iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 7 Issue: I Month of publication: January 2019

DOI: http://doi.org/10.22214/ijraset.2019.1143

Analysis and Optimization of Gate Location in Thin Walled Injection Molded Part

Mr. S.T. Saravanan[1], Mr. G. Arangadurai[2], Mr. K. Sivamurugan[3]
[1, 2]*Department Of Plastic Technology, 3department Of Mechanical Engineering, Virudhunagar S. Vellaichamy Nadar Polytechnic College, Virudhunagar, India*

Abstract: During the last few decades, considerable developments have been achieved in the area of computer-aided polymer processing. Injection molding is one among the most important processes which is benefited from the computer aided engineering. Even though injection molding is simple in principle the real process is quite complex. Simulation and optimization algorithms can be development tools that make the difference between success and failure. The processors are more interested in manufacturing good quality products with lower cycle time. It is difficult to compensate both two factors since the complexibility of multiple process parameters. Location of gates in the mould plays a vital role in processing of injection moulding machine. With different gate location the quality criteria may change and also the aesthetic appearance of the product affected. In our research the closely same designed gates are designed for given part and optimized mould design before mould making. The gate location optimization is done by the simulation software itself to focus on lower quality prediction. we have modeled a simple injection molded component in Proengineer. The same model is analyzed for two different gate location as centre gate and edge gate using plastic advisor, simulation software from Proengineer. The process ability of each gate is analyzed against the process variables like fill time, injection pressure profile, and cooling quality, for optimized quality predictions and optimized cycle time.

***Keywords**: gate location, proengineer, Centre gate and edge gate*

I. INTRODUCTION

An injection molding process has been used popularly in plastic processing technologies. It has high productive, automatic process and highly précised for repeated production and making complex geometry.

The production cycle of the injection moulding can be divided into mould clamping, melt filling, packing, cooling, mould opening and product ejecting. The mould is the heart of the injection moulding process which can be made with respect to the product drawing. Reducing quality problems in moulding process are the major area to be study. Depending upon the product design, a mould design is done with all geometrical details of product.

Once mould is made it is difficult to modify. The location of the gate position in a mould is an important one to predict quality problems.

To overcome this, mold flow simulation software is used to find out the best suitable gate type and its location. This problem was generally solved using numerical techniques and realistic rheological models for quantitative analysis. But this needed a complicated algorithm and calculation which is difficult to analyze and it takes more time to solve the problems. In the last decade, a considerable progress has been made in incorporating computer software that is beginning to be of tremendous help to the injection molding engineers.

A. Importance Of Gate And Its Location

A gate is a small orifice in which through the polymer melt can entered into it to the cavity. Gate design depending on its type, dimensions and location.

The gate design is largely determined by geometry of the part, Material used, cycle time etc.Unless it is necessary to use multiple gates , a single gate is generally preferred. Multiple gates always create problems of weld lines. The cross-section of the gate is typically smaller than that of the runner.

The gate location should be selected where uniform mould filling is possible. Also to reduce weld lines. The gate should be positioned away from load-bearing areas. This is because the high melt pressure and high velocity of flowing material at a gate causes the area near a gate to be highly stressed.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 7 Issue I, Jan 2019- Available at www.ijraset.com

II. EXPERIMENTAL PART

A. Product Selection
The product has been selected as rectangular box. The model has been created in the pro engineer and this can be move for analysis

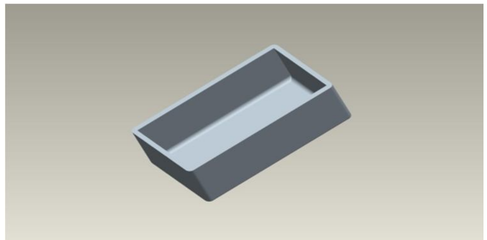


FIG-1

B. Material Selection
The ABS (Acrylonitrile Butadiene Styrene) plastics material is selected for analysis. It is easy to moldable and easy to accessible. The following fig 2 shows the selection of material in the plastic advisor software. The selected material has to be checked with the properties of the given material data sheet.

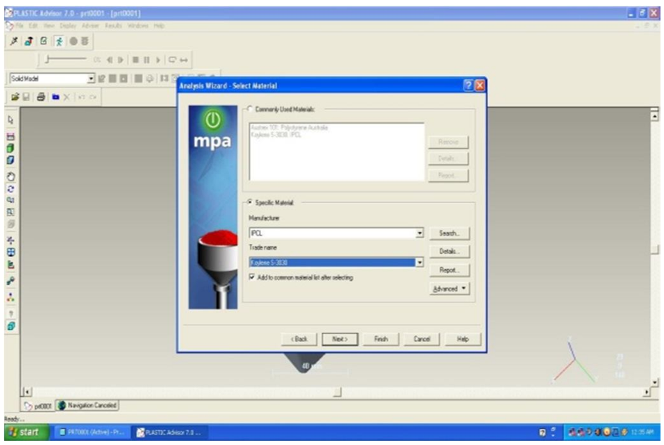


FIG-2

C. Selection Of Gate Location
For one model a centre gate position is selected for analysis. Another model edge gate has been selected to analysis. Both the gate shows uniform filling pattern and can be easily moldable.

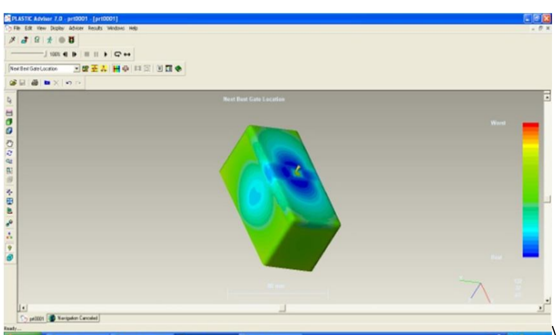
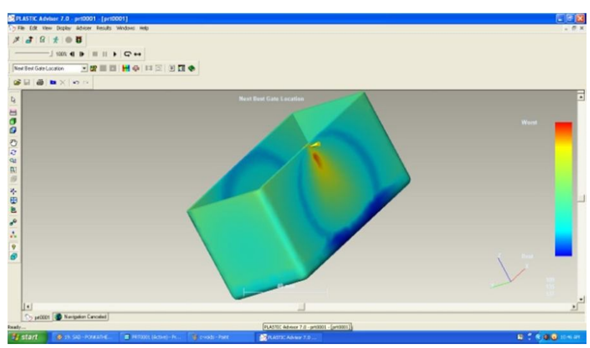
CENTRE GATE LOCATION	EDGE GATE LOCATION
	

FIG-3

D. Fill Time Analysis
The Fill time result shows how the position of the flow front moves as the cavity fills.the flow pattern is should be balanced in a part with a good fill time result that all flow paths finish at the same time and reach the edges of the model simultaneously. Also it is important that the contours are evenly spaced and indicate the speed at which the polymer is flowing.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 7 Issue I, Jan 2019- Available at www.ijraset.com

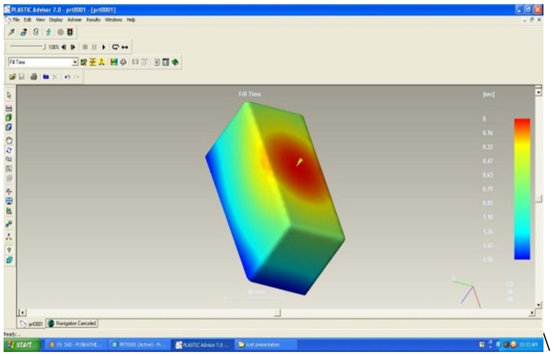
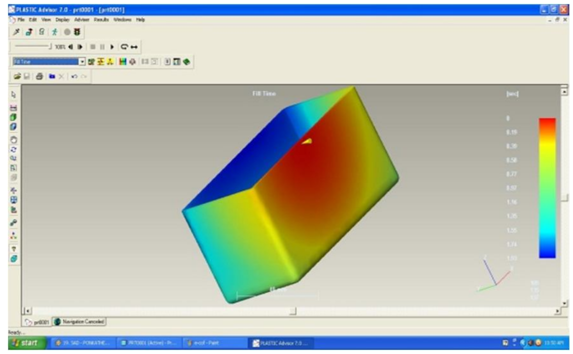
Centre gate location	Edge gate location
	

FIG-4

E. Injection Pressure

Injection Pressure is the pressure applied on the injection screw when a material is being injected into the mold. It enables test fluids to be injected into the cavity with high degree of polymer melt. Normally the maximum injection pressure at the nozzle is about 140 MPa (20,000 psi). We recommend having a maximum pressure of 100 MPa (14,500 psi) for the mold (part and feed system) and 70 MPa (10,000 psi) maximum for the part. If the pressure capacity of the molding machine is known, use no more than about 75% of the pressure capacity for a design guide for the entire mold, and 50% for just the part.

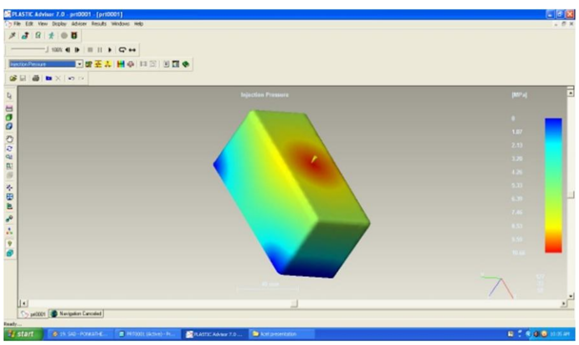
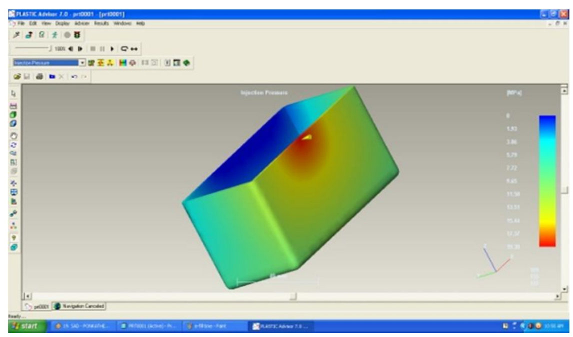
CENTRE GATE LOCATION	EDGE GATE LOCATION
	

FIG-5

F. Voids

Voids are a somewhat common defective occurrence in injection molding applications. They present as “air pockets” in transparent molding but can also be present in colored or opaque molding applications.

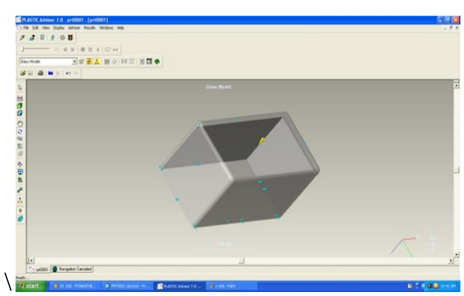
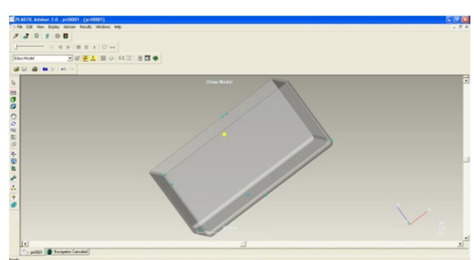
CENTRE GATE LOCATION	EDGE GATE LOCATION
	

FIG-6

Voids tend to appear on thicker part surfaces and can occur when the outside layer of the part cools off and solidifies faster than the internal which can in turn cause a void. The voids result shows how severe air traps will be and where they are likely to occur on the part. Air traps may be acceptable if they occur on a surface that does not have to be visually perfect .The Fill time result is used in conjunction with the Air traps result to confirm the filling behavior and assess the likelihood of air traps appearing.

G. Cooling Quality

Cooling quality shows the percentage of cooling in the part. Both the gates shows perfect cooling . but when compared to edge gate the centre gate shows corners to be cooled later.

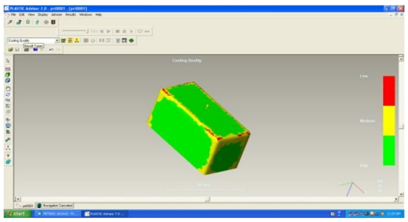
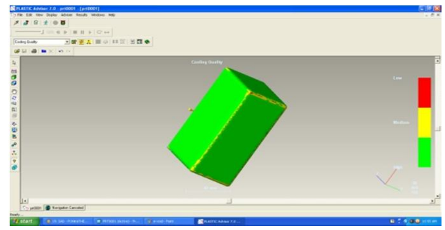
CENTRE GATE LOCATION	EDGE GATE LOCATION
	

FIG-7

H. Sink Mark

Sink marks in injection molded plastic parts develop when material in the region of thick features such as ribs or bosses shrinks more than the material in the adjacent wall. This type of defect presents a major limitation to designing and molding plastic products, particularly in thin-wall applications.

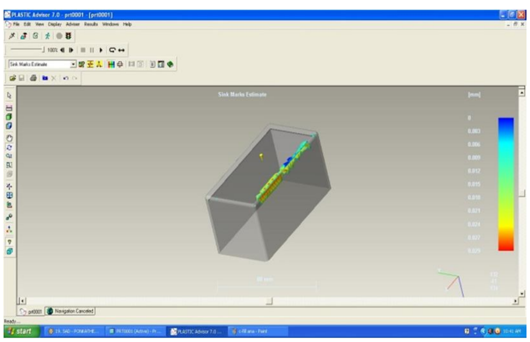
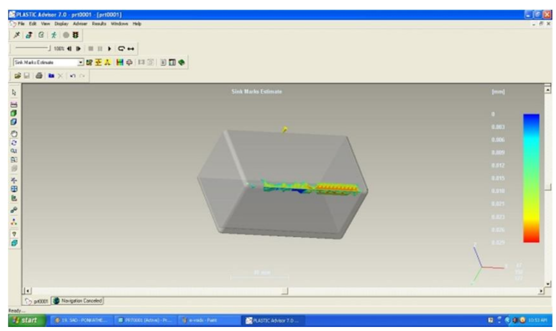
CENTRE GATE LOCATION	EDGE GATE LOCATION
	

FIG-8

I. Skin Orientation

The Orientation at skin result provides a good indication of how molecules will be oriented on the outside of the part, showing the average principal alignment direction for the whole local area at the end of filling.Because the melt freezes very quickly when it contacts the mold for the first time, the velocity vector provides the most probable molecular orientation at the skin.

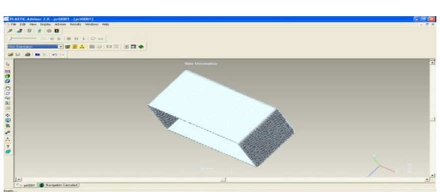
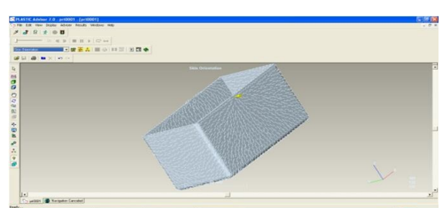
CENTRE GATE LOCATION	EDGE GATE LOCATION
	

FIG-9

III. RESULTS AND DISCUSSION

The results are shown in table-1.

PARAMETER	CENTER GATE	EDGE GATE
GATE LOCATION	GOOD POSITION	MEDIUM POSITION
FILL TIME	1.58 SEC	1.93 SEC
INJECTION PR	10.66 MPa	19.3MPa
PRESSURE DROP	MEDIUM	MORE
SKIN	UNIFORM	NON UNIFORM
CONF.OF FILL	HIGH	HIGH
VOIDS	MORE	COMPARATELY LESS
SINK MARK	LESS	MORE ON EDGE
COOLING QUALITY	GOOD	LESS AT BOTTOM

Technically speaking, the material freezing off at the gate indicates the end of the cavity packing phase. Hence there is no point in maintaining pressure after this point is reached.This in turn means the gate must be large enough to make sure that the moulding is properly filled before gate freezing occurs. A larger gate dimension will reduce viscous (frictional) heating, permit lower velocities. So the position of gate and allow the application of high packing pressure to increase the density of the material
in the cavity. If low stress is a requirement, owing to concerns of aesthetic appearance or dimensional stability, a larger gate may be necessary. The injection pressure is more in the edge gate but low in centre gate. When comparing the cycle time of the part the total fill time is 1.93 secs in edge gate whereas 1.53 secs in centre gate. Voids and sink marks are more in edge gate but these are less in centre gate. So by analyzing these factors centre gate is selected and best suitable for box type of product.

REFERENCES

[1] Single gate optimization for plastic injection mold by CuiZhendong WangXicheng LiZhijie Dalian Univ. of Technol., Dalian This paper appears in: Network and Parallel Computing Workshops, 2007. NPC Workshops. IFIP InternationalConferenceon,issueDate:18-21Sept.2007Onpage(s):449-454Location:Liaoning Print ISBN: 978-0-7695-2943-1

[2] Optimization of Gate, Runner and Sprue in Two-Plate Family Plastic Injection Mould By LIJi-quan (Department of Plasticity Technology, Shanghai Jiao Tong University,Shanghai200030,China)E-mail:hutli@163.com
Received Nov. 22, 2006 revision accepted Mar. 19, 2007

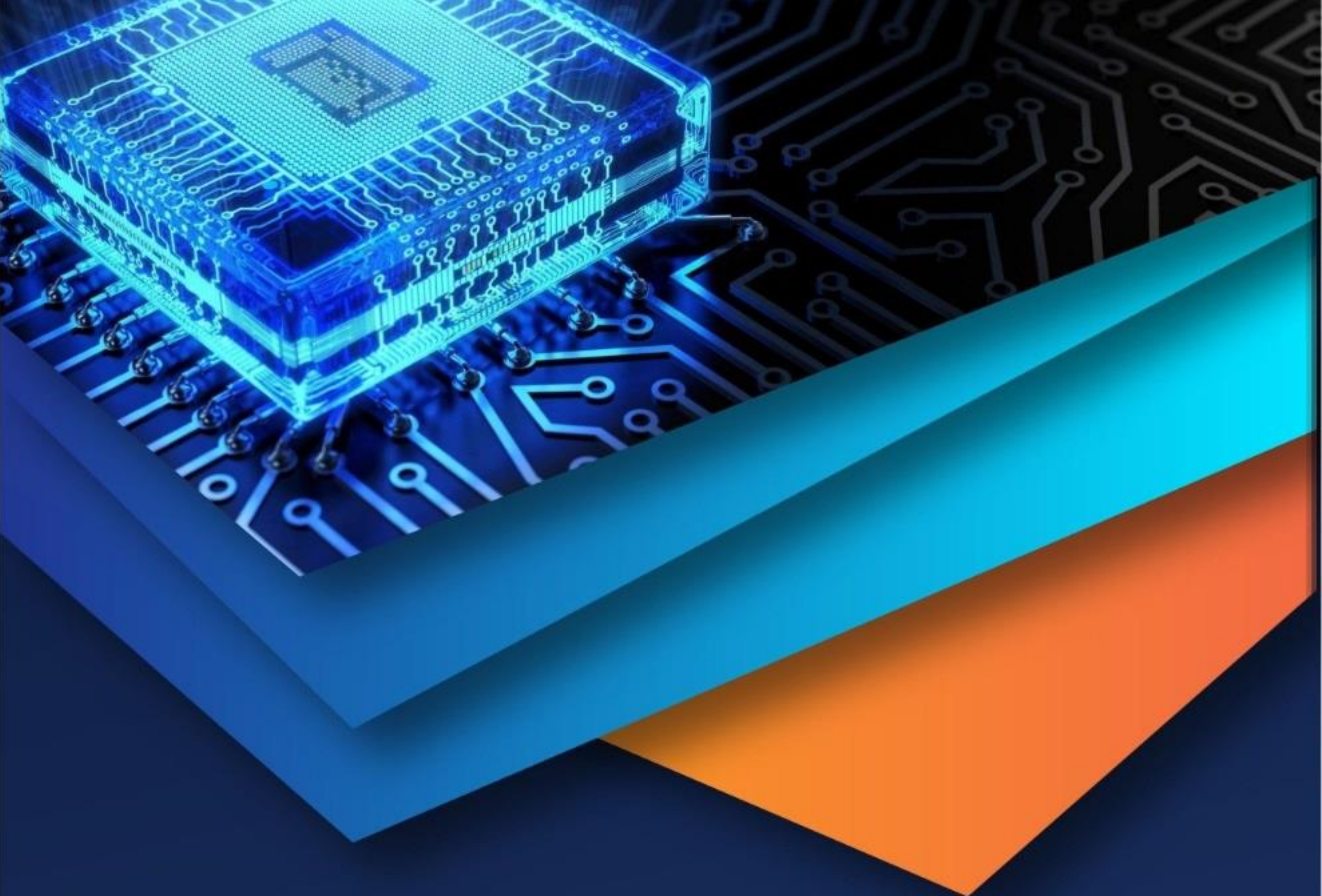
[3] Hand book of plastic materials and technology By Irvin.I.Rubin,wiley interscience

[4] Injection mould design by R.G.W pye published East West Press Ltd

[5] Modeling and simulation of weld line in injection moulding process by khaled mezghani released the 6th Saudi engineering conference,dec 2002

[6] Delrin moulding manual by Dupont plastics.

[7] Plastics processing hand bookBy dominic rosato,Chapman& hall



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY