent of product thickness. Shrinkage can occur when runner and gate are too thin compared to product thickness so that insufficient pressure is applied within the mold.

Therefore, runner and gate dimensions shall be increased. Failure to make this correction and increasing injection pressure will cause the resin applied pressure be absorbed at sprue and runner. In addition, small gate dimension causes fast gate sealing and pressure drop becomes large so that insufficient pressure is sent to cavities. Then, if resin temperature is too high, sinks marks can occur at thick areas or ribs. The solution is to maintain suitable resin temperature (Brydson, J, 1999). Figure 2.4 shows creating design, rib and serrations and table 2.3 shows the cause of sink mark.

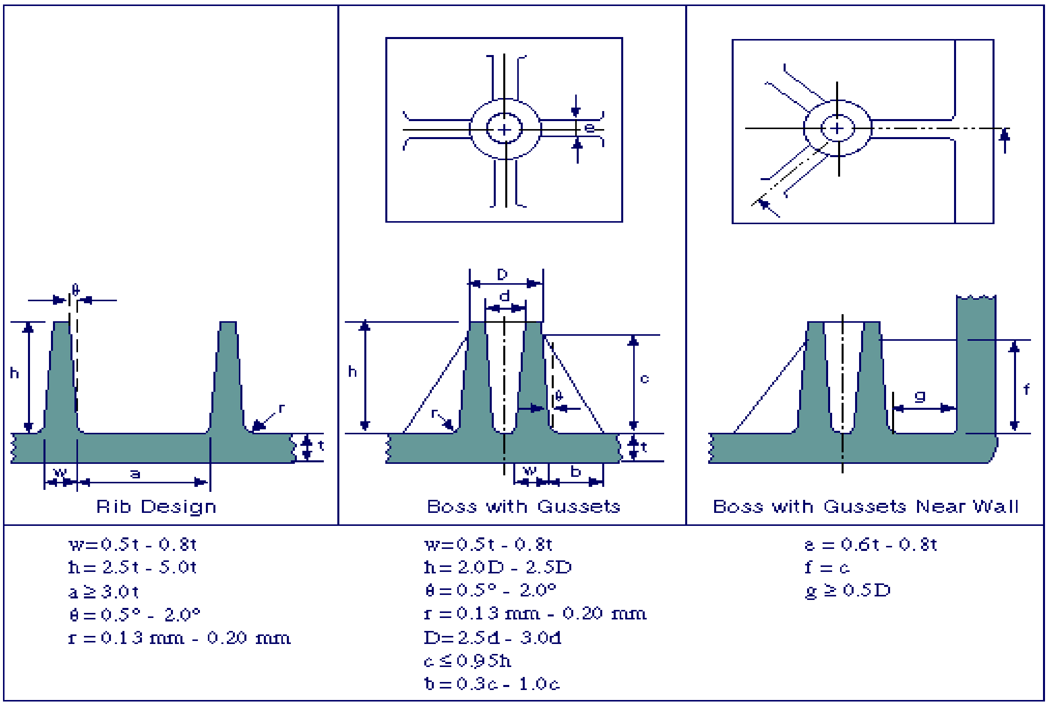


Figure 2.4: Sink Marks can be eliminated by creating a design, rib, serrations

Table 2.3: Causes of Sink Marks

Cause	**Explanation**
Injection time	If the part weight is increasing as the injection time is increase, the gate is not sealing before the pressure is reduced. Lengthen injection time.
Melt temperature	Part may be too hot when ejector and sink marks could occur post molding.
Pressure on melt	Increase pressure to pack more.
Gate design	Too small gates will prevent adequate packing. When molding parts with large areas and thin walls, it may be necessary to use large or multiple gates.

2.5.2 Weld Lines

Various defects can develop in manufacturing processes depending on the factors such as materials, part design, and processing techniques. While some defects affect only the appearance of the parts made, others can have major adverse effects on the structural integrity of the parts. One of the examples of defect is *weld line* (also known as weld mark or a knit line). A weld line is formed when separate melt fronts travelling in opposite direction meet as the mould cavity is filled (Jay Shoemaker, 2006) as shown in Figure 2.5.

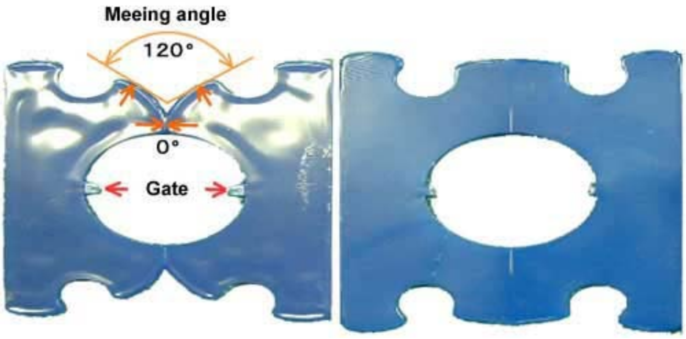


Figure 2.5: Formation of Weld Lines

Weld lines are generally undesirable when part strength and surface appearance are major concerns. It creates an unsightly line or even voids on the surface of the product (Steven Schmid, 2006). Weld lines create a weakness and change material strength characteristics where cold melt fronts reunite. The extent of the property change depends on the ability of the two melt fronts to knit together homogeneously.

The following conditions affect weld line integrity: base resin type, part thickness, mould design, resin impact modifiers, resin mould released additives, reinforcements, moulding process conditions (such as temperature and viscosity of the molten thermoplastic when they come together), and lubricants sprayed on the mould cavity surfaces. Different resins will exhibit different characteristics of tensile strength retention at the weld line (E. Alfredo Campo, 2006)

Strong weld lines are critical, because the properties in the weld line region decline significantly compared to those in the rest of the part. These lines become likely points of part failure. Weld lines can also cause irregularities in the surface appearance of the moulded part, making it more prone to wear. Table 2.4 shows the causes of weld lines

Table 2.4: Causes of Weld Lines

Cause	**Explanation**
Melt temperature	Low temperature can prevent resin from knitting together across a weld line. Temperatures that are too high can cause degrading and gassing, which can also prevent good knitting.
Mold surface temperature	If the mold temperature too low, material flowing into cavity will be cooled excessively and prevent knitting.
Pressure on melt	Low injection pressure will not force the molten material together at the weld line. Runners and gates may be too small, resulting in low injection pressure.
Choice of material	Materials that flow more easily may be needed if part performance allows.
Mold lubricants	Lubricant may be push into weld by advancing molten polymer.
Venting	Vent may be too small or improperly positioned, causing gases to be trapped at weld line.

Therefore, weld lines should be located in less critical areas if possible. Weld lines can be eliminated by altering the product design as follows:

i) Increase the wall thickness to permit easier melt flow.
ii) Use thick ribs to act as a conveying melt channel to improve and redirect the melt flow in the cavity
iii) Modify the part design to shift and/ or eliminate obstructions to flow
iv) Holes may be moulded partially to eliminate weld lines
v) Allow the use of an overflow weld line tab pocket in front of the weld line that will be removed after moulding, the weld line melt will be transferred from the cavity to the tab pocket.

2.6 COMPUTER-AIDED DESIGN MODEL

Computer-Aided Design (CAD), also known as Computer-Aided Drafting, is the use of computer software and systems to design and create 2D and 3D virtual models of goods and products for the purposes of testing. It is also sometimes referred to as computer assisted drafting. Common examples of CAD software are SOLIDWORK, AutoCAD and Mastercam. As computer-aided design has become more popular, reverse engineering has become a viable method to create a 3D virtual model of an existing physical part for use in 3D CAD, CAM, CAE and other software.. Figure 2.6 shows an example of CAD model of a card holder

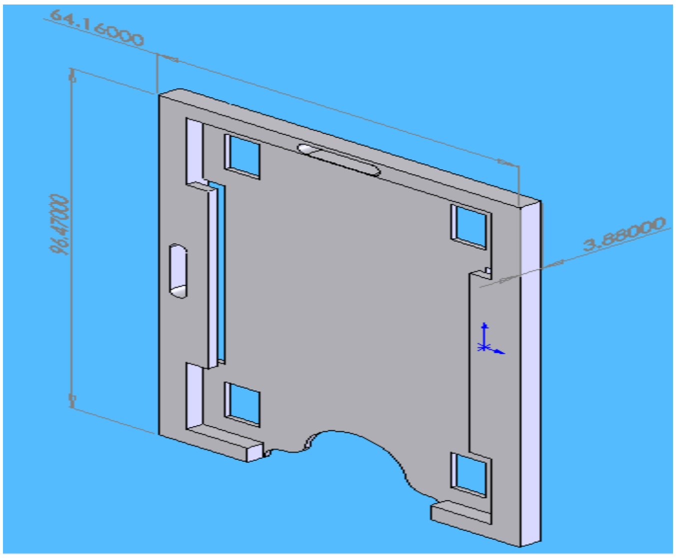


Figure 2.6: CAD Model of a Card Holder

2.7 MOLDFLOW SIMULATION SOFTWARE

Advanced manufacturing, now relies heavily on the use of simulation codes to optimize part and mould design. Flow simulation software allows a component manufacturer to predict and eliminate potential manufacturing problems as well as optimize part design, mould design, and the injection molding process (Jay Shoemaker, 2006). The use of these cost effective technologies is becoming popular as it allows a product manufacturer to remain competitive in the global market. To avoid the high costs and time delays associated with problems discovered in the manufacturing environment, it is necessary to consider the combined effects of part geometry, material selection, mold design, and processing conditions on the manufacturability of a part. Using predictive analysis tools to simulate the molding process, companies can optimize these variables in the part and mould design phases of a project, where the cost of change is minimal and the impact of change is greatest. Figure2.7 shows an injection molding simulation using MOLDFLOW simulation software.

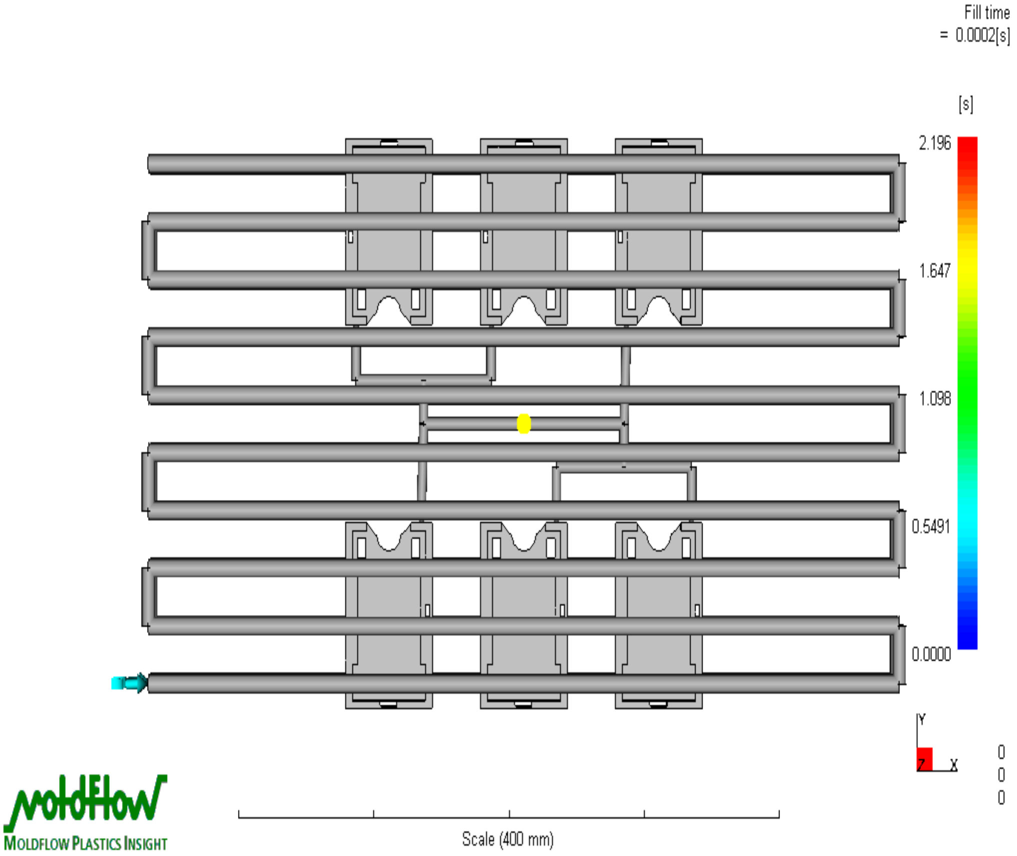


Figure 2.7: Moldflow Simulation

2.8 DESIGN OF EXPERIMENT (DOE)

Design of Experiment (DOE) is a structured, organized method that is used to determine the relationship between the different factors (Xs) affecting a process and the output of that process (Y). This method was first developed in the 1920s and 1930, by Sir Ronald A. Fisher, the renowned mathematician and geneticist (John, 1972).

Design of experiment (DOE) has been implemented to select manufacturing process parameters that could result in a better quality product. The DOE is an effective approach to optimize the various manufacturing process parameters.

DOE methods require well-structured data matrices. When applied to a well-structured matrix, analysis of variance delivers accurate results, even when the matrix that is analyzed is quite small. Today, Fisher's methods of design and analysis are international standards in business and applied science.

Experimental design is a strategy to gather empirical knowledge, i.e. knowledge based on the analysis of experimental data and not on theoretical models. It can be applied whenever you intend to investigate a phenomenon in order to gain understanding or improve performance.

Design of Experiments (DOE) is widely used in research and development, where a large proportion of the resources go towards solving optimization problems. The key to minimizing optimization costs is to conduct as few experiments as possible. DOE requires only a small set of experiments and thus helps to reduce costs

2.9 TAGUCHI METHOD

Taguchi methods have been used widely in engineering analysis to optimize the performance characteristics through the setting of design parameters. Taguchi method is also strong tool for the design of high quality systems. To optimize designs for quality, performance, and cost, Taguchi method presents a systematic approach that is simple and effective. Taguchi method was developed by Taguchi. In the Taguchi method, three stages such as system design, parameters design, and tolerance design are utilized. System design involves the application of scientific and engineering knowledge required in manufacturing a product. Parameter design is employed to find optimal process values for improving the quality characteristics. Tolerance design consists of the determining and analyzing of the tolerances in optimal settings recommended by parameter design (Taguchi G, 1990).

By applying Taguchi method based on orthogonal arrays, time and cost required to conduct the experiments can be reduced. In doing this, Taguchi method employs a special design of orthogonal arrays to learn the whole parameters space with a small number of experiments only. Taguchi recommends the use of the S/N ratio for determination of the quality characteristics implemented in engineering design problems. The S/N ratio characteristics can be divided into three stages: the smaller the better, the nominal the best, and the larger the better, signed-target type. Since the purpose of this study is to minimize sink mark within the optimal levels of process parameters, the smaller the better quality characteristic is selected (Taguchi G, 1990).

2.10 ORTHOGONAL ARRAYS

The orthogonal array (OA) has been highly utilized in engineering analysis and consists of a design of experiments with the objective of acquiring data in a controlled way, to take information about the behavior of a given process. The effects of several process parameters can be determined effectively by carrying out matrix experiments based on the Taguchi’s orthogonal design (Phadke MS, 1989).

2.11 SIGNAL TO NOISE RATIO (S/N)

The signal to noise ratio measures the sensitivity of the quality investigated to those uncontrollable factors (error) in the experiment. The higher value of *S/N* ratio is always desirable because greater *S/N* ratio will result in smaller product variance around the target value. The S/N ration characteristic can be divide into three stages: the nominal-the better, the smaller-the better, the higher-the better. In order to perform *S/N* ratio analysis, mean square deviation (*MSD*) for "the smaller-the better" quality characteristic and *S/N* ratio were calculated from the following equations (Ranjit K Roy, 1990):

$$S / N = -10 Log \, (M.S.D) \qquad (1)$$

Where M.S.D is the mean-square deviation for output characteristics

$$\text{M.S.D} = 1/\text{N} \, (\text{Y}_{\text{i}}^{2}) \qquad (2)$$

Where Y_i is the value of the sink mark index for $_i$th test. N is the total number of data point.

To calculate minimum sink index, it can be expressed by (Hasan et. al., 2007):

$$\text{S1}_{\text{op1}} + \text{S2}_{\text{op2}} + \text{S3}_{\text{op3}} + \text{S4}_{\text{op4}} - 3 \text{ x Y} \qquad (3)$$

Where S_{op} is the optimum sink index value for its level Y is total defect for sink index in the cycle.

Thus, the S/N ratio values are calculated by taking into consideration Equation (1) and (2).

2.12 ANALYSIS OF VARIANCE (ANOVA)

The purpose of the analysis of variance (ANOVA) was to investigate which parameters significantly affected the quality characteristic. In order to determine the parameters that contribute to sink index defect, the result has been analyzing using Analysis of variance (ANOVA).

ANOVA carried out to examine the influence of process parameters on the quality characteristic in this study. If some parameters do not significantly affect sink mark index they can be fixed to the recommended value of mold analyzer and excluded model generation and optimization process. This will increase the efficiency of the process. The ANOVA will compute the quantities such as degree of freedom, sum of square variance, F-ratio and percentage contribution.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

Methodology properly refers to the theoretical analysis of the methods appropriate to conduct a project or study which is of utmost importance to ensure a smooth development of the study.

In this project, sink marks defect was investigated by using injection molding and was tested using Moldflow software. Analysis on sink marks defect was carried out and figures out the factors governing the defect occurrence followed by the elimination of the defect by using the optimum parameters. Figure 3.1 shows the project methodology flow.

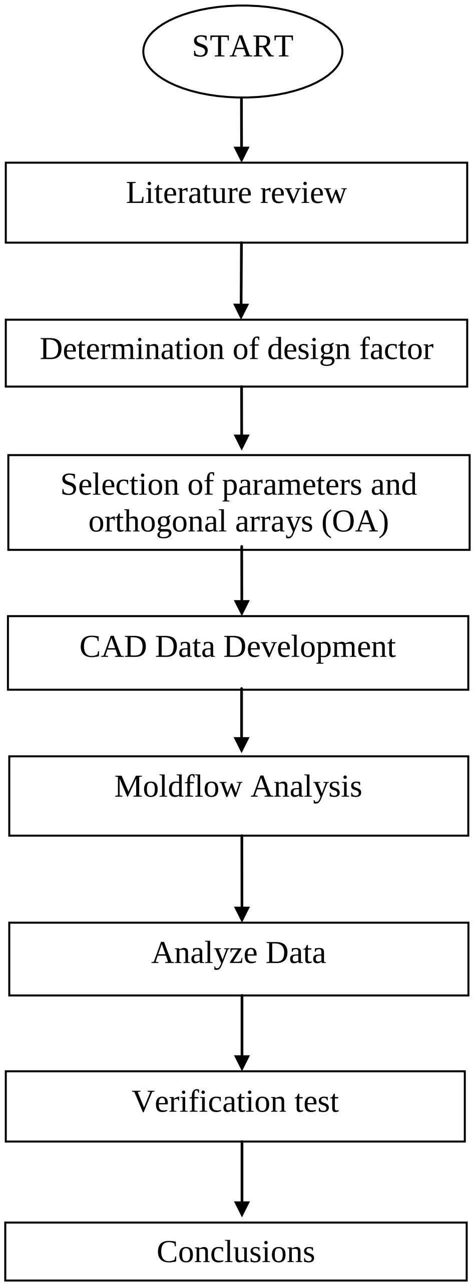


Figure 3.1: Project Methodology Flow

3.2 SELECTION OF PARAMETERS AND ORTHOGONAL ARRAYS (OAs)

The selection of the parameters involved in this experiment based on the literature studies that have been made before. Only four injection molding parameters i.e. packing pressure, mold temperature, melt temperature, and packing time were investigated in this study.

For ABS material, the range of the packing pressure was selected to be 300-450 MPa and the mold temperature was selected in the range between 40-80 ^{o}C. The melt temperature and packing time were chosen to be in the range of 200-240^{o}c and 0.6-1.0 sec respectively. The above ranges of the process parameters were selected in light of the data available in the literature (Brown, R.P., 1988).

For Polyethylene (PE) material, the range of the packing pressure was selected to be 60-90 MPa and the mold temperature was selected in the range between 30-70 ^{o}C. The melt temperature and packing time were chosen to be in the range of 190-250^{o}c and 0.5-1.5 sec respectively. The selected injection molding process parameters along with their levels are shown in Table3.1. Each parameter had three levels low, medium and high.

Table 3.1: The parameters for 3 levels of selected factors

Material	Level	Packing Pressure (MPa) A	Mold temperature (^{0}c) B	Melt temperature (^{0}c) C	Packing time (s) D
ABS	1	300	40	200	0.6
	2	375	60	220	0.8
	3	450	80	240	1.0
PE	1	60	30	190	0.5
	2	75	50	220	1.0
	3	90	70	250	1.5

The selection of an appropriate orthogonal array (OAs) depends on the total degrees of freedom of process parameters. Degrees of freedom are defined as the number of comparisons between process parameters that need to be made to determine which level is better and specifically how much better it is. In this study, since each parameter has three levels therefore, the total degrees of freedom (DOF) for the parameters are equal to 17. Basically, the degrees of freedom for the OA should be greater than or at least equal to those for the process parameters. The standard L_{18} orthogonal array has four 3 level columns with 17 DOF. Therefore, an L_{18} orthogonal array with four columns and 18 rows was appropriate and used in this study. The experimental layout for the injection molding parameters using the L_{18} OA is shown in Table 3.2. Each row of this table represents an experiment with different combination of parameters and their levels.

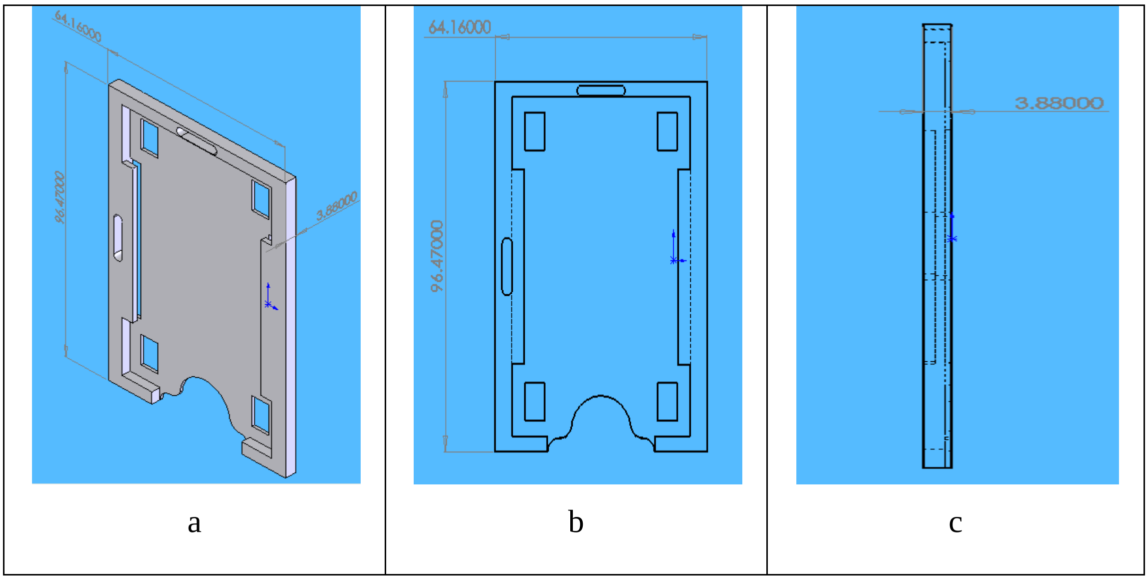
Table 3.2: Experimental plan using L_{18} Orthogonal Array

	Parameters/Level			
Test no.	**A**	**B**	**C**	**D**
1	1	1	1	1
2	1	2	2	2
3	1	3	3	3
4	2	1	1	2
5	2	2	2	3
6	2	3	3	1
7	3	1	2	1
8	3	2	3	2
9	3	3	1	3
10	1	1	3	3
11	1	2	1	1
12	1	3	2	2
13	2	1	2	3
14	2	2	3	1
15	2	3	1	2
16	3	1	3	2
17	3	2	1	3
18	3	3	2	1

3.3 CAD DATA DEVELOPMENT

CAD model of the matric card holder was created using SOLIDWORK 2006 with dimensions identical the original physical model. Generally, the part design has significant effects on the injection moulding process. Therefore, the part design requirements include uniform wall thickness, parting line location to balance the heat removal from both sides of the cavities, smooth internal corners, draft walls (to facilitate part removal from the cavity), elimination of feather edges, elimination of fragile deep pockets (long thin cores), provide location for the gate, allow large permissible surface area for ejection, specify typical part dimension tolerances for plastics, and avoid the use of high-gloss surface finishing for the product. The design was save as IGES file to import in Moldflow Plastics Insight software.

Figure 3.2 shows CAD model of the matric card holder will be created using SOLIDWORK 2006 with dimensions identical the original physical model.



a b c

Figure 3.2: (a) Matric Card Holder 3D view, (b) Front View (c) Side View

3.4 MOLDFLOW ANALYSIS USING MPI 5.0

The simulation on moldflow of the designed injection mold using Moldflow Plastics Insight 5.0 is to see whether the part produced is free from defects. This software used the finite element and finite difference methods to calculate a series of mathematical functions representing the mold process.

This simulation provide information such as distribution and variation of temperature, pressure, flow rate, skin property, shear stress and shear rate of the material in filling, packing and cooling stages. Figure 3.3 shows Moldflow analysis using MPI 5.0.

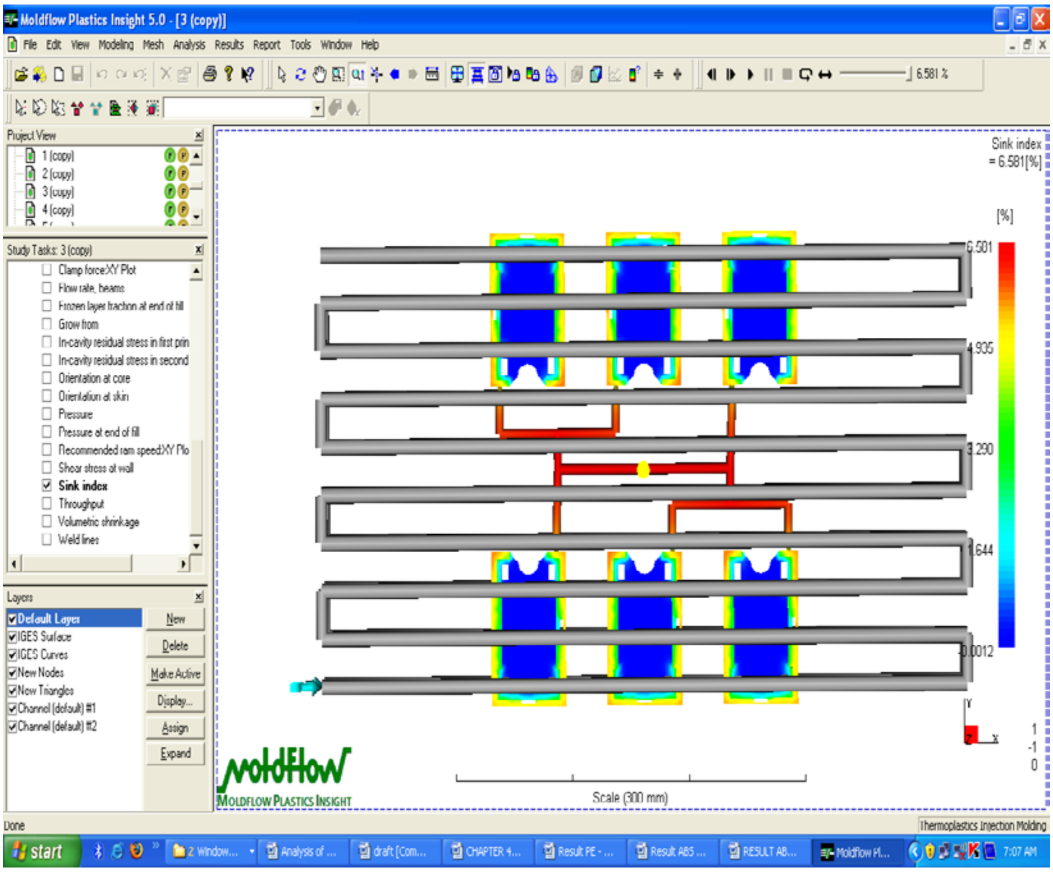


Figure 3.3: Moldflow Analysis using MPI 5.0

3.5 ANALYZE DATA

After the simulation, the result obtained from moldflow analysis software. The signal to noise ratio (S/N ratio) was used to measures the sensitivity of the quality investigated to those uncontrollable factors (error) in the experiment. The signal to noise ratio (S/N ratio) was calculated by using Minitab software. The quality characteristic used in this study was “the smaller-the better”, in order to reduce sink mark index through optimum parameters in injection molding process.

Then Analysis of variance (ANOVA) is then used to determine which process parameter is statistically significant and the contribution of each process parameter towards the output characteristic. With the main effect and ANOVA analyses, possible combination of optimum parameters can be predicted. Finally, a confirmation experiment is conducted to verify the optimal process parameters obtained from the process parameter design.

.

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 INTRODUCTION

The results, in terms of sink index were obtained after conducting a series of simulations using Moldflow Plastic Insight 5.0. Each test represented one experiment in the orthogonal arrays.

Then, the results were analyzed by employing main ANOVA, and the signal-to-noise ratio (S/N) approach. Finally, a verification test was carried out to compare the simulation values with the calculated values and indicated the effectiveness of the Taguchi method.

4.2 MOLDFLOW ANALYSIS RESULT FOR ABS AND POLYETHYLENE (PE)

Part was designed by using Solidworks 2006 then imported to MPI 5.0 to analyze the sink index and most effective parameters to prevent sink index. Figure 4.1 and figure 4.2 shows moldflow analysis result for both materials ABS and PE for test run no.1 and other results can be refer to appendix B1 and B2.

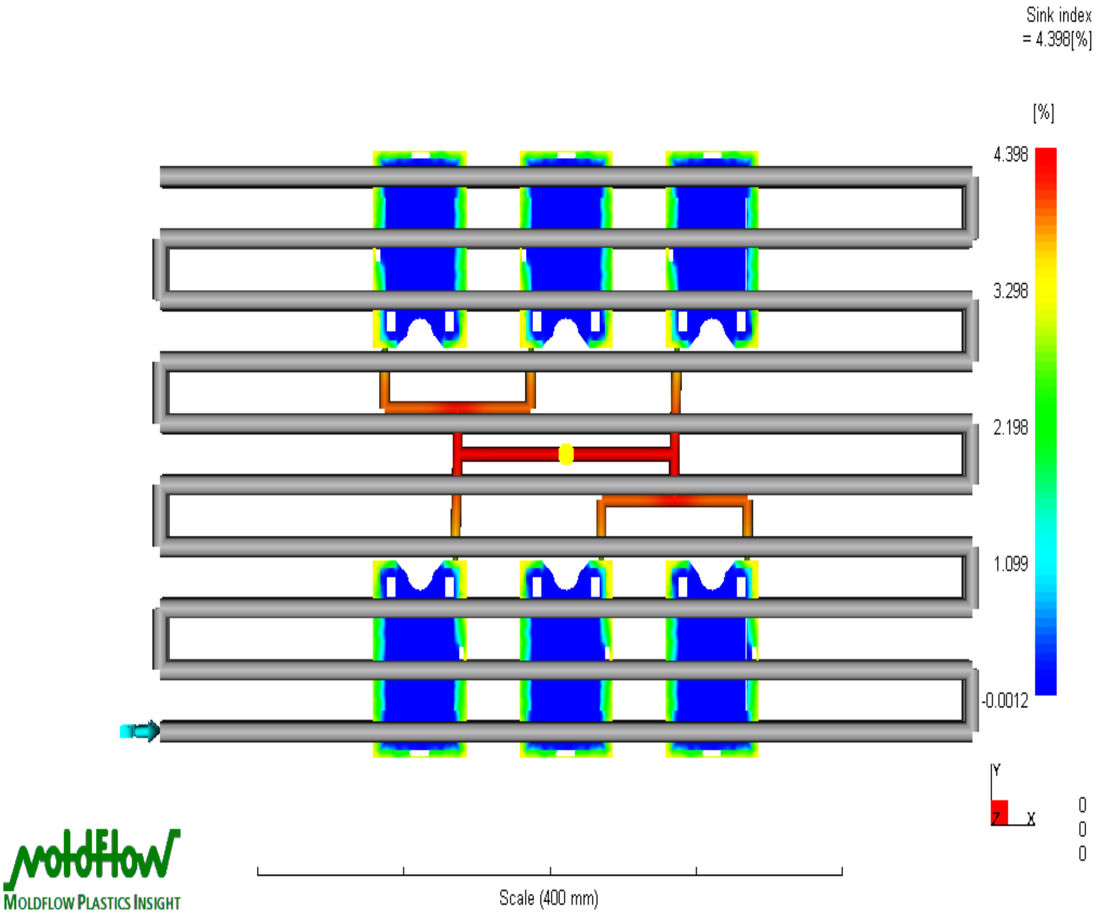


Figure 4.1: Moldflow Analysis for ABS (test no.1)

Below here shows the results summary of sink index after packing phase for the part (test no.1):

Sink index – maximum = 3.5489 %
Sink index - 95th percentile = 3.3519 %
Sink index – minimum = 1.7485 %
Sink index - RMS deviation = 1.4276 %
Average sink index = 2.6487 %

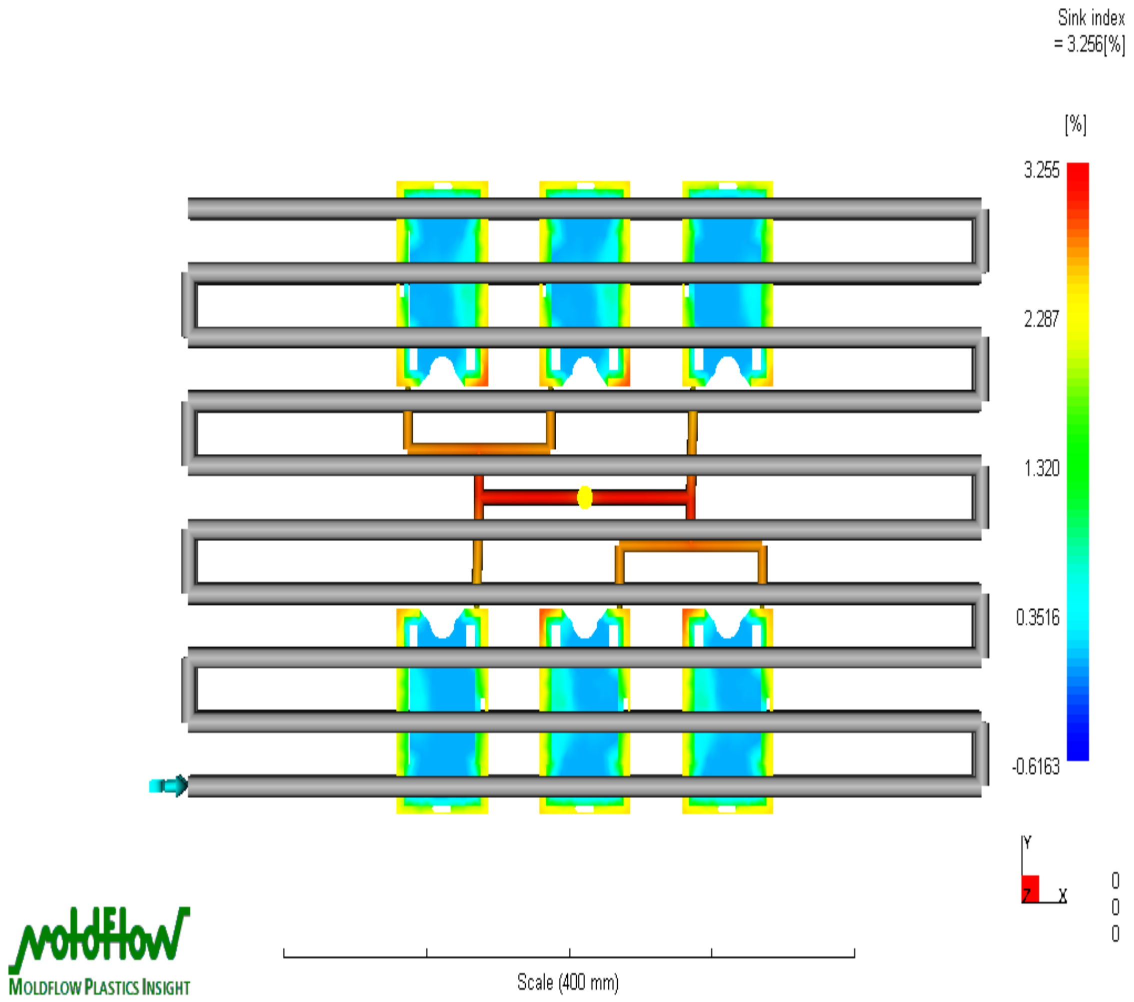


Figure 4.2: Moldflow Analysis for Polyethylene (test no.1)

Below here shows the results summary of sink index after packing phase for the part (test no.1):

Sink index – maximum = 2.8937 %
Sink index - 95th percentile = 2.6084 %
Sink index – minimum = 1.5183 %
Sink index - RMS deviation = 1.0428 %
Average sink index = 2.2060 %

4.3 SIGNAL TO NOISE RATIO (S/N)

From moldflow analysis result, the signal to noise ratio (S/N) obtained by using Minitab software. The quality characteristic used in this study was “the smaller-the better”, in order to reduce sink mark index through optimum parameters in injection molding process. Table 4.1 and 4.2 shows the result for S/N values ratio for ABS and PE material.

Table 4.1: The Sink Index and S/N ratio for ABS

Test no	Packing Pressure A (MPa)	Mold Temp. B (^{0}C)	Melt Temp. C (^{0}C)	Packing Time D (s)	Sink Index (%)	S/N(dBi)
1	300	40	200	0.6	2.6487	-8.4607
2	300	60	220	0.8	3.5473	-10.9980
3	300	80	240	1.0	4.6292	-13.3101
4	375	40	200	0.8	2.5181	-8.0215
5	375	60	220	1.0	3.3974	-10.6229
6	375	80	240	0.6	5.0714	-14.1026
7	450	40	220	0.6	3.6022	-11.1314
8	450	60	240	0.8	4.5774	-13.2124
9	450	80	200	1.0	2.7044	-8.6414
10	300	40	240	1.0	4.2248	-12.5161
11	300	60	200	0.6	2.7322	-8.7302
12	300	80	220	0.8	3.7686	-11.5236
13	375	40	220	1.0	3.2298	-10.1835
14	375	60	240	0.6	4.7785	-13.5858
15	375	80	200	0.8	2.8265	-9.0250
16	450	40	240	0.8	4.2662	-12.6008
17	450	60	200	1.0	2.4892	-7.9212
18	450	80	220	0.6	3.9970	-12.0347

Table 4.2: The Sink Index and S/N ratio for Polyethylene

Test no	Packing Pressure A (MPa)	Mold Temp. B (^{0}C)	Melt Temp. C (^{0}C)	Packing Time D (s)	Sink Index (%)	S/N(dBi)
1	60	40	200	0.6	2.2060	- 6.8721
2	60	60	220	0.8	3.1565	-9.98412
3	60	80	240	1.0	4.0640	-12.1791
4	75	40	200	0.8	1.6982	-4.59978
5	75	60	220	1.0	2.6076	-8.3248
6	75	80	240	0.6	2.3090	-7.2684
7	90	40	220	0.6	3.7318	-11.4384
8	90	60	240	0.8	4.4539	-12.9748
9	90	80	200	1.0	1.6654	-4.43037
10	60	40	240	1.0	3.8488	-11.7065
11	60	60	200	0.6	2.1183	-6.5198
12	60	80	220	0.8	3.2847	-10.3299
13	75	40	220	1.0	2.5999	-8.2991
14	75	60	240	0.6	4.2077	-12.4809
15	75	80	200	0.8	2.0262	-6.1337
16	90	40	240	0.8	3.9355	-11.9000
17	90	60	200	1.0	1.6368	-4.2799
18	90	80	220	0.6	3.3077	-10.3905

4.4 S/N RESPONSE DIAGRAM FOR ABS AND POLYETHYLENE (PE)

The sink index response diagram for each parameter at level 1(lowest), level2 (medium) and level 3 were created by utilizing the S/N ratio values. The value obtained shown in figure 4.3 for ABS and figure 4.4 for polyethylene (PE):

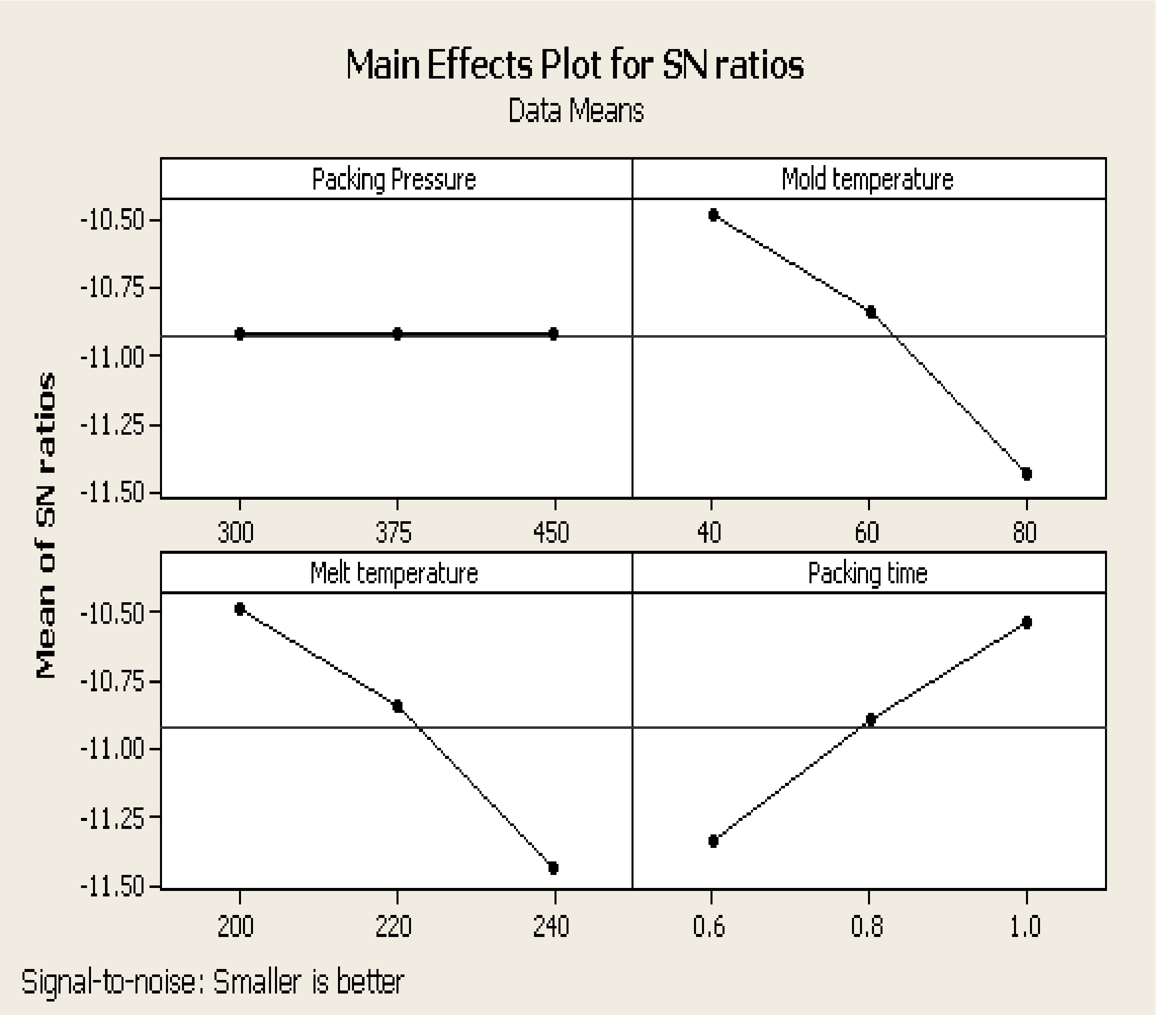


Figure 4.3: S/N response diagram for ABS

From the graph as shown in figure 4.3, the best set of combination parameters can be determined by selecting the level with the highest value of each factor Thus the results obtained for ABS are:

4.4.1 Best parameters for ABS

- Packing pressure A, level 2 (375 MPa)
- Mold temperature B, level 1 (40^0c)
- Melt temperature C, level 1 (200^0c)
- Packing time D, level 3 (1s)

From simulation the average sink mark index for best parameter is 2.205%

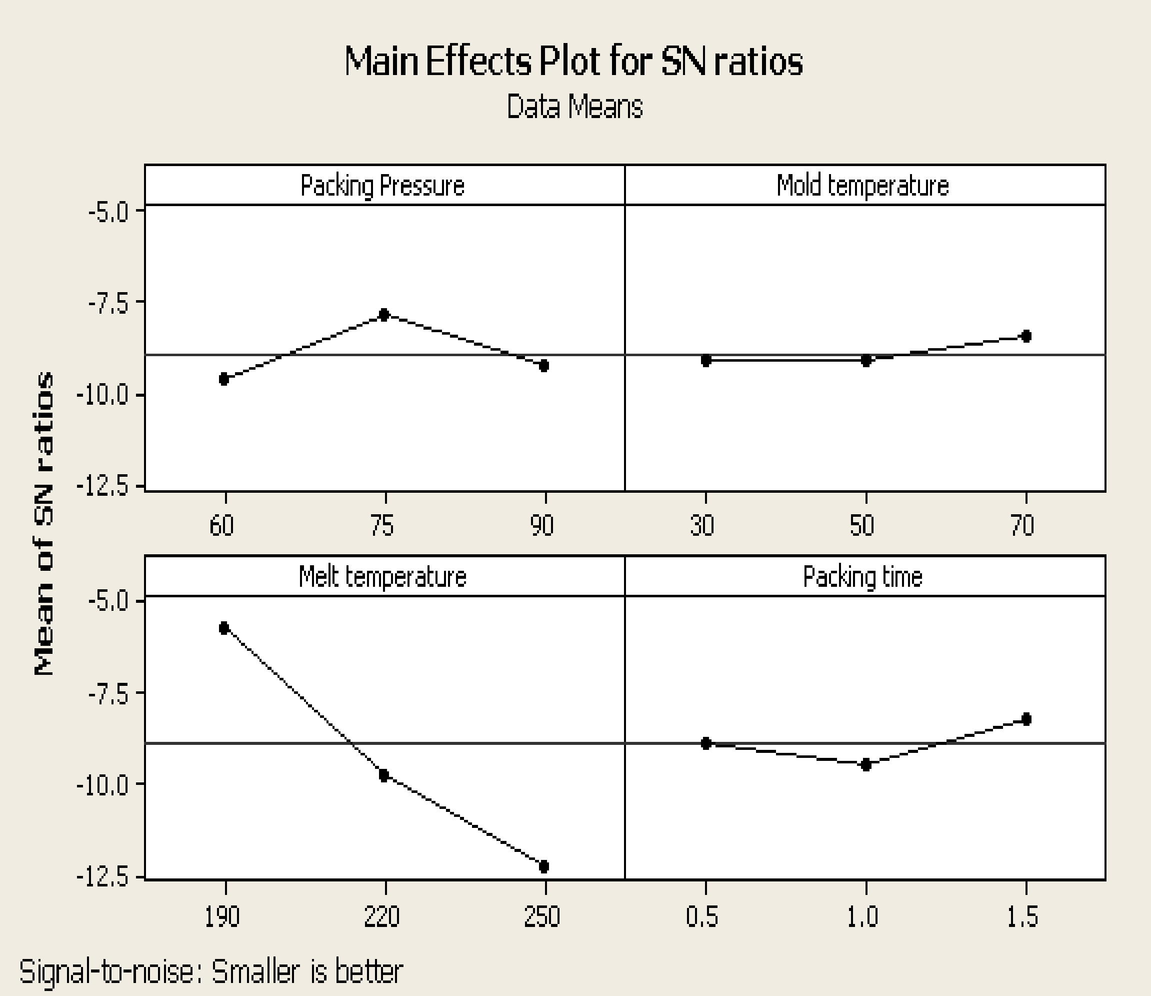


Figure 4.4: S/N response diagram for Polyethylene (PE)

For polyethylene (PE), the best set of combination parameters can be determined by selecting the level with the highest value of each factor from the graph as shows in figure 4.10. Thus the results obtained for PE are:

4.4.2 Best parameters for polyethylene

- Packing pressure A, level 2 (75 MPa)
- Mold temperature B, level 3 (70^0c)
- Melt temperature C, level 1 (190^0c)
- Packing time D, level 3 (1.5s)

From simulation the average sink mark index for best parameter is 1.3691 %

4.5 ANALYSIS OF VARIANCE (ANOVA)

The purpose of the analysis of variance (ANOVA) was to investigate which parameters significantly affected the quality characteristic. The analysis of variance was obtained by using Minitab software.

Table 4.3 shows the results of ANOVA for sink index in ABS material. The F-ratios were obtained for 90% level of confidence. In addition to this, percent contribution of each parameter was also calculated. It can also be seen from this table that the contribution of parameter i.e. the melt temperature contributes the most to the quality characteristic which is 92.17%. The contribution of other parameters in descending order mold temperature contributed 4.33%, packing time 3.22% and lastly packing pressure just contributed 0.05% and it can said this 3 parameters is not significant factor for sink mark index for ABS material.

Thus, based on the main effect and ANOVA analyses, the optimal combination of parameters and their levels for achieving minimum sink mark is $A_2B_1C_1D_3$ i.e. packing pressure at level 2 (375 MPa), mold temperature at level 1 (40^0c), melt temperature at level 1 (200^0c.), and packing time at level 3 (1s). The error values are very small which is contributed 0.23% where not influenced the sink mark index values for ABS material. The results for ABS can be summarizing as shown in table 4.3:

Table 4.3 Analysis of Variance for ABS

Source	DF	$S(10^{-3})$	$V(10^{-3})$	F	P(%)
Packing Pressure, A	2	0.0064	0.0032	1.03	0.05
Mold temperature, B	2	0.5293	0.2647	85.20	4.33
Melt temperature, C	2	11.2724	5.6362	1814.35	92.17
Packing time, D	2	0.3939	0.1970	63.40	3.22
Error	9	0.0280	0.0031		0.23
Total	17	12.2300			100

Table 4.4 shows the results of ANOVA for sink index in polyethylene (PE) material. The F-ratios were obtained for 90% level of confidence. In addition to this, percent contribution of each parameter was also calculated.

It can also be seen from this table that the contribution of parameter i.e. the melt temperature contributes the most to the quality characteristic which is 81.30%.The contribution of other parameters in descending order mold temperature contributed 7.97%, packing time 7.63% and lastly packing pressure just contributed 1.52% and it can said this 3 parameters is not significant factor for sink mark index for PE material.

Thus, based on the main effect and ANOVA analyses, the optimal combination of parameters and their levels for achieving minimum sink mark is $A_2B_3C_1D_3$ i.e. packing pressure at level 2 (75 MPa), mold temperature at level 3 (70^0c), melt temperature at level 1 (190^0c.), and packing time at level 3 (1.5s). The error values are small which is contributed 1.58% where not influenced the sink mark index values for polyethylene (PE) material. The results for polyethylene (PE) can be summarizing as shown in table 4.4:

Table 4.4 Analysis of Variance for PE

Source	DF	$S(10^{-3})$	$V(10^{-3})$	F	P(%)
Packing Pressure, A	2	1.1783	0.0781	2.87	7.63
Mold temperature, B	2	0.2337	0.0713	2.62	1.52
Melt temperature, C	2	12.5575	6.6519	244.55	81.30
Packing time, D	2	1.2309	0.6155	22.63	7.97
Error	9	0.2448	0.0272		1.58
Total	17	15.4452			100

4.6 VERIFICATION TEST

The minimum sink index was estimated based on equation 3 by using the optimum parameters,. The calculations for minimum sink index are shown as follow:

4.6.1 Calculation for minimum sink mark index for ABS

= $S1_{op1} + S2_{op2} + S3_{op3} + S4_{op4} – 3 \text{ x } Y$

= 3.2911 + 3.1551 + 3.3916 + 3.1691 – 3 x (3.6116)

= 2.1721%

Meanwhile for the minimum sink mark index from simulation is 2.2050%. Thus the error between simulation and calculation value is 1.49%

4.6.2 Calculation for minimum Sink mark Index for polyethylene (PE)

= $S1_{op1} + S2_{op2} + S3_{op3} + S4_{op4} – 3 \text{ x } Y$

= 2.3710 + 2.4131 + 2.8598 + 2.4522 – 3 x (2.9366)

= 1.2863%

Meanwhile for the minimum sink mark index from simulation is 1.3691%. Thus the error between simulation and calculation value is 6.05%

CHAPTER 5

CONCLUSIONS AND RECOMMENDATIONS

5.1 CONCLUSIONS

On the basis of the results obtained from the present case study the following can be concluded:

a) In this study, the sink mark index for ABS and Polyethylene (PE) material show the different values when the parameter settings were changed. The use of Taguchi's OAs simplified the experiment runs and ANOVA show the influenced factor that contributed to the sink index defect.

b) The optimum parameters for ABS material are packing pressure at level 2(375 Mpa), Mold Temperature at level 1 (40^0C), melt temperature at level 1 (200^0C) and packing time at level 3 (1s).

c) For polyethylene, the optimum parameters for PE material are packing pressure at level 2(75 Mpa), Mold Temperature at level 1 (70^0C), melt temperature at level 1 (190^0C) and packing time at level 3 (1.5s)

d) From the ANOVA, both materials are have the higher contributions of melt temperature, 92.17% for ABS and 81.30%.for PE that significantly affected the quality characteristic.

e) From verification test polyethylene is better material to reduce sink mark compared to ABS because from simulation with the optimum parameter setting and calculated value it give small percent of sink index which is 1.3691% and 1.2863% while ABS are 2.2050% from simulation and 2.1721% from calculated value.

5.2 RECOMMENDATIONS

a) For the next studies are using other materials in order to get different result and to compare which is the best material to minimize sink marks defect.
b) Besides that, use other parameters setting such as injection time, injection pressure to investigate the defect.

REFERENCES

Douglas M. Bryce. *Plastic Injection Molding Manufacturing Process Fundamentals*. SME, 1996

Tony Whelan. *Polymer Technology Dictionary* Springer, 1994

Tim A. Osswald. *International Plastics Handbook.* Hanser Verlag., 2006

Donald V. Rosato, Marlene G. Rosato. *Concise Encyclopedia of Plastics*. Springer, 2000

Micheal Dominick , Rosato Marlene, and Rosato Donald *Injection Molding Handbook 3rd Ed.* Kluwer Academic Publishers, 1997

Robert H. Todd, Dell K. Allen and Leo Alting *Manufacturing Processes Reference Guide*. Industrial Press Inc., 1994

Brydson, J, *Plastics Materials*, Butterworths 9th Ed (1999)

Brown, R.P., 1988, *"Handbook of Plastic Test Methods"*. 3rd Edition, Longman Scientific & Technical, New York.

MPI (2006) MPI user guide version 5, Moldflow. Notes on sink mark index

Belofsky, H., 1995, *"Plastics: Product Design and Process Engineering"*. Hanser Publishers, New York

Rubin, I.I., 1972, *"Injection Moulding Theory and Practice"*. John Wiley & Sons, New York.

Shi L, Gupta M (1998) A localized shrinkage analysis for predicting sink marks in injection-molded plastics parts. J Inj Moulding Technol 2(4):149–155

Glanvill, A .B., and Denton, E.N., 1988, *"Injection-Mould Design Fundamentals"*. Industrial Press Inc., New York.

Hasan Oktem, Tuncay Ezurumulu, and Ibrahim Uzman, 2007. *Application of taguchi optimization technique in determining plastic injection molding process parameters for thin-shell part.* Material and Design 28,2007 1271- 1278

APPENDICES

APPENDIX B1

MOLDFLOW ANALYSIS RESULTS FOR ABS MATERIAL

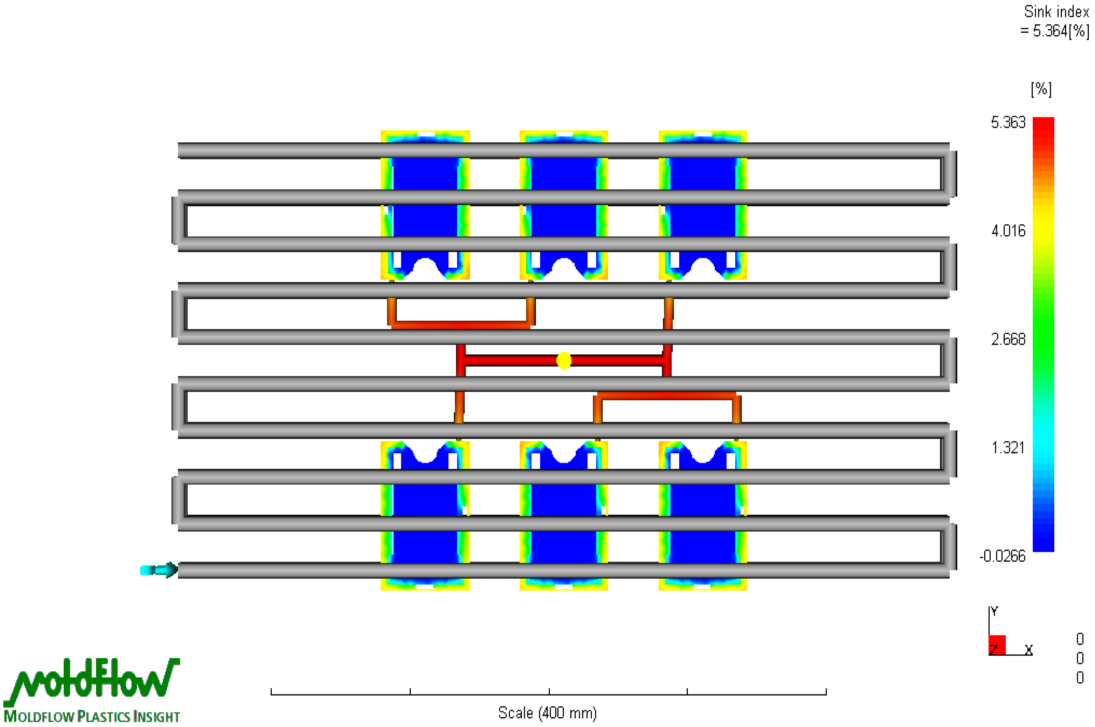


Figure B1 (1): Test no. 2

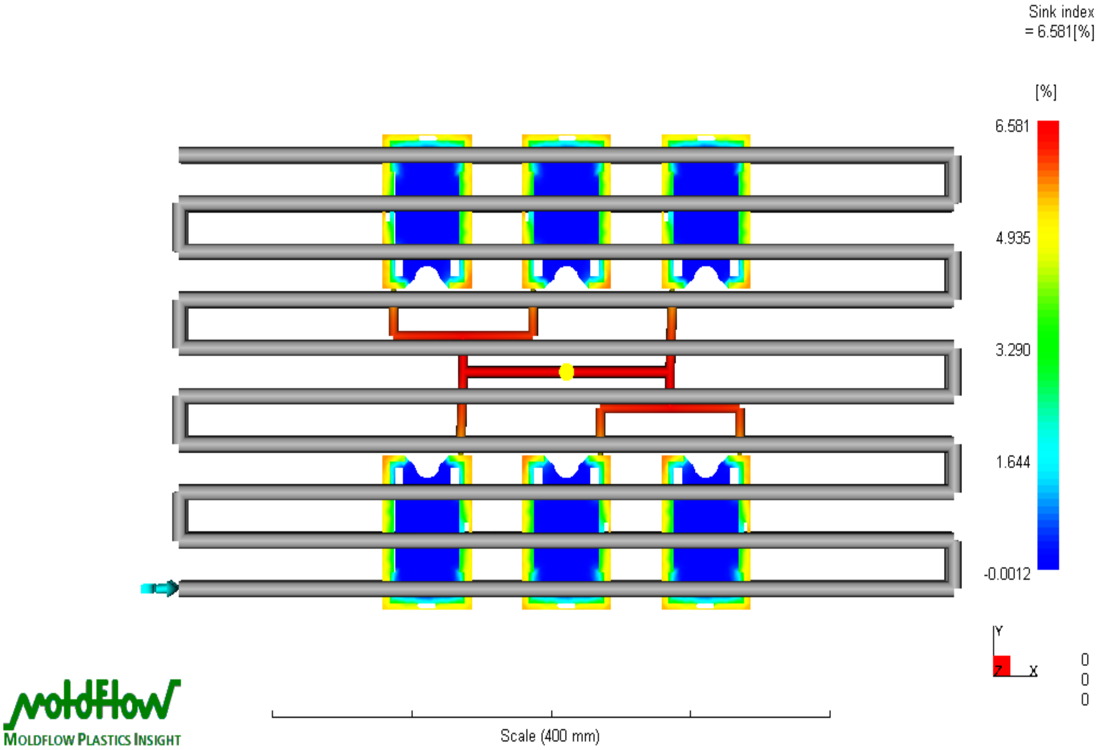


Figure B1 (2): Test no. 3

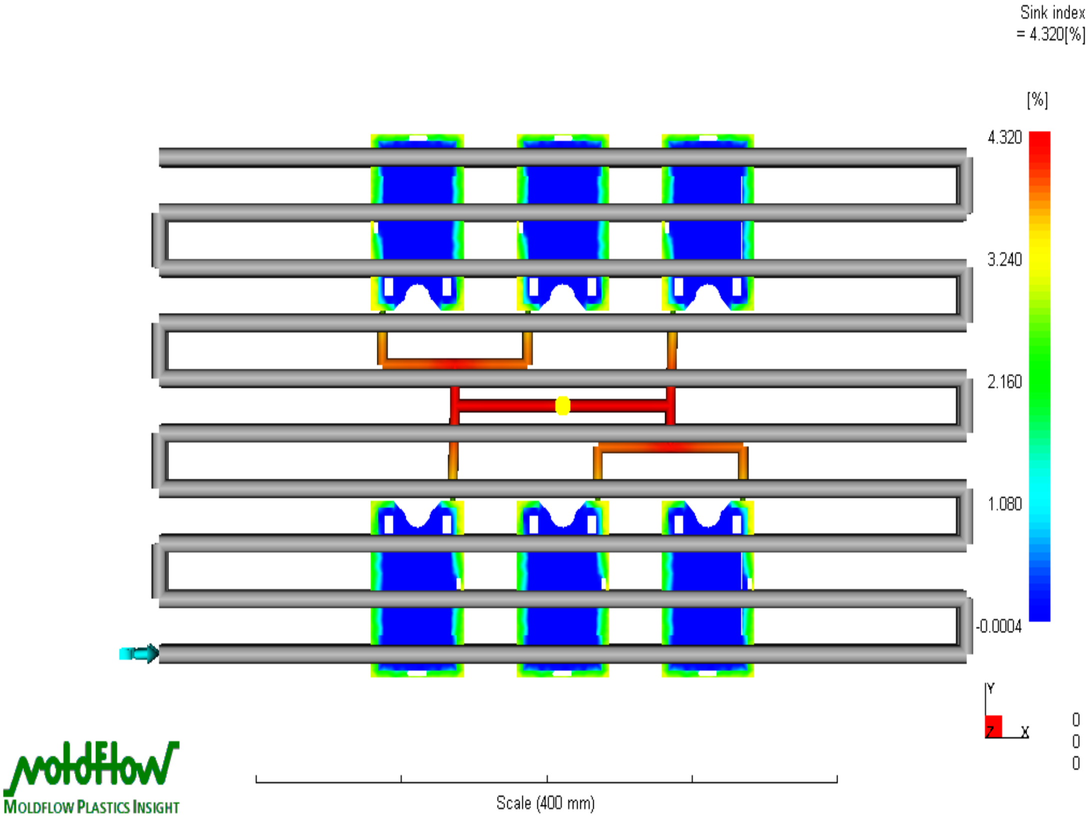


Figure B1 (3): Test no. 4

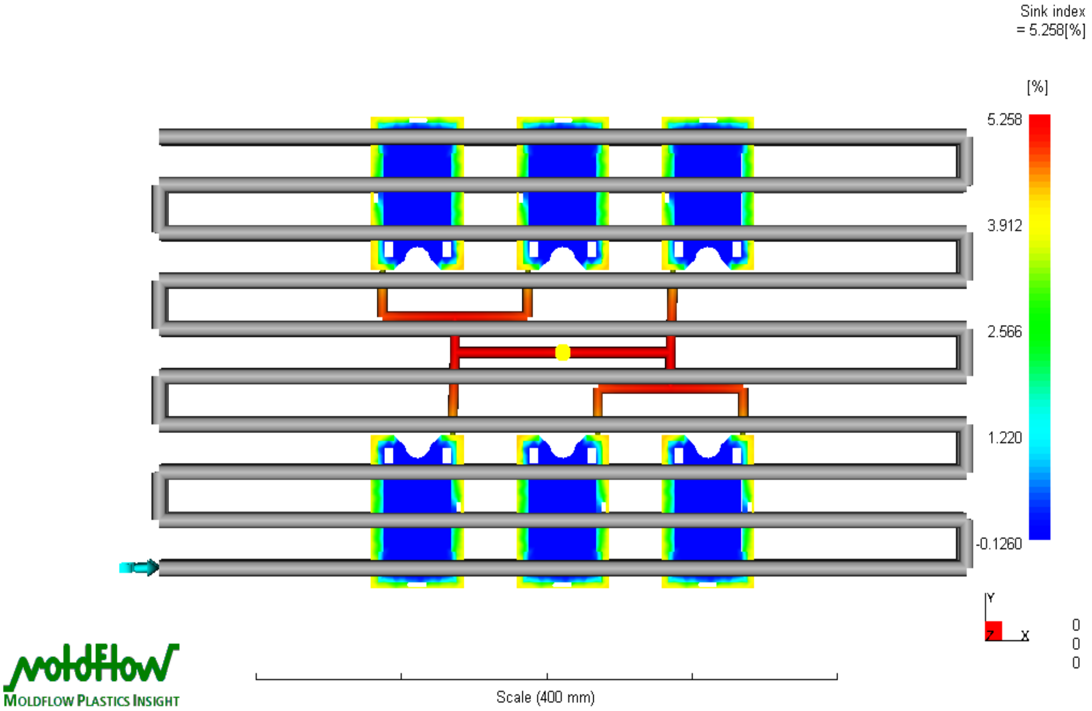


Figure B1 (4): Test no. 5

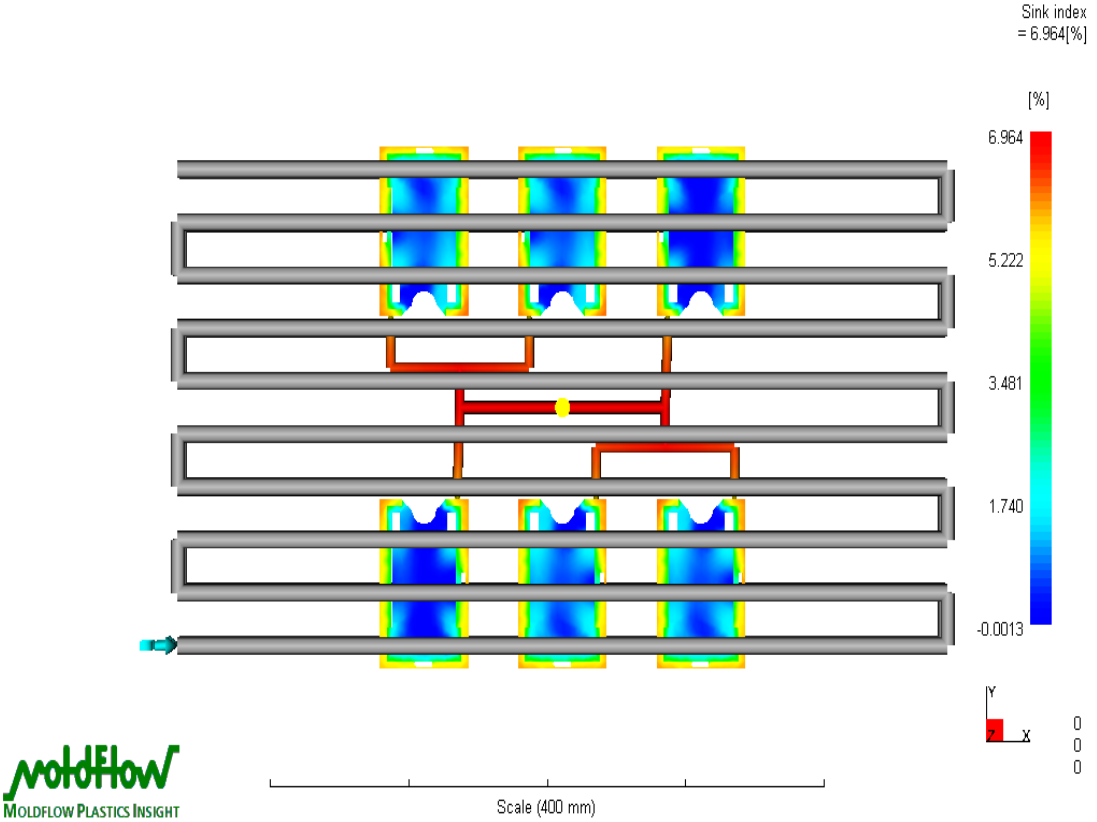


Figure B1 (5): Test no. 6

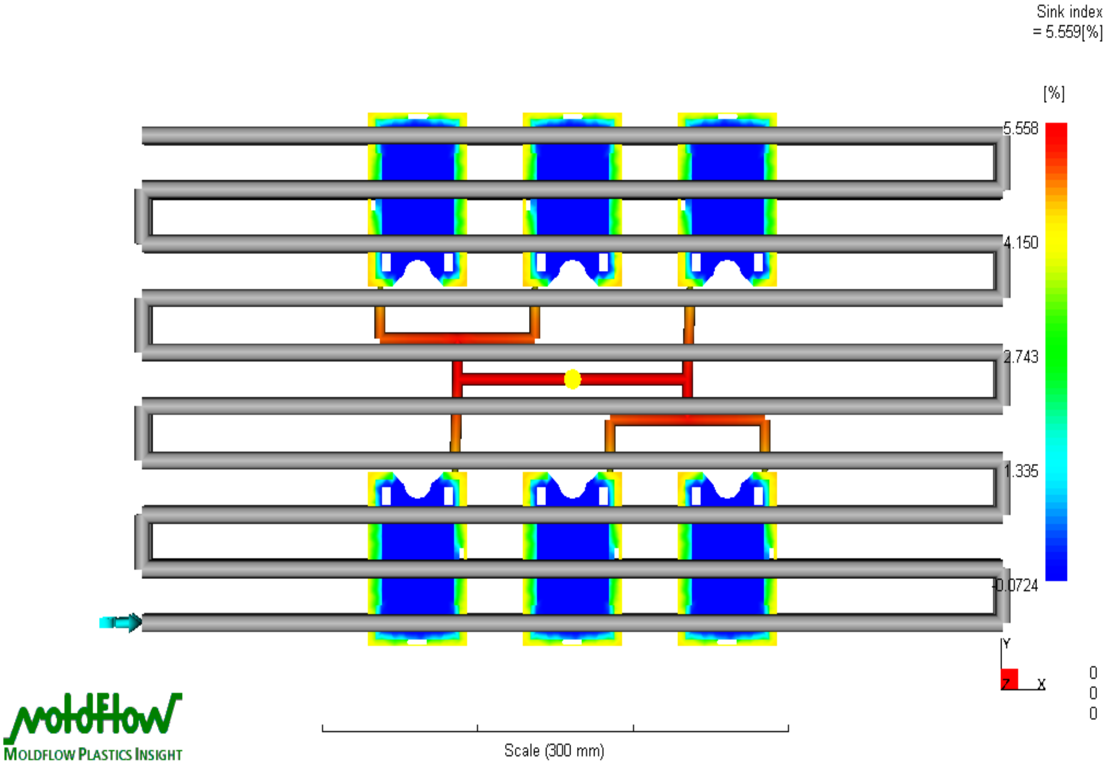


Figure B1 (6): Test no. 7

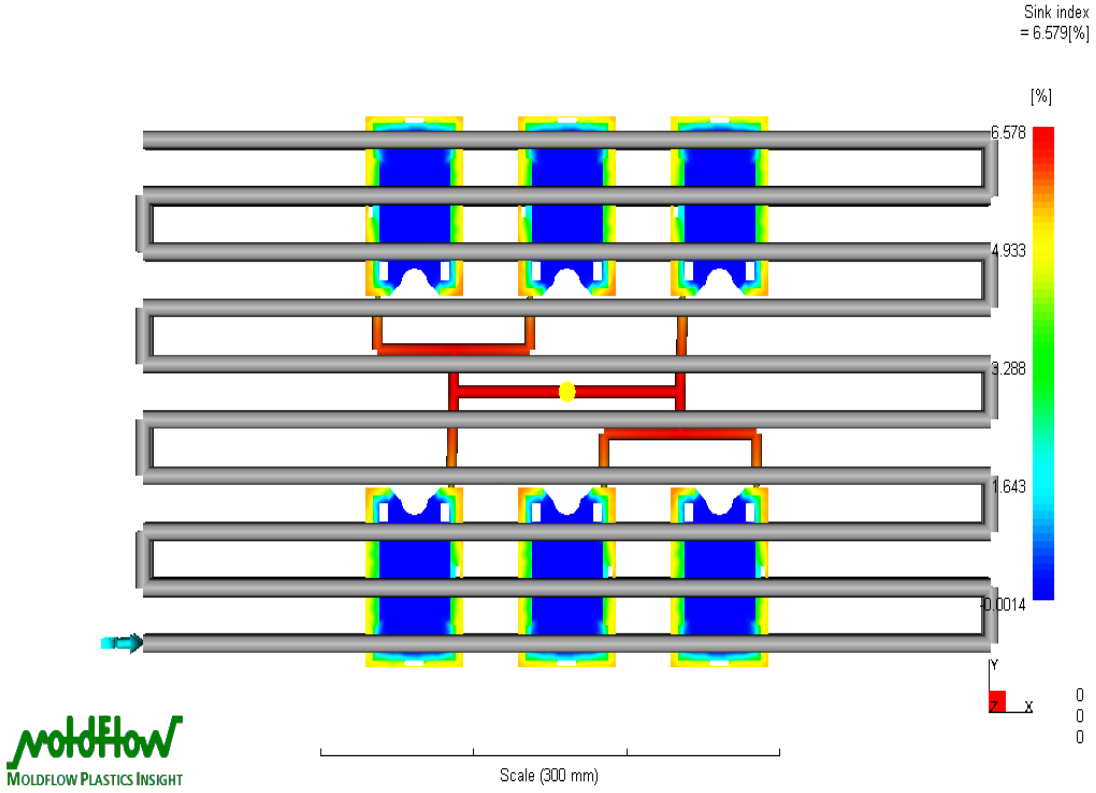


Figure B1 (7): Test no. 8

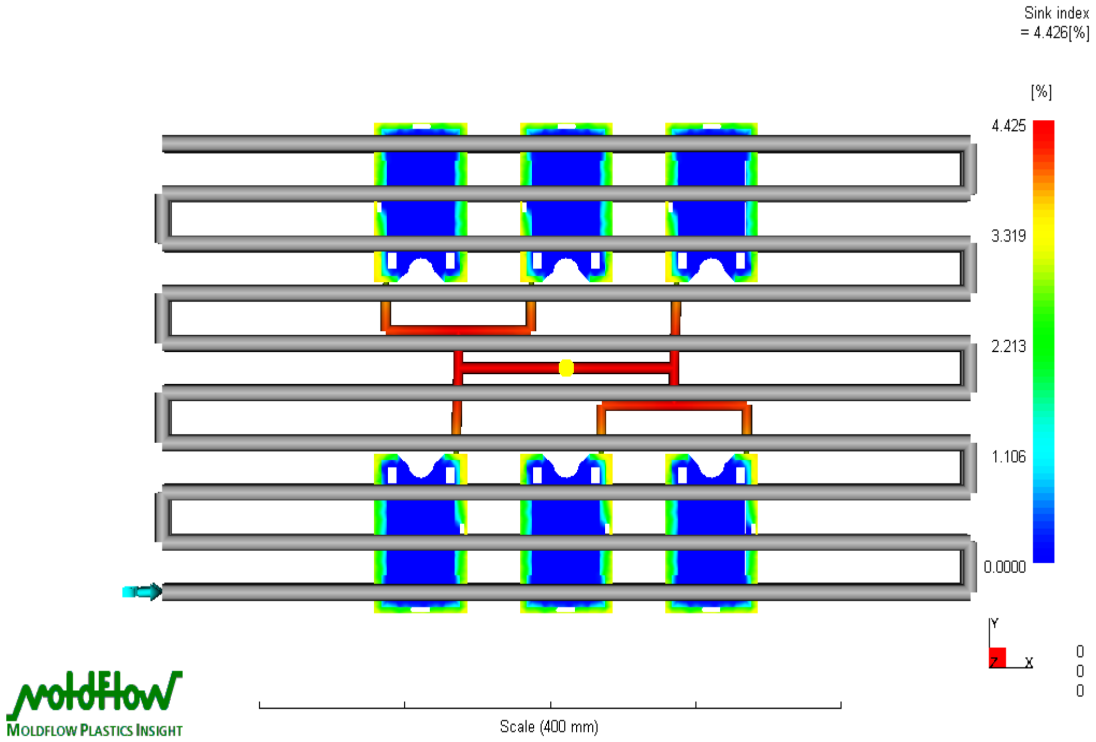


Figure B1 (8): Test no. 9

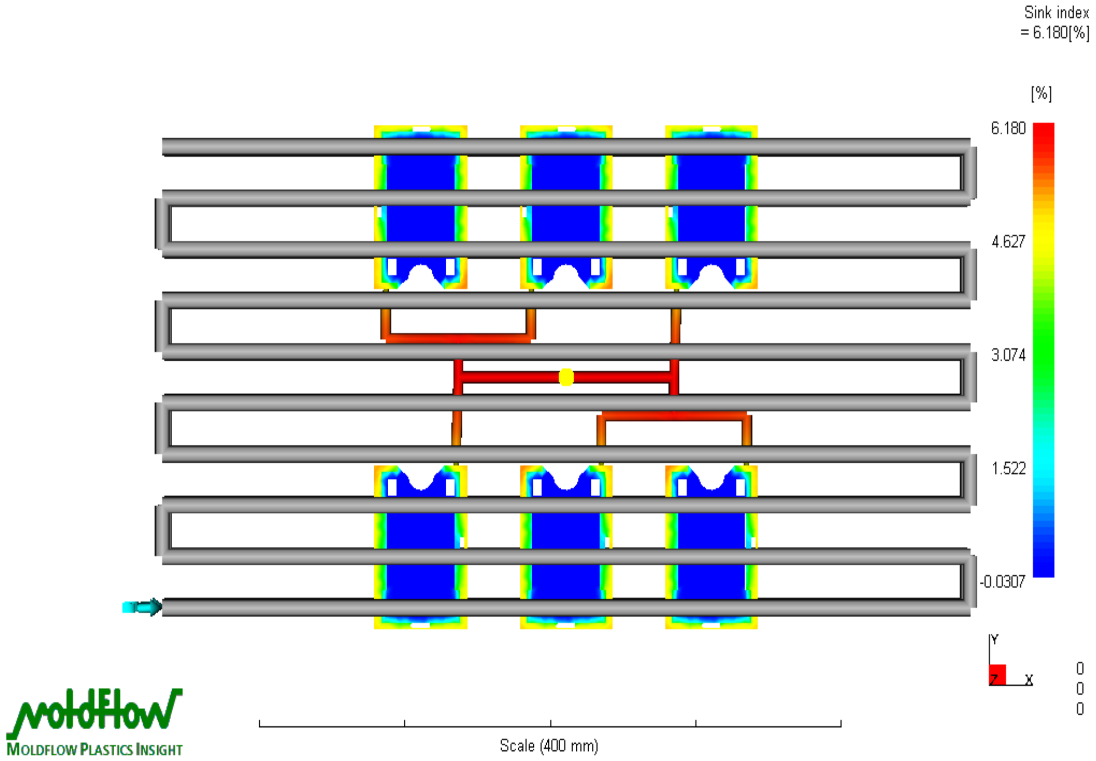


Figure B1 (9): Test no. 10

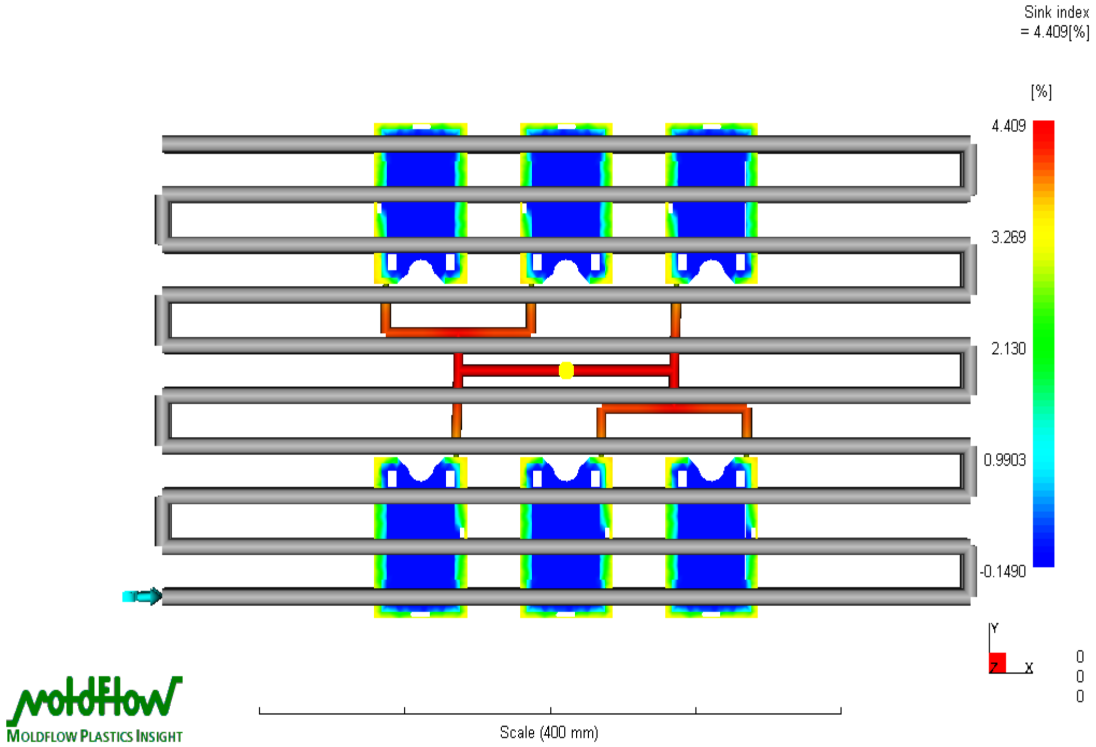


Figure B1 (10): Test no. 11

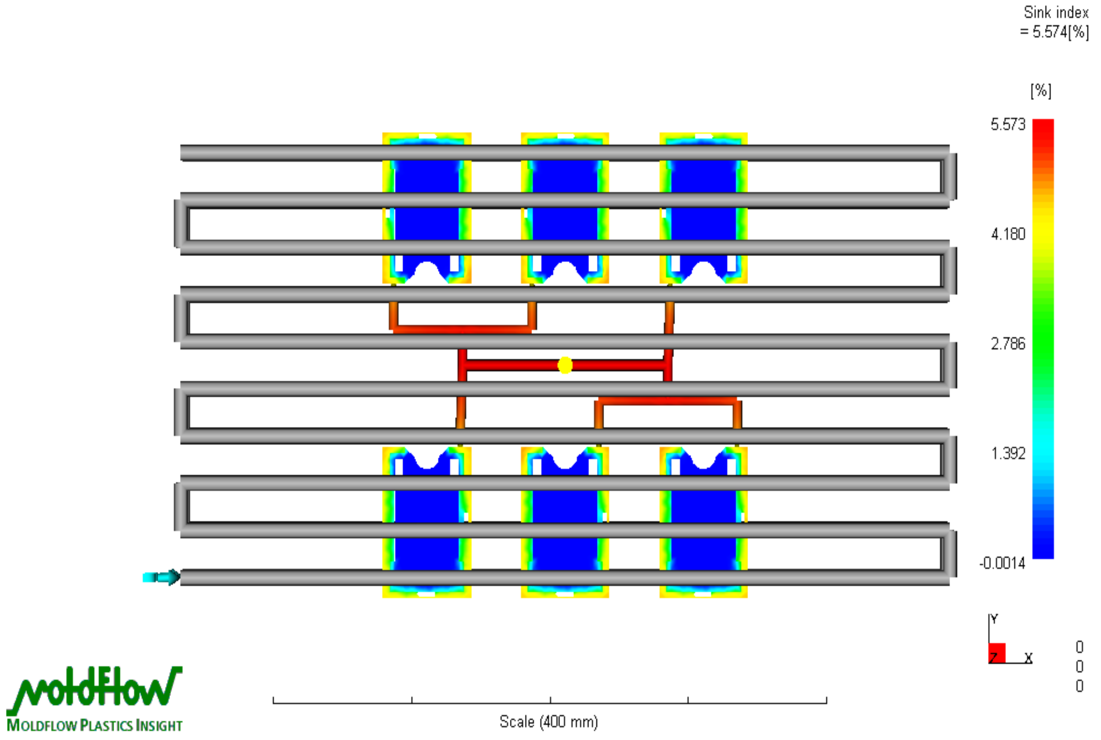


Figure B1 (11): Test no. 12

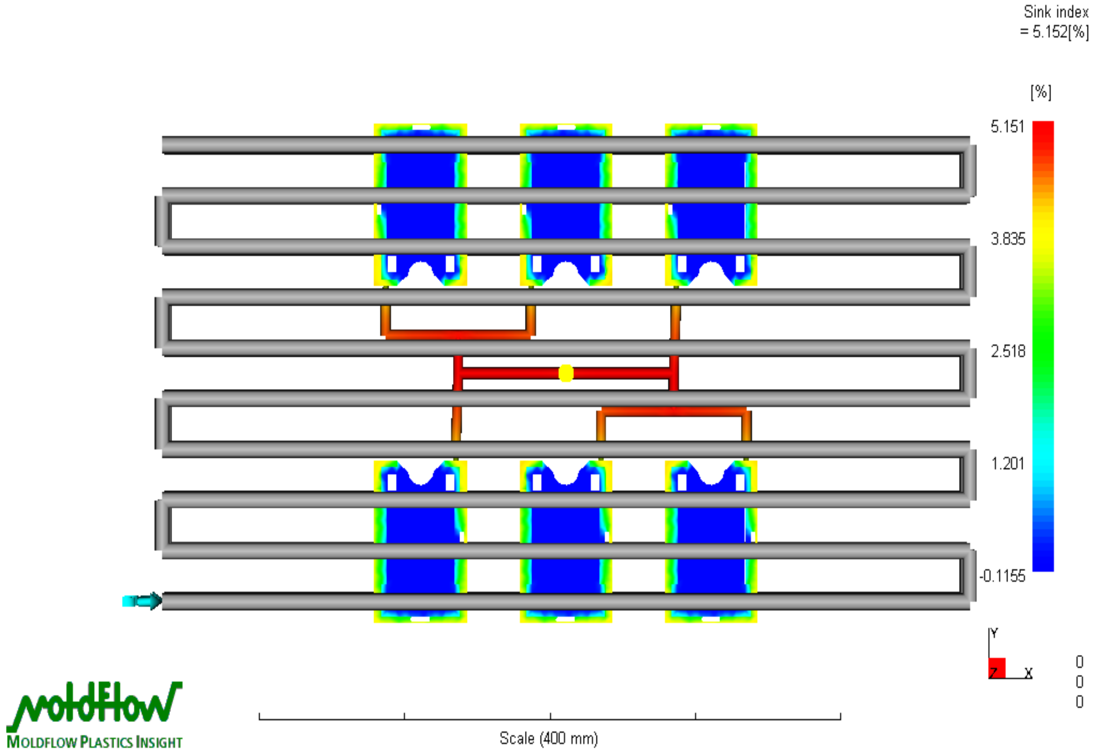


Figure B1 (12): Test no. 13

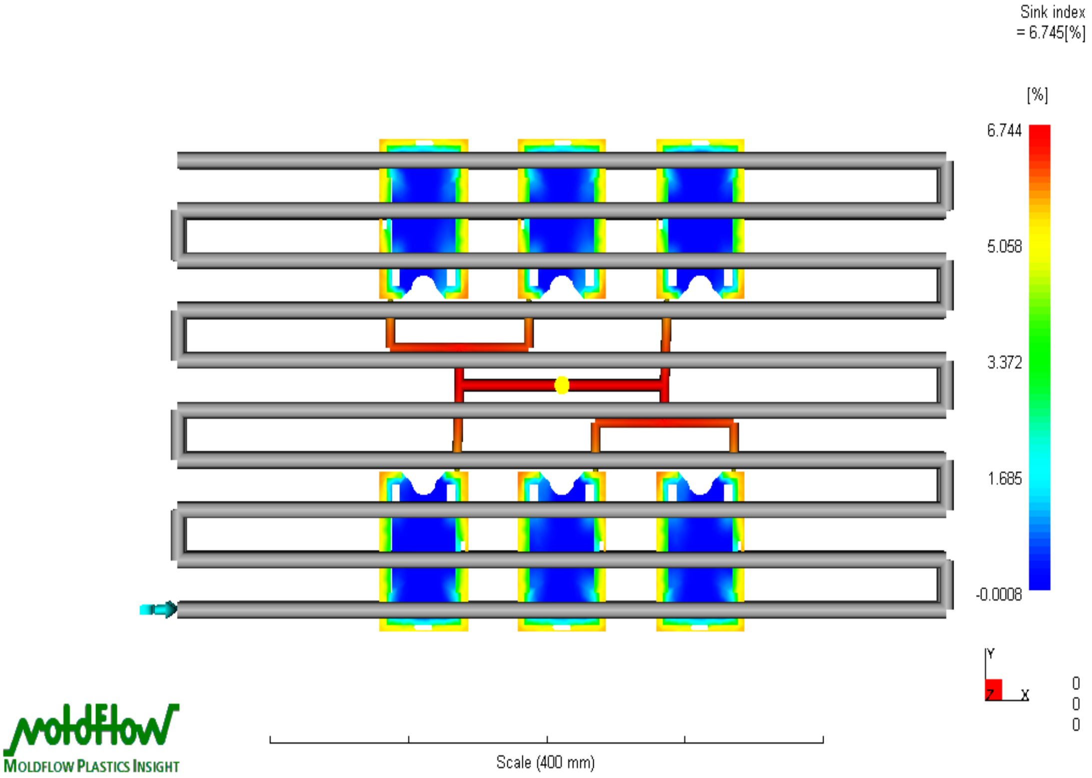


Figure B1 (13): Test no. 14

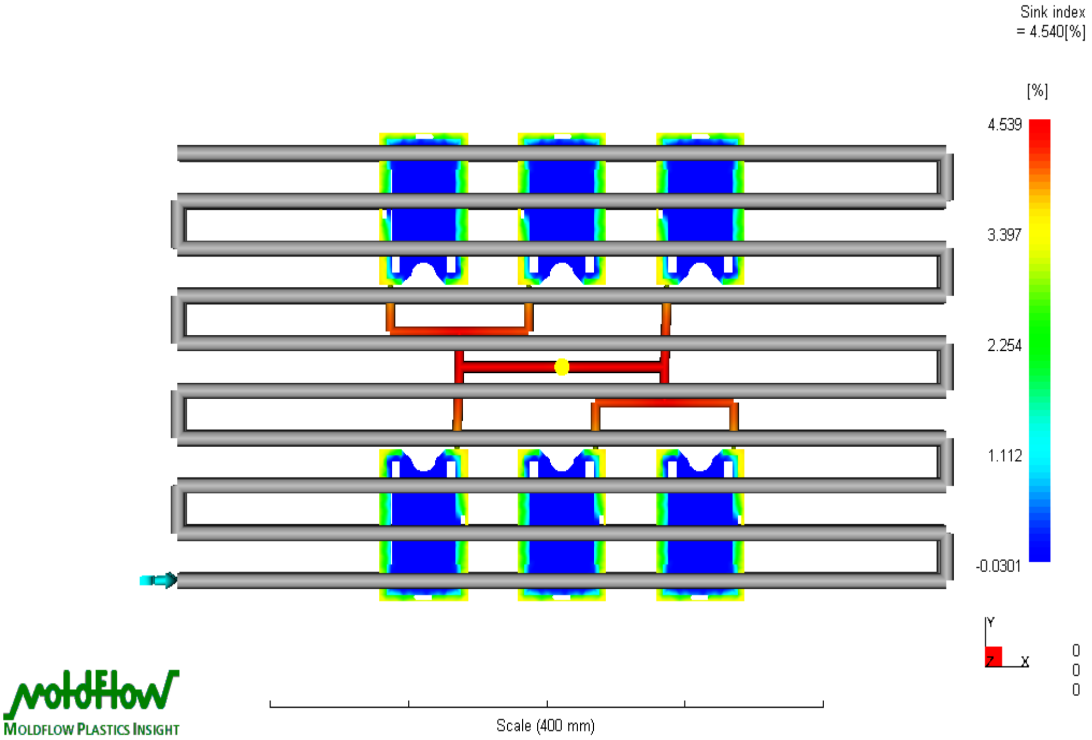


Figure B1 (14): Test no. 15

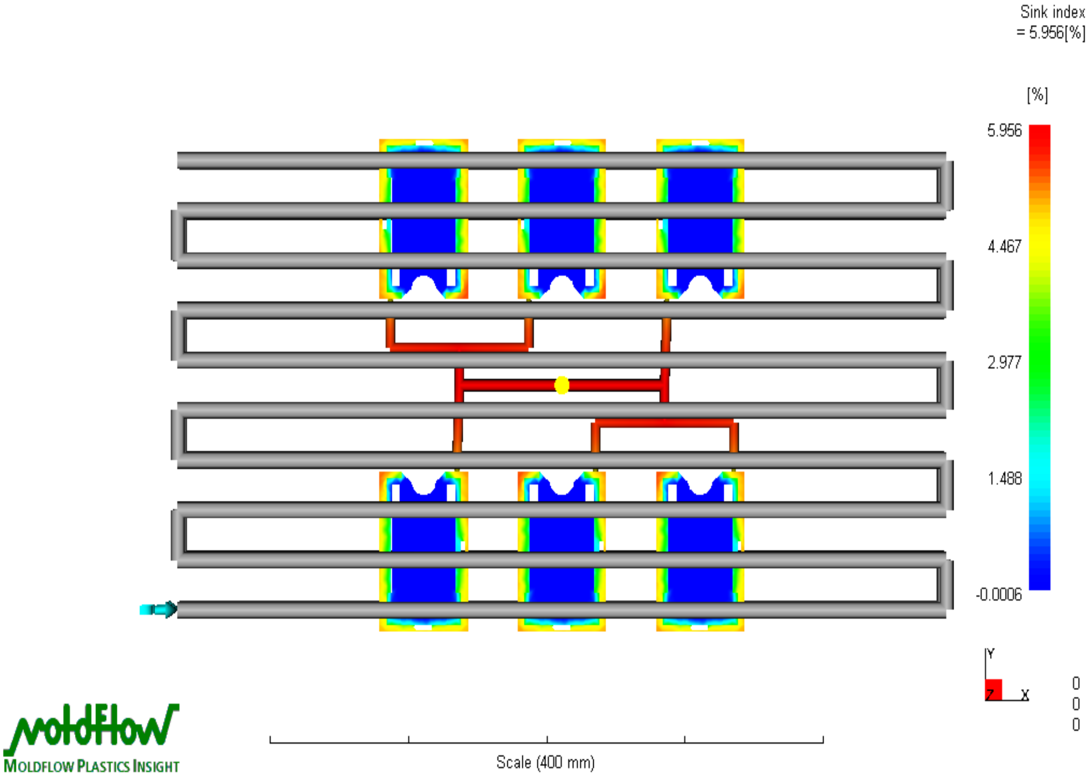


Figure B1 (15): Test no. 16

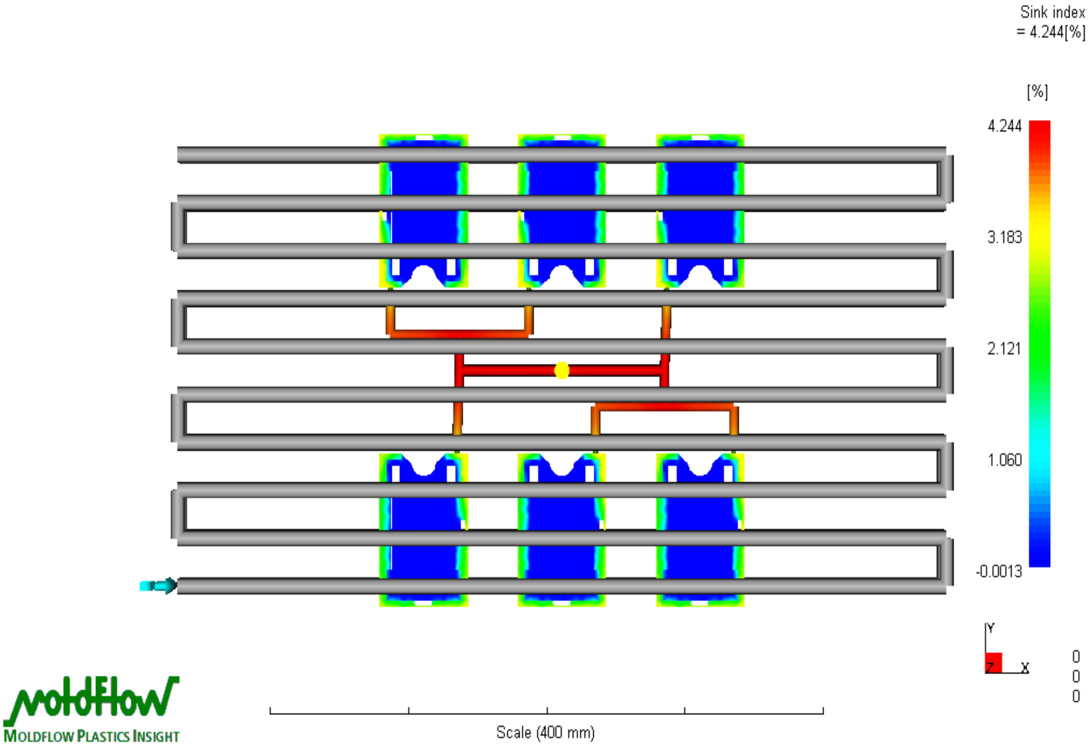


Figure B1 (16): Test no. 17

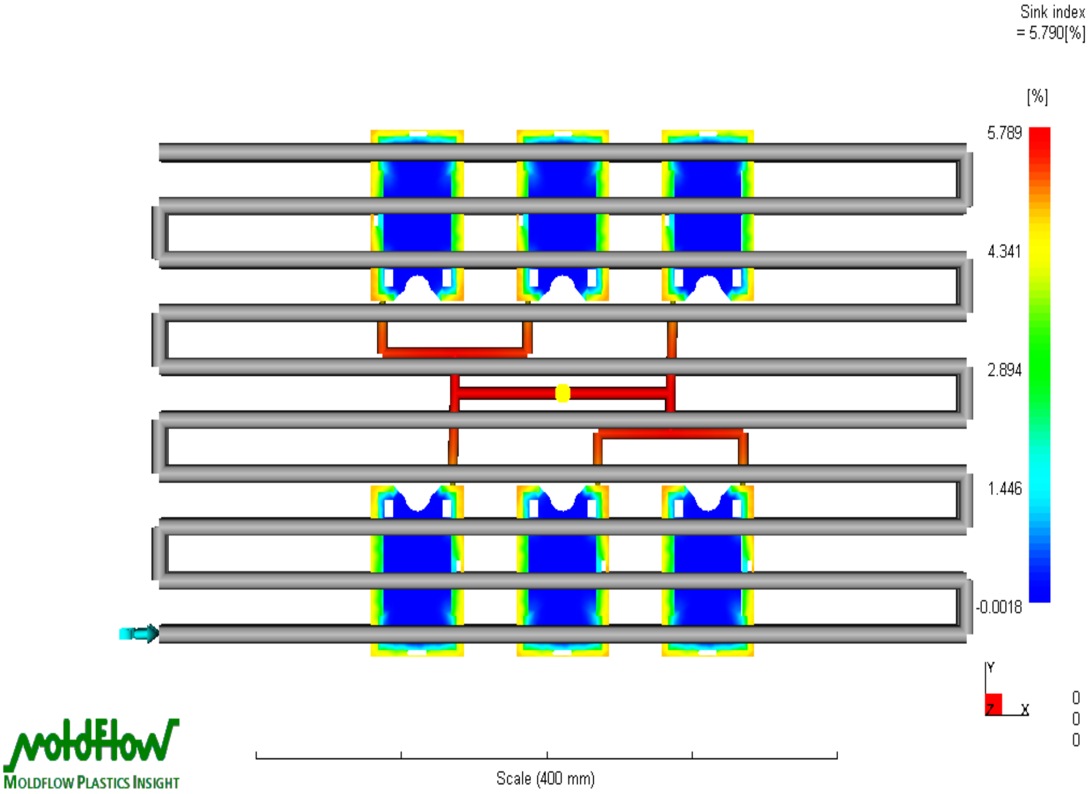


Figure B1 (17): Test no. 18

APPENDIX B2

MOLDFLOW ANALYSIS RESULTS FOR POLYETHYLENE (PE) MATERIAL

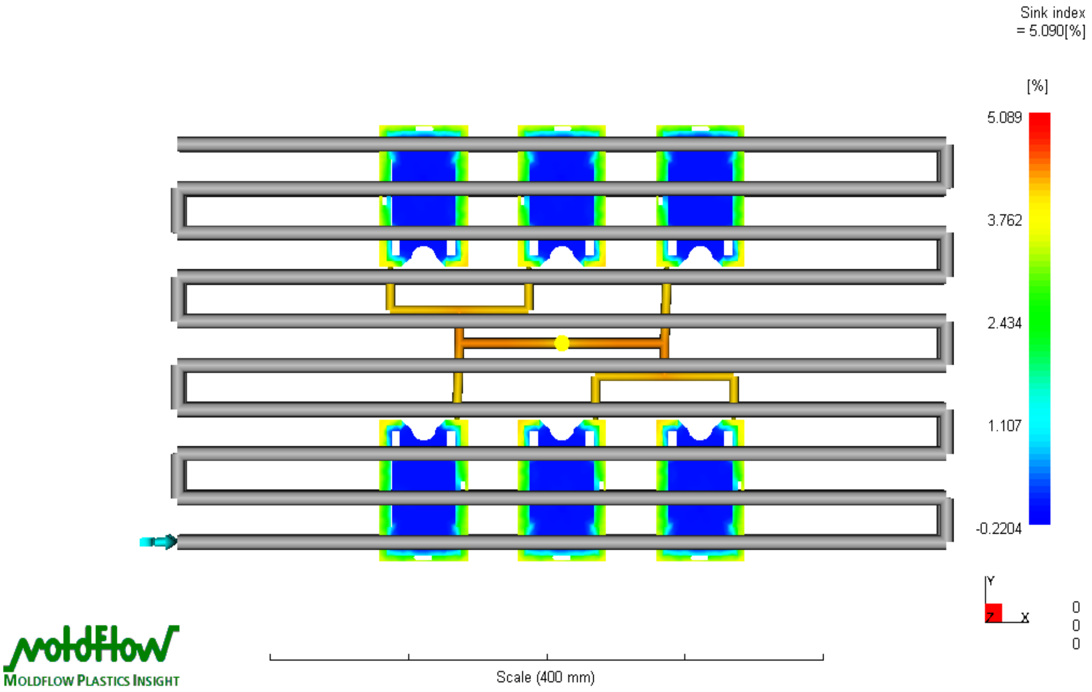


Figure B2 (1): Test no. 2

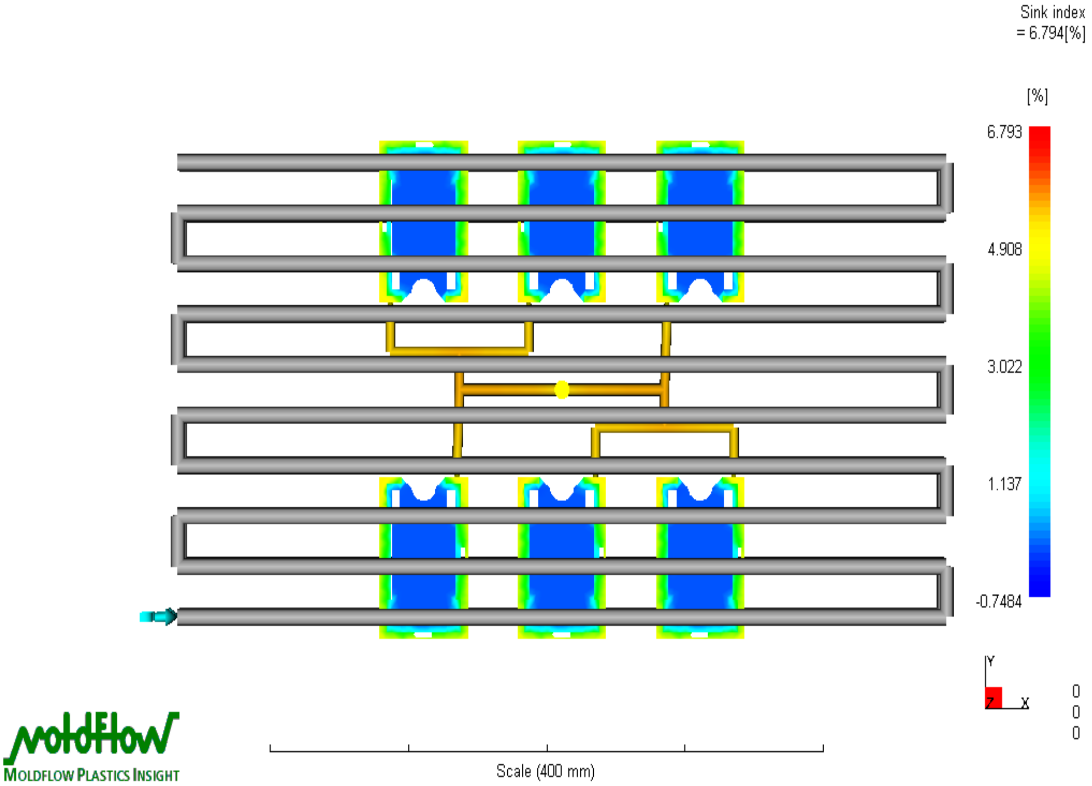


Figure B2 (2): Test no. 3

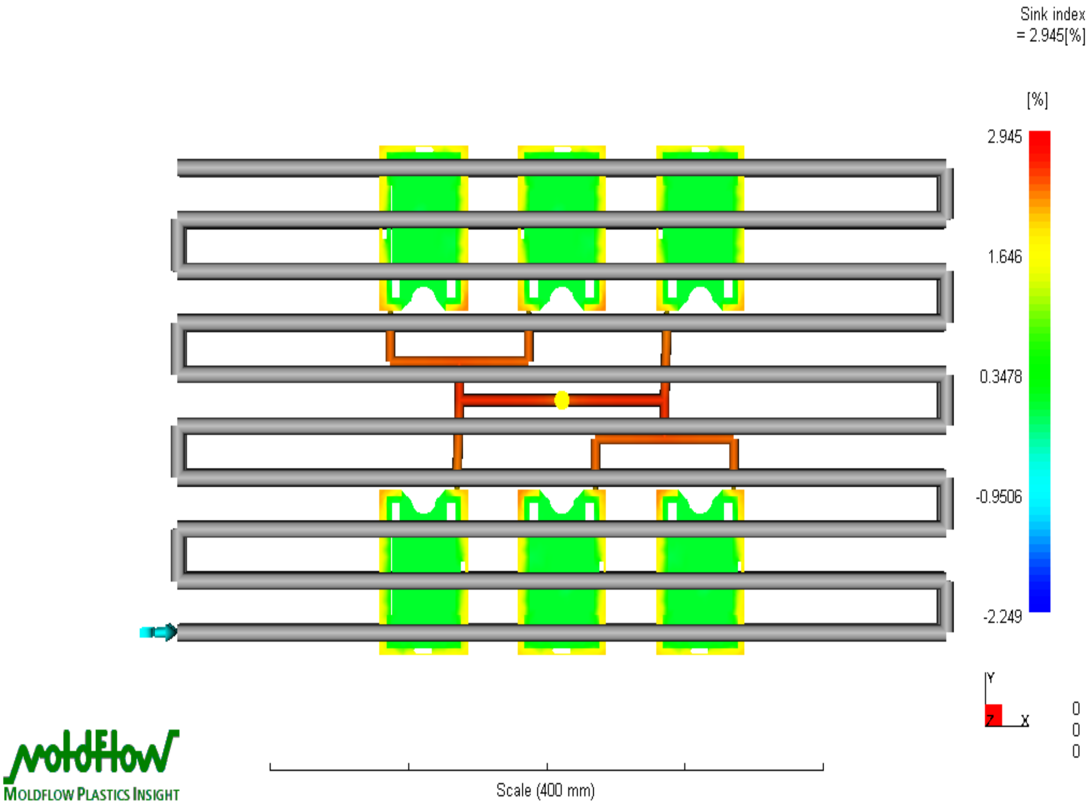


Figure B2 (3): Test no. 4

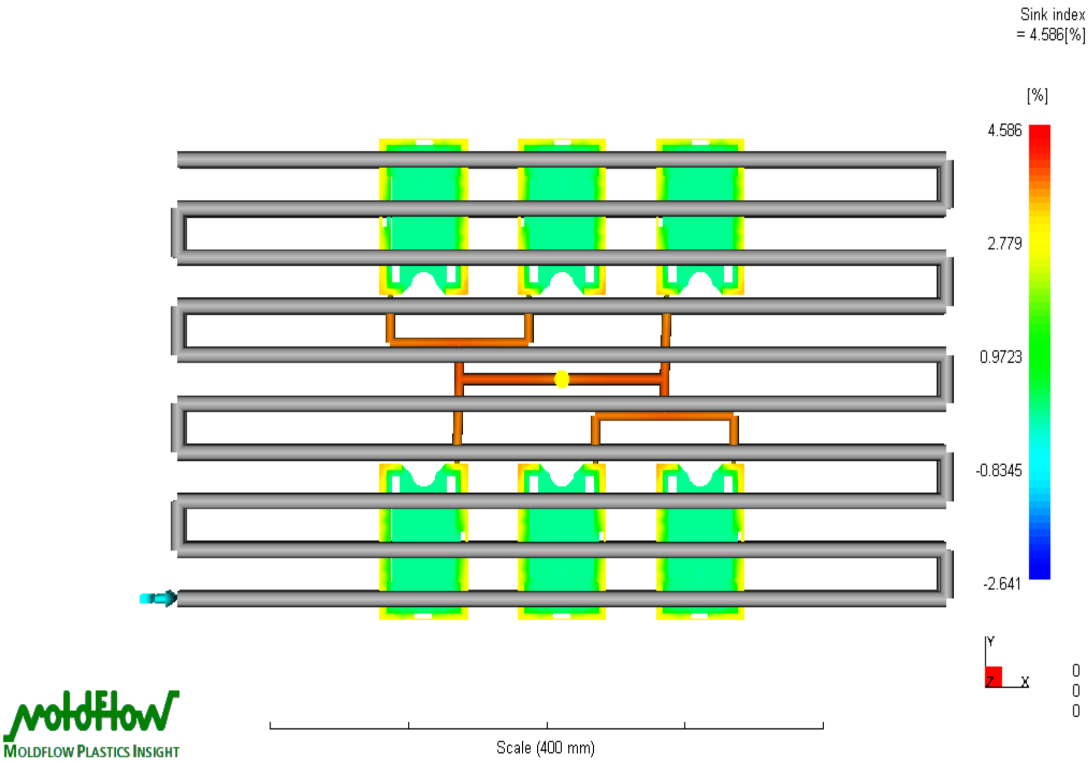


Figure B2 (4): Test no. 5

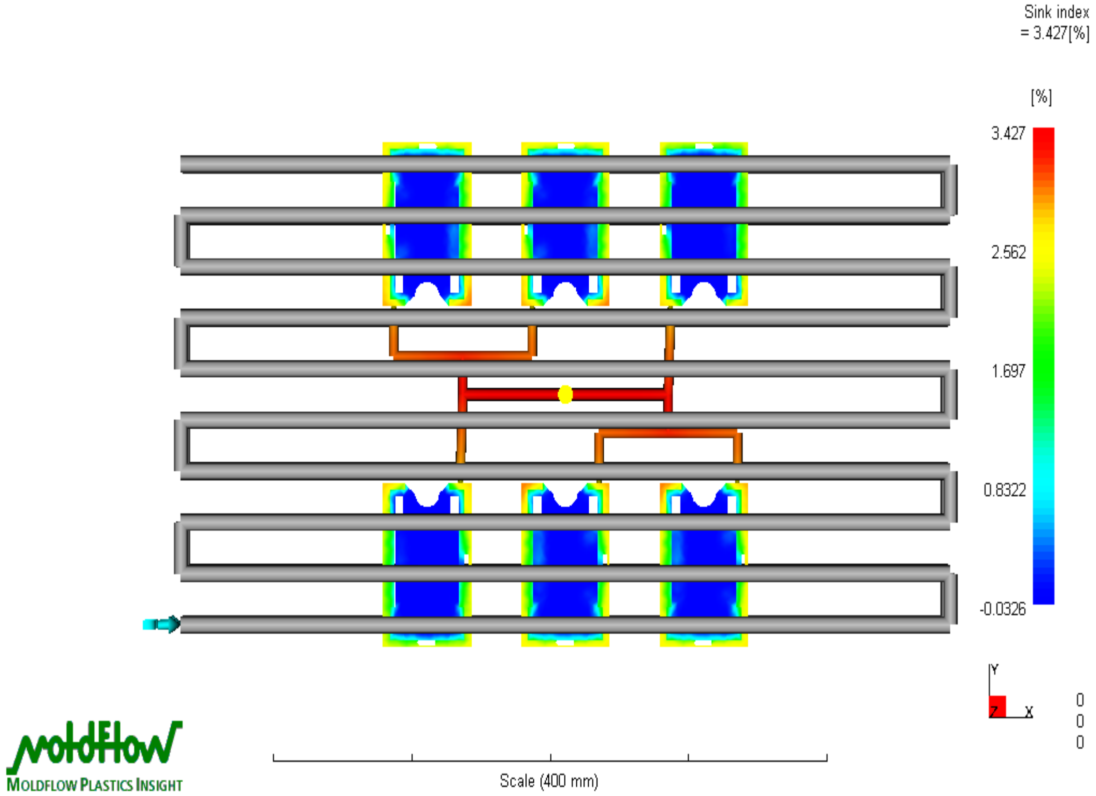


Figure B2 (5): Test no. 6

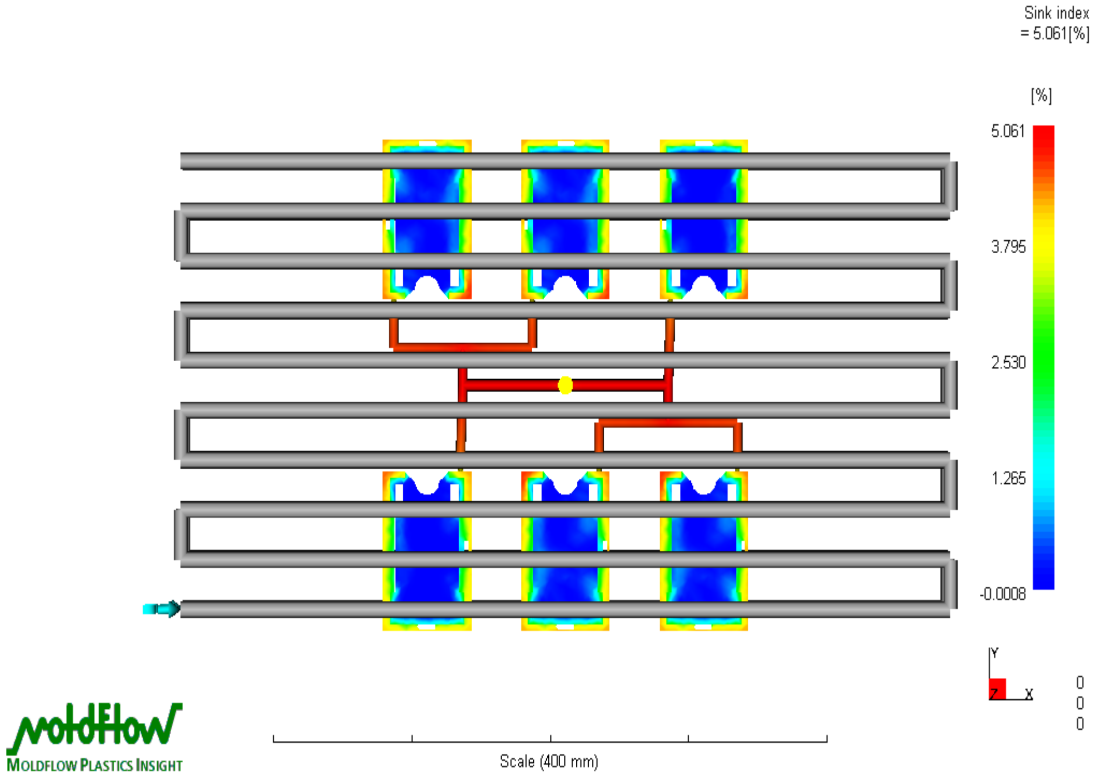


Figure B2 (6): Test no. 7

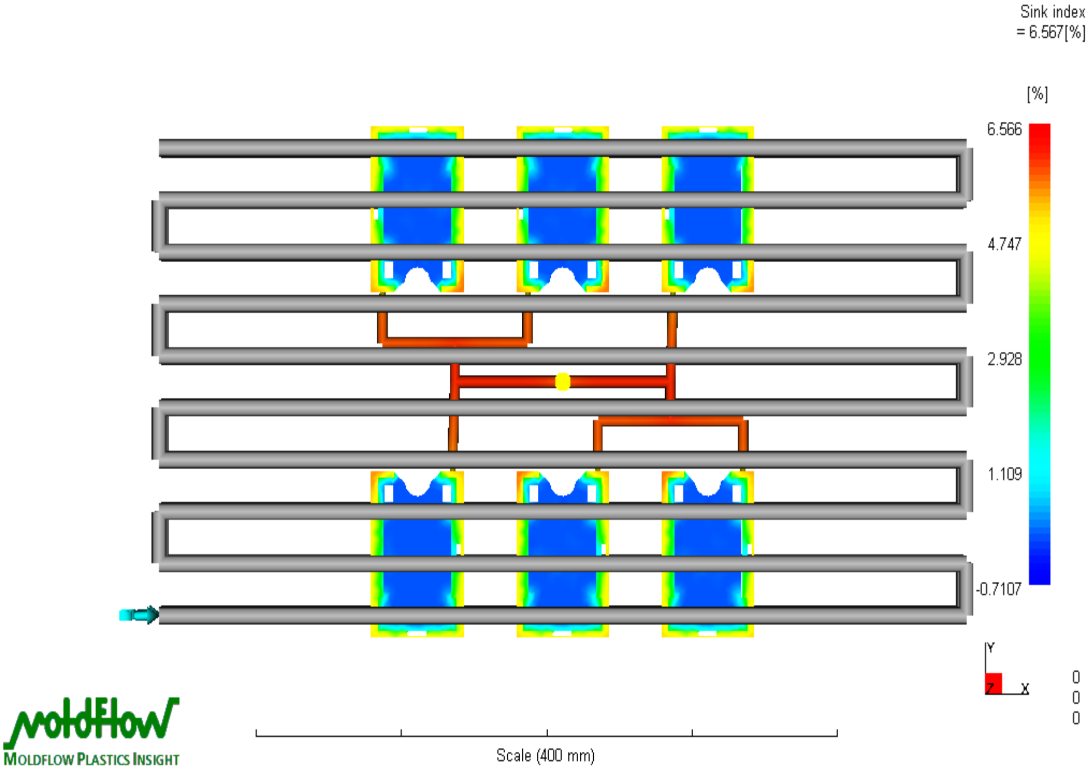


Figure B2 (7): Test no. 8

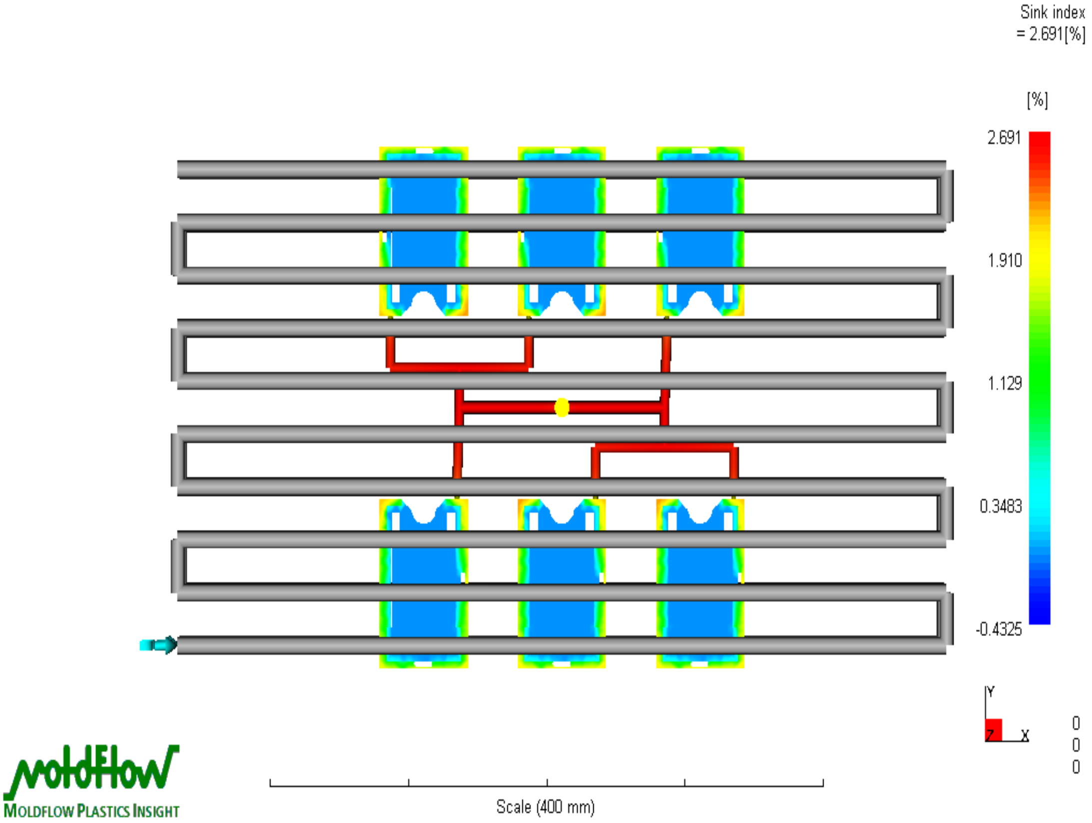


Figure B2 (8): Test no. 9

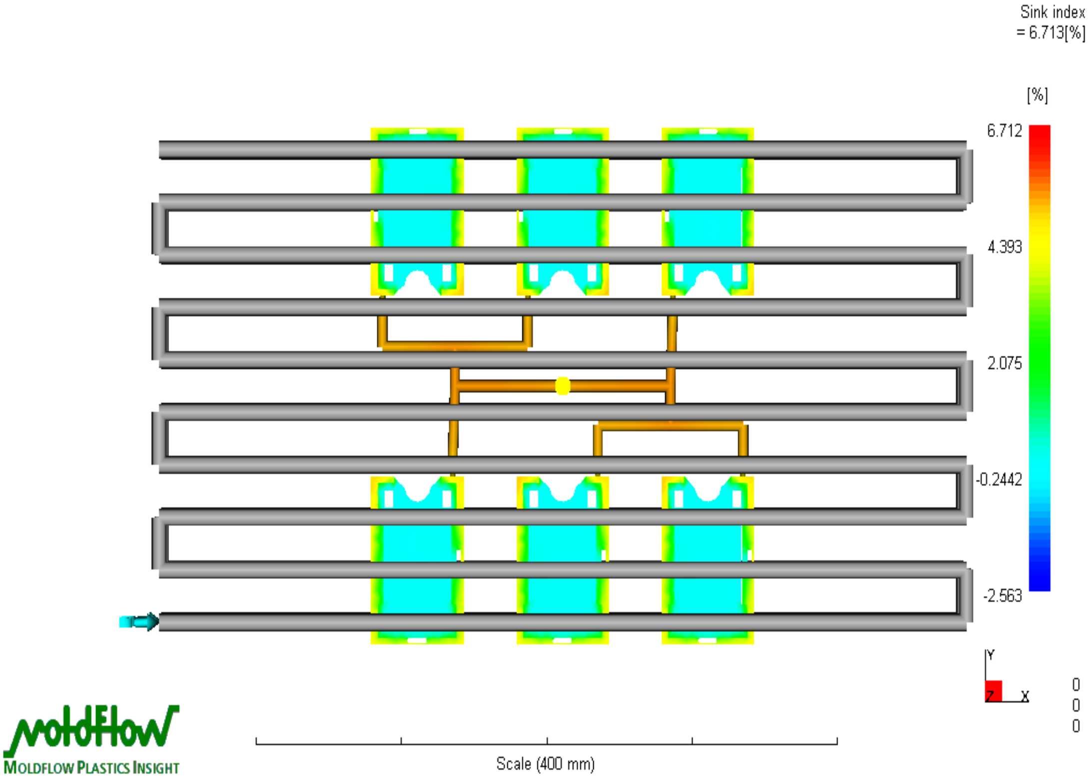


Figure B2 (9): Test no. 10

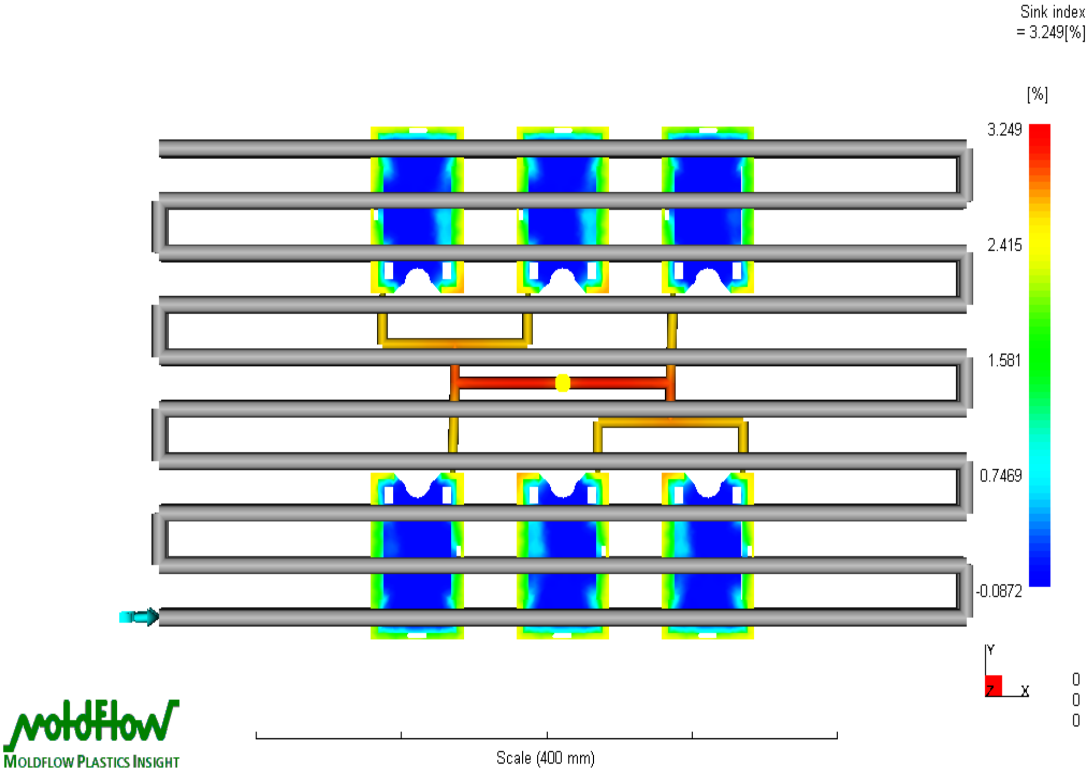


Figure B2 (10): Test no. 11

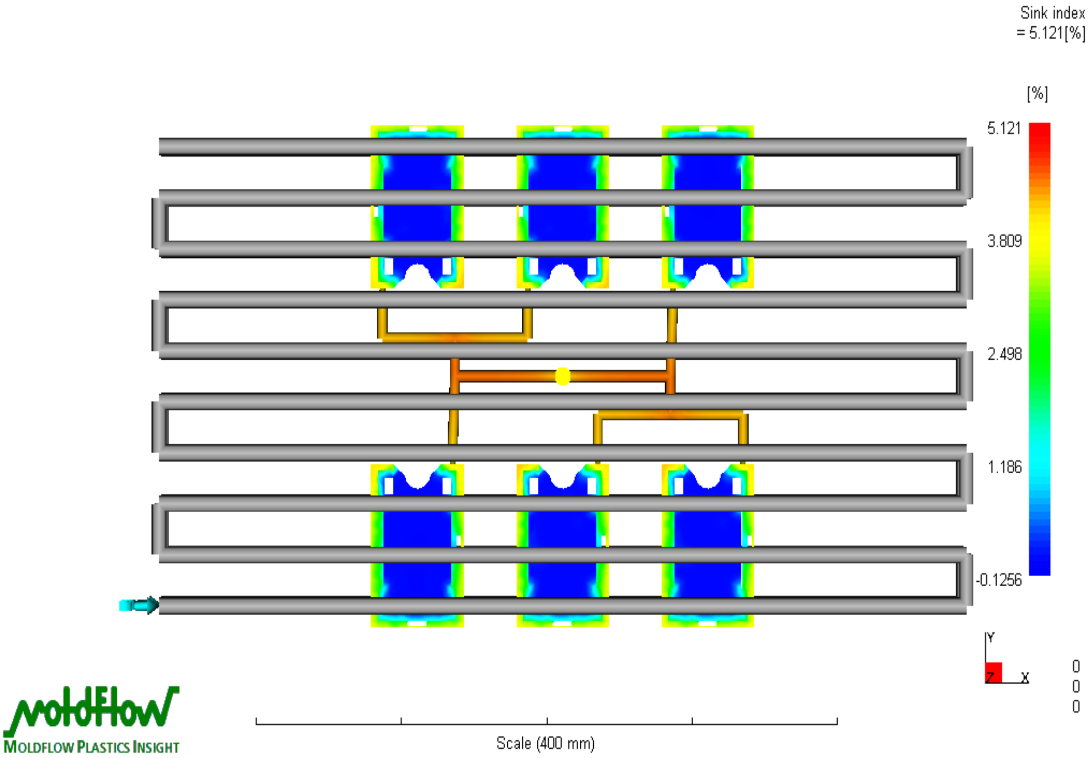


Figure B2 (11): Test no. 12

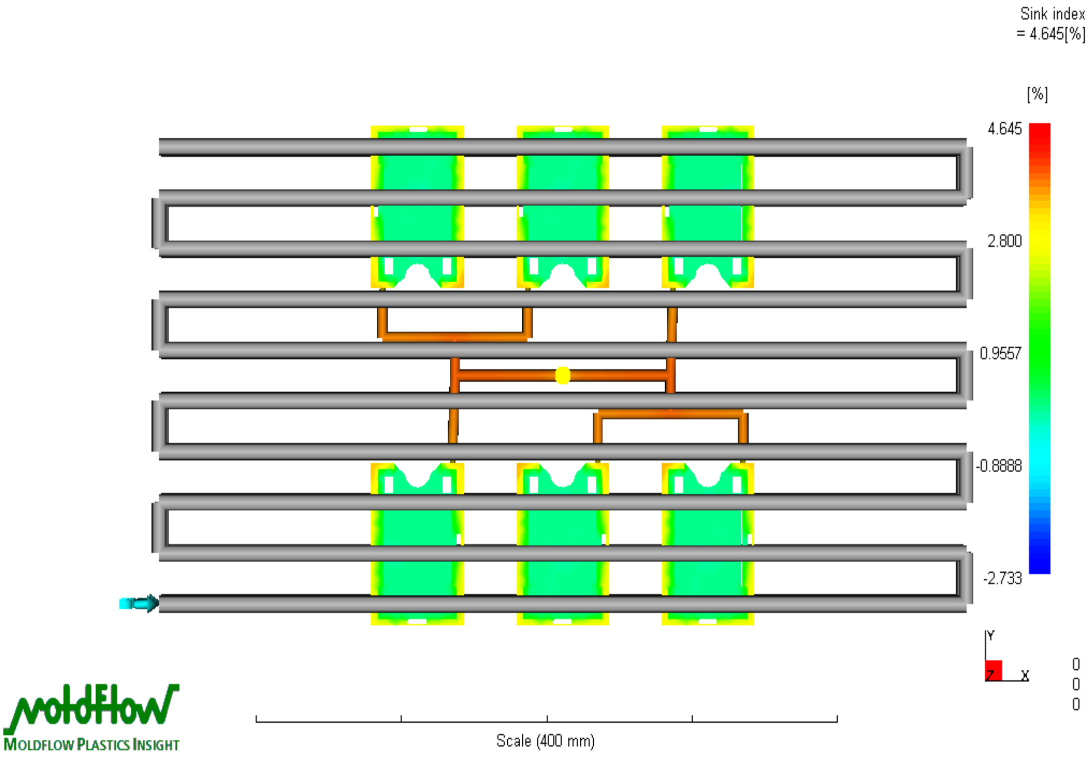


Figure B2 (12): Test no. 13

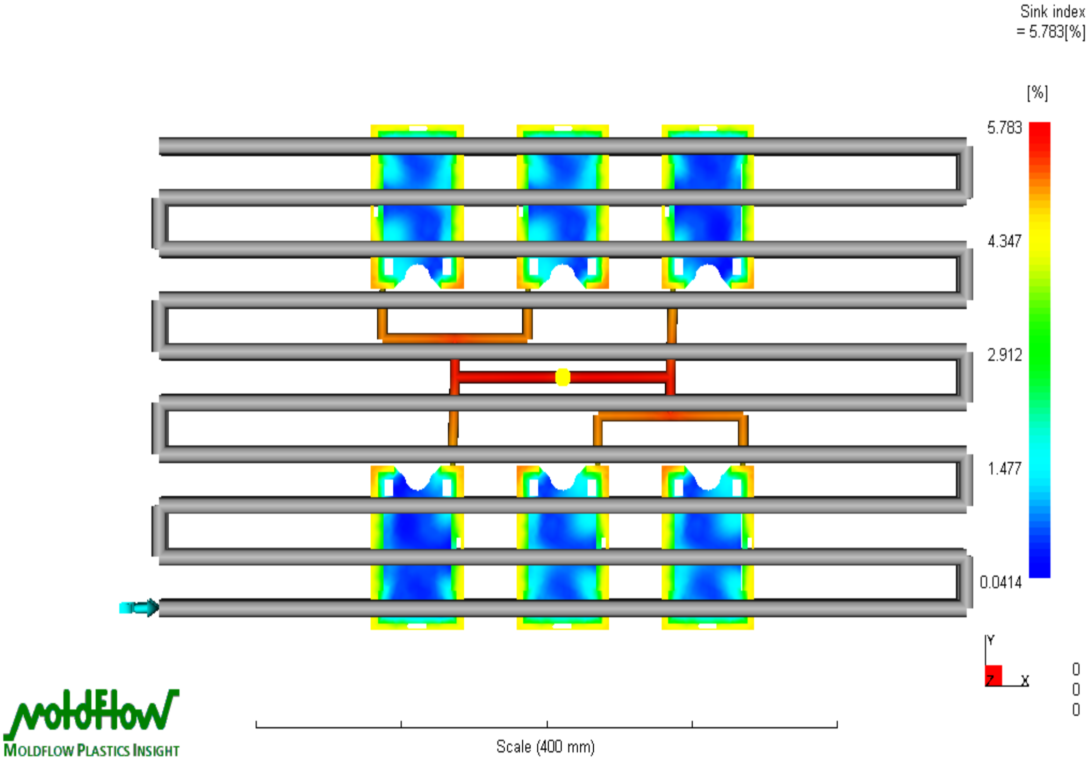


Figure B2 (13): Test no. 14

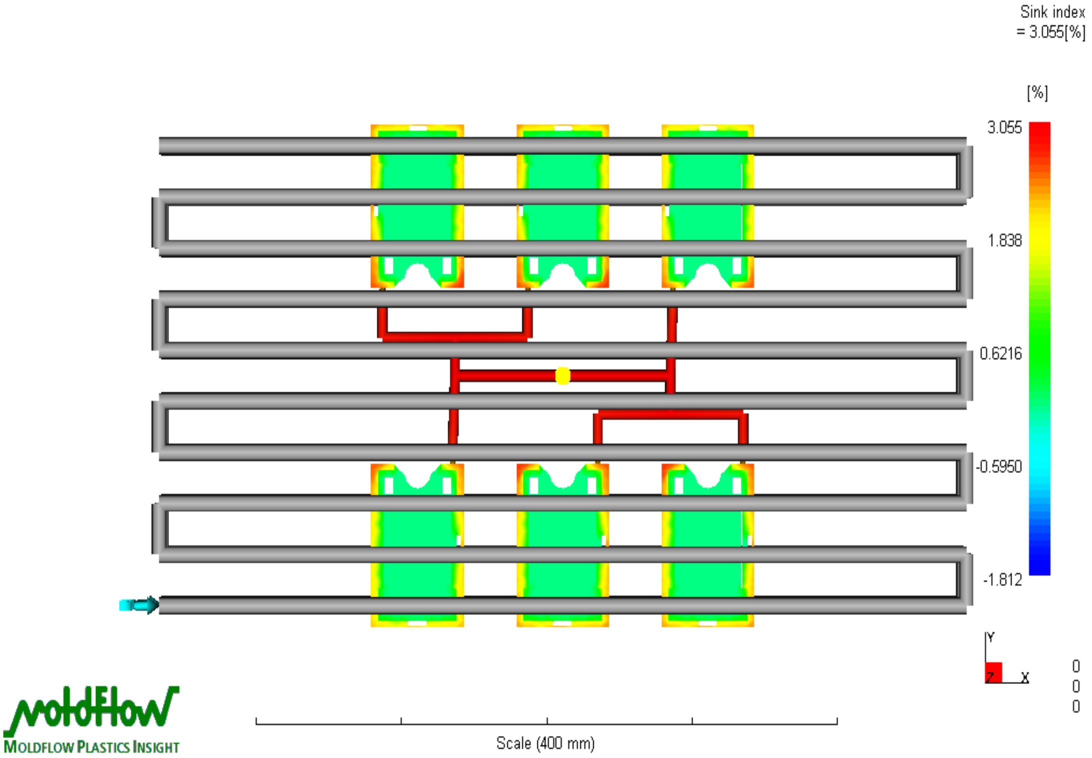


Figure B2 (14): Test no. 15

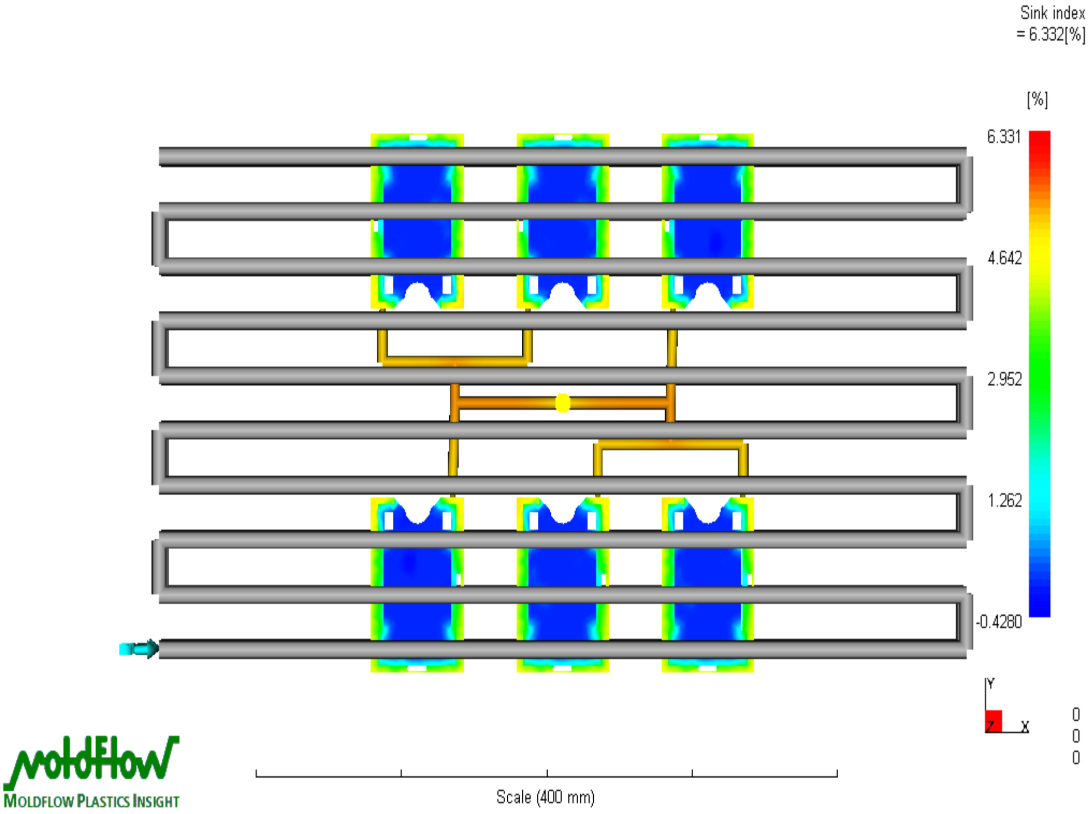


Figure B2 (15): Test no. 16

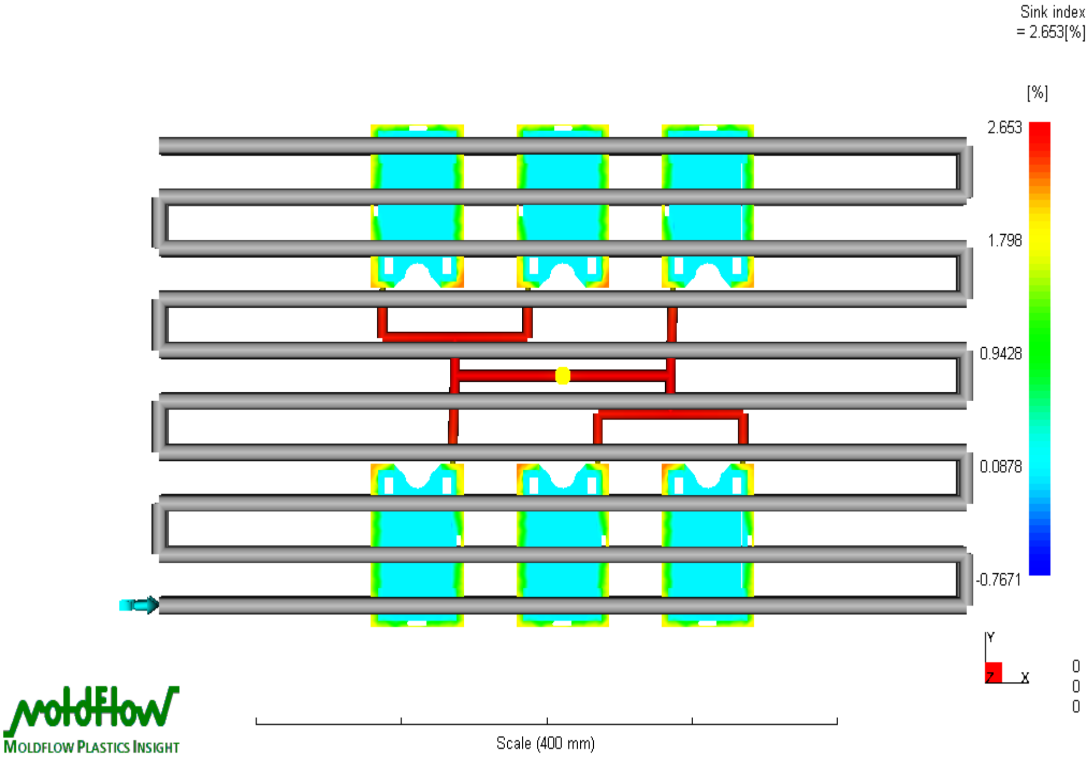


Figure B2 (16): Test no. 17

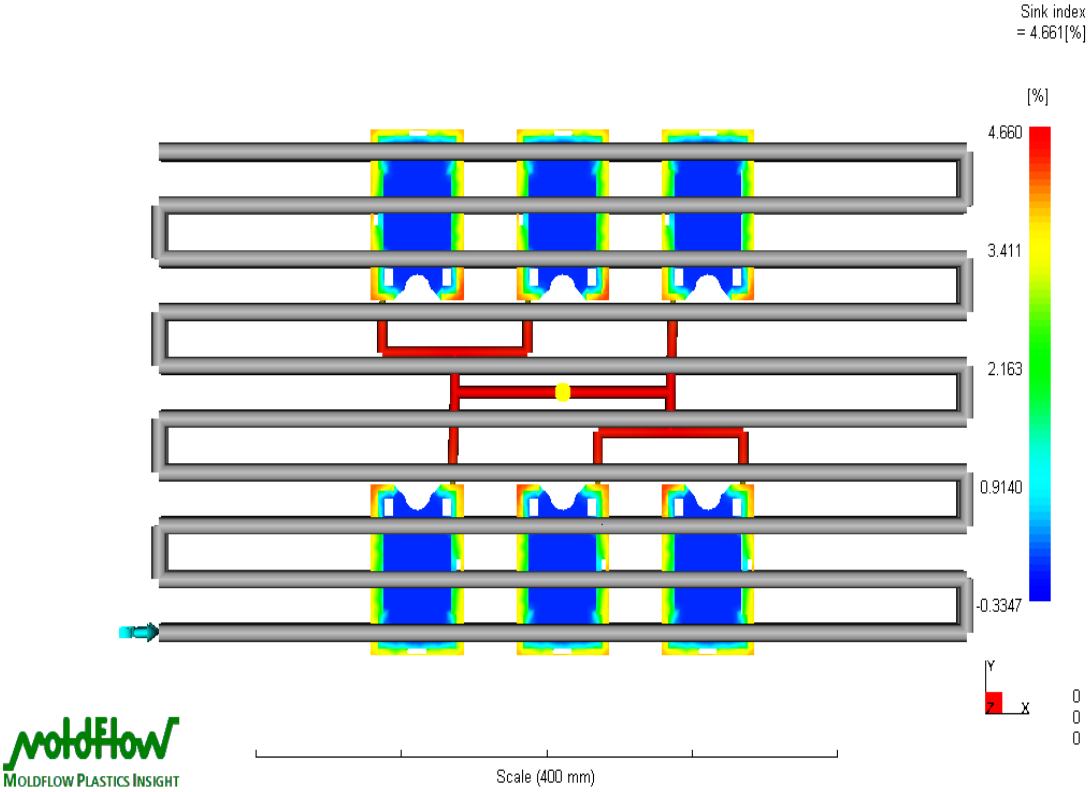


Figure B2 (17): Test no.18